
STAAD.Pro

V8i (SELECTseries 4)

Technical Reference Manual



DAA037780-1/0005
Last updated: 19 November 2012

Copyright Information

Trademark Notice

Bentley, the "B" Bentley logo, STAAD.Pro are registered or nonregistered trademarks of Bentley Systems, Inc. or Bentley Software, Inc. All other marks are the property of their respective owners.

Copyright Notice

© 2012, Bentley Systems, Incorporated. All Rights Reserved.

Including software, file formats, and audiovisual displays; may only be used pursuant to applicable software license agreement; contains confidential and proprietary information of Bentley Systems, Incorporated and/or third parties which is protected by copyright and trade secret law and may not be provided or otherwise made available without proper authorization.

Acknowledgments

Windows, Vista, SQL Server, MSDE, .NET, DirectX are registered trademarks of Microsoft Corporation.

Adobe, the Adobe logo, Acrobat, the Acrobat logo are registered trademarks of Adobe Systems Incorporated.

Restricted Rights Legends

If this software is acquired for or on behalf of the United States of America, its agencies and/or instrumentalities ("U.S. Government"), it is provided with restricted rights. This software and accompanying documentation are "commercial computer software" and "commercial computer software documentation," respectively, pursuant to 48 C.F.R. 12.212 and 227.7202, and "restricted computer software" pursuant to 48 C.F.R. 52.227-19(a), as applicable. Use, modification, reproduction, release, performance, display or disclosure of this software and accompanying documentation by the U.S. Government are subject to restrictions as set forth in this Agreement and pursuant to 48 C.F.R. 12.212, 52.227-19, 227.7202, and 1852.227-86, as applicable.

Contractor/Manufacturer is Bentley Systems, Incorporated, 685 Stockton Drive, Exton, PA 19341- 0678.

Unpublished - rights reserved under the Copyright Laws of the United States and International treaties.

End User License Agreements

Table of Contents

To view the End User License Agreement for this product, review: [eula_en.pdf](#).

Table of Contents

About this Manual	1
Document Conventions	2
Section 1 General Description	5
1.1 Introduction	6
1.2 Input Generation	7
1.3 Types of Structures	7
1.4 Unit Systems	9
1.5 Structure Geometry and Coordinate Systems	9
1.6 Finite Element Information	21
1.7 Member Properties	37
1.8 Member/ Element Release	46
1.9 Truss and Tension- or Compression-Only Members	47
1.10 Tension, Compression - Only Springs	47
1.11 Cable Members	47
1.12 Member Offsets	50
1.13 Material Constants	52
1.14 Supports	52
1.15 Master/Slave Joints	53
1.16 Loads	53
1.17 Load Generator	60
1.18 Analysis Facilities	63
1.19 Member End Forces	87
1.20 Multiple Analyses	93
1.21 Steel, Concrete, and Timber Design	94
1.22 Footing Design	94
1.23 Printing Facilities	94
1.24 Plotting Facilities	94
1.25 Miscellaneous Facilities	94

1.26 Post Processing Facilities	95
Section 2 American Steel Design	97
2.1 Design Operations	97
2.2 Member Properties	98
2.3 Steel Design per AISC 360 Unified Specification	102
2.4 Steel Design per AISC 9th Edition	112
2.5 Steel Design per AASHTO Specifications	175
2.6 Design per American Cold Formed Steel Code	209
Section 3 American Concrete Design	217
3.1 Design Operations	217
3.2 Section Types for Concrete Design	218
3.3 Member Dimensions	219
3.4 Design Parameters	220
3.5 Slenderness Effects and Analysis Consideration	224
3.6 Beam Design	225
3.7 Column Design	231
3.8 Designing elements, shear walls, slabs	236
Section 4 American Timber Design	247
4.1 Design Operations	247
4.2 Allowable Stress per AITC Code	250
4.3 Input Specification	252
4.4 Naming Conventions for Sections	254
4.5 Design Parameters	257
4.6 Member Design Capabilities	262
4.7 Orientation of Lamination	263
4.8 Tabulated Results of Member Design	263
4.9 Examples	265
Section 5 Commands and Input Instructions	271
5.1 Command Language Conventions	275
5.2 Problem Initiation and Model Title	280

5.3 Unit Specification	282
5.4 Input/Output Width Specification	284
5.5 Set Command Specification	284
5.6 Data Separator	294
5.7 Page New	295
5.8 Page Length/Eject	295
5.9 Ignore Specifications	296
5.10 No Design Specification	296
5.11 Joint Coordinates Specification	296
5.12 Member Incidences Specification	300
5.13 Elements and Surfaces	303
5.14 Plate Element Mesh Generation	309
5.15 Redefinition of Joint and Member Numbers	317
5.16 Entities as Single Objects	318
5.17 Rotation of Structure Geometry	324
5.18 Inactive/Delete Specification	324
5.19 User Steel Table Specification	326
5.20 Member Property Specification	339
5.21 Element/Surface Property Specification	368
5.22 Member/Element Releases	370
5.23 Axial Member Specifications	375
5.24 Element Plane Stress and Ignore Inplane Rotation Specification	382
5.25 Member Offset Specification	383
5.26 Specifying and Assigning Material Constants	385
5.27 Support Specifications	405
5.28 Rigid Diaphragm Modeling	420
5.29 Draw Specifications	427
5.30 Miscellaneous Settings for Dynamic Analysis	427
5.31 Definition of Load Systems	429
5.32 Loading Specifications	539

5.33 Reference Load Cases - Application	669
5.34 Frequency Calculation	670
5.35 Load Combination Specification	672
5.36 Calculation of Problem Statistics	676
5.37 Analysis Specification	676
5.38 Change Specification	709
5.39 Load List Specification	711
5.40 Load Envelope	712
5.41 Section Specification	713
5.42 Print Specifications	714
5.43 Stress/Force output printing for Surface Entities	722
5.44 Printing Section Displacements for Members	724
5.45 Printing the Force Envelope	726
5.46 Post Analysis Printer Plot Specifications	727
5.47 Size Specification	727
5.48 Steel and Aluminum Design Specifications	728
5.49 Code Checking Specification	731
5.50 Group Specification	734
5.51 Steel and Aluminum Take Off Specification	736
5.52 Timber Design Specifications	737
5.53 Concrete Design Specifications	739
5.54 Footing Design Specifications	742
5.55 Shear Wall Design	742
5.56 End Run Specification	745
Index of Commands	759
A, B	759
C	759
D	759
E	760
F	760

G	760
H	760
I	760
J, K	760
L	760
M	760
N	761
O	761
P, Q	761
R	761
S	761
T	762
U, V, W, X, Y, Z	762
Technical Support	763

About this Manual

Section 1 of the manual contains a general description of the analysis and design facilities available in the STAAD engine.

Specific information on steel, concrete, and timber design is available in Sections 2, 3, and 4 of this manual, respectively.

Detailed STAAD engine STD file command formats and other specific user information is presented in Section 5.

Document Conventions

The following typographical and mathematical conventions are used throughout this manual. It is recommended you spend some time to familiarize yourself with these as to make comprehension of the content easier.

Notes, Hints, and Warnings

Items of special note are indicated as follows:

Note: This is an item of general importance.

Hint: This is optional time-saving information.

Warning: This is information about actions that should not be performed under normal operating conditions.

File Path/File Name.extension

A fixed width typeface is used to indicate file names, file paths, and file extensions (e.g., C:/SPROV8I/STAAD/STAADPRO.EXE)

Interface Control

A bold typeface is used to indicate user controls. Menu and sub-menu items are indicated with a series of > characters to distinguish menu levels. (e.g., **File > Save As...**).

User Input

A bold, fixed width typeface is used to indicate information which must be manually entered. (e.g., Type **DEAD LOAD** as the title for Load Case 1).

STAAD Page Controls

A " | " character is used to represent the page control levels between pages and sub-pages. (e.g., Select the **Design | Steel** page).

Terminology

- Click - This refers to the action of pressing a mouse button to "click" an on screen interface button. When not specified, click means to press the left mouse button.

- Select - Indicates that the command must be executed from a menu or dialog (synonymous with Click). Used when referring to an action in a menu, drop-down list, list box, or other control where multiple options are available to you.
- pop-up menu - A pop-up menu is displayed typically with a right-click of the mouse on an item in the interface.
- Window - Describes an on screen element which may be manipulated independently. Multiple windows may be open and interacted with simultaneously.
- Dialog - This is an on screen element which (typically) must be interacted with before returning to the main window.
- Cursor - Various selection tools are referred to as "cursors" in STAAD.Pro. Selecting one of these tools will change the mouse pointer icon to reflect the current selection mode.

Mathematical Conventions

Similar to spelling conventions, American mathematical notation is used throughout the documentation.

- Numbers greater than 999 are written using a comma (,) to separate every three digits. For example, the U.S. value of Young's Modulus is taken as 29,000,000 psi.

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

- Numbers with decimal fractions are written with a period to separate whole and fraction parts. For example, a beam with a length of 21.75 feet.
- Multiplication is represented with a raised, or middle, dot (\cdot). For example, $P = F \cdot A$.
- Operation separators are used in the following order:
 1. parenthesis ()
 2. square brackets []
 3. curly brackets (i.e., braces) { }

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

Section 1

General Description

1.1 Introduction	6
1.2 Input Generation	7
1.3 Types of Structures	7
1.4 Unit Systems	9
1.5 Structure Geometry and Coordinate Systems	9
1.6 Finite Element Information	21
1.7 Member Properties	37
1.8 Member/ Element Release	46
1.9 Truss and Tension- or Compression-Only Members	47
1.10 Tension, Compression - Only Springs	47
1.11 Cable Members	47
1.12 Member Offsets	50
1.13 Material Constants	52
1.14 Supports	52

Section 1 General Description

1.1 Introduction

1.15 Master/Slave Joints	53
1.16 Loads	53
1.17 Load Generator	60
1.18 Analysis Facilities	63
1.19 Member End Forces	87
1.20 Multiple Analyses	93
1.21 Steel, Concrete, and Timber Design	94
1.22 Footing Design	94
1.23 Printing Facilities	94
1.24 Plotting Facilities	94
1.25 Miscellaneous Facilities	94
1.26 Post Processing Facilities	95

1.1 Introduction

The STAAD.Pro V8i Graphical User Interface (GUI) is normally used to create all input specifications and all output reports and displays (See the Graphical Environment manual). These structural modeling and analysis input specifications are stored in STAAD input file – a text file with extension, .STD. When the GUI opens an existing model file, it reads all of the information necessary from the STAAD input file. You may edit or create this STAAD input file and then the GUI and the analysis engine will both reflect the changes.

The STAAD input file is processed by the STAAD analysis “engine” to produce results that are stored in several files (with file extensions such as ANL, BMD, TMH, etc.). The STAAD analysis text file (file extension .ANL) contains the printable output as created by the specifications in this manual. The other files contain the results (displacements, member/element forces, mode shapes, section forces/moments/displacements, etc.) that are used by the GUI in the post processing mode.

This section of the manual contains a general description of the analysis and design facilities available in the STAAD engine. Specific information on steel, concrete, and timber design is available in Sections 2, 3, and 4 of this manual, respectively. Detailed STAAD engine STD file command formats and other specific input information is presented in Section 5.

The objective of this section is to familiarize you with the basic principles involved in the implementation of the various analysis/design facilities offered by the STAAD engine. As a general rule, the sequence in which the facilities are discussed follows the recommended sequence of their usage in the STAAD input file.

1.2 Input Generation

The GUI (or you, the user) communicates with the STAAD analysis engine through the STAAD input file (file extension .STD). That input file is a text file consisting of a series of commands in the STAAD command language which are executed sequentially. The commands contain either instructions or data pertaining to analysis and/or design. The elements and conventions of the STAAD command language are described in Section 5 of this manual.

The STAAD input file can be created through a text editor or the Graphical User Interface (GUI) modeling facility. In general, any plain-text editor may be utilized to edit or create the STAAD input file. The GUI Modeling facility creates the input file through an interactive, menu-driven graphics oriented procedure.

Note: Some of the automatic generation facilities of the STAAD command language will be re-interpreted by the GUI as lists of individual model elements upon editing the file using the GUI. A warning message is presented prior to this occurring. This does not result in any effective difference in the model or how it is analyzed or designed.

It is important to understand that STAAD.Pro is capable of analyzing a wide range of structures. While some parametric input features are available in the GUI, the formulation of input is the responsibility of you, the user. The program has no means of verifying that the structure input is that which was intended by the engineer.

1.3 Types of Structures

A **STRUCTURE** can be defined as an assemblage of elements. STAAD is capable of analyzing and designing structures consisting of both frame, plate/shell and solid elements. Almost any type of structure can be analyzed by STAAD.

SPACE

A₃D framed structure with loads applied in any plane. This structure type is the most general.

PLANE

Section 1 General Description

1.3 Types of Structures

This structure type is bound by a global X-Y coordinate system with loads in the same plane.

TRUSS

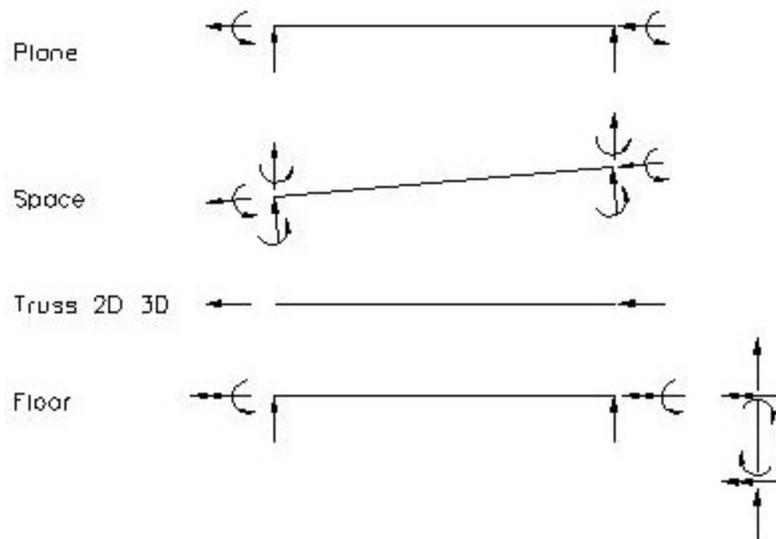
This structure type consists of truss members which can have only axial member forces and no bending in the members.

FLOOR

A 2D or 3D structure having no horizontal (global X or Z) movement of the structure [FX, FZ, and MY are restrained at every joint]. The floor framing (in global X-Z plane) of a building is an ideal example of a this type of structure. Columns can also be modeled with the floor in a **FLOOR** structure as long as the structure has no horizontal loading. If there is any horizontal load, it must be analyzed as a **SPACE** structure.

Specification of the correct structure type reduces the number of equations to be solved during the analysis. This results in a faster and more economic solution for the user. The degrees of freedom associated with frame elements of different types of structures is illustrated in the following figure.

Figure 1-1: Degrees of freedom in each type of Structure



1.4 Unit Systems

You are allowed to input data and request output in almost all commonly used engineering unit systems including **MKS**¹, **SI**², and **FPS**³. In the input file, the user may change units as many times as required. Mixing and matching between length and force units from different unit systems is also allowed.

The input unit for angles (or rotations) is degrees. However, in **JOINT DISPLACEMENT** output, the rotations are provided in radians.

For all output, the units are clearly specified by the program.

1.5 Structure Geometry and Coordinate Systems

A structure is an assembly of individual components such as beams, columns, slabs, plates etc.. In STAAD, frame elements and plate elements may be used to model the structural components. Typically, modeling of the structure geometry consists of two steps:

- A. Identification and description of joints or nodes.
- B. Modeling of members or elements through specification of connectivity (incidences) between joints.

In general, the term **MEMBER** will be used to refer to frame elements and the term **ELEMENT** will be used to refer to plate/shell and solid elements. Connectivity for **MEMBERS** may be provided through the **MEMBER INCIDENCE** command while connectivity for **ELEMENTS** may be provided through the **ELEMENT INCIDENCE** command.

STAAD uses two types of coordinate systems to define the structure geometry and loading patterns. The **GLOBAL** coordinate system is an arbitrary coordinate system in space which is utilized to specify the overall geometry & loading pattern of the structure. A **LOCAL** coordinate system is associated with each member (or element) and is utilized in **MEMBER END FORCE** output or local load specification.

¹Metre, Kilogram, and Second - A physical system of units with these fundamental units of measurement.

²International System of Units - From the French "Système international d'unités", which uses metres, kilograms, and seconds and the fundamental units of measurement.

³Foot, Pound, and Second - A physical system of units with these fundamental units of measurement.

Section 1 General Description

1.5 Structure Geometry and Coordinate Systems

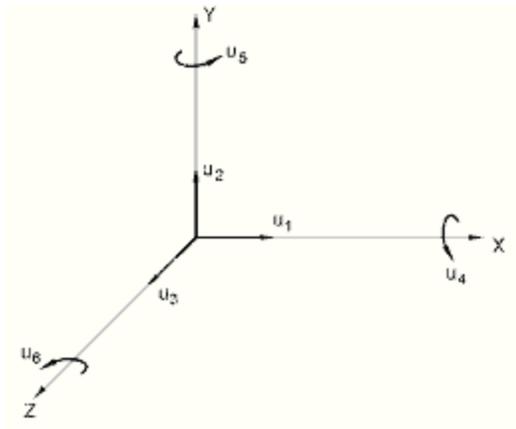
1.5.1 Global Coordinate System

The following coordinate systems are available for specification of the structure geometry.

Conventional Cartesian Coordinate System

This coordinate system is a rectangular coordinate system (X, Y, Z) which follows the orthogonal right hand rule. This coordinate system may be used to define the joint locations and loading directions. The translational degrees of freedom are denoted by u_1 , u_2 , u_3 and the rotational degrees of freedom are denoted by u_4 , u_5 & u_6 .

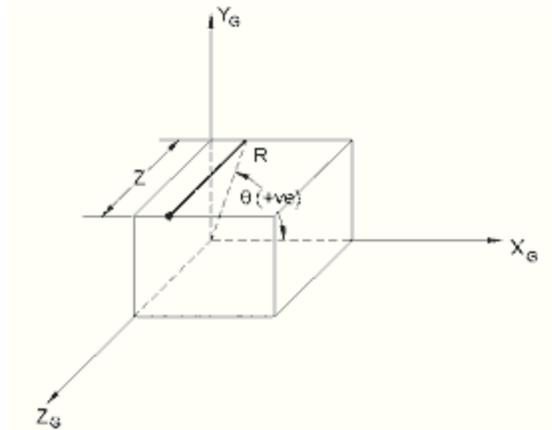
Figure 1-2: Cartesian (Rectangular) Coordinate System



Cylindrical Coordinate System

In this coordinate system, the X and Y coordinates of the conventional Cartesian system are replaced by R (radius) and θ (angle in degrees). The Z coordinate is identical to the Z coordinate of the Cartesian system and its positive direction is determined by the right hand rule.

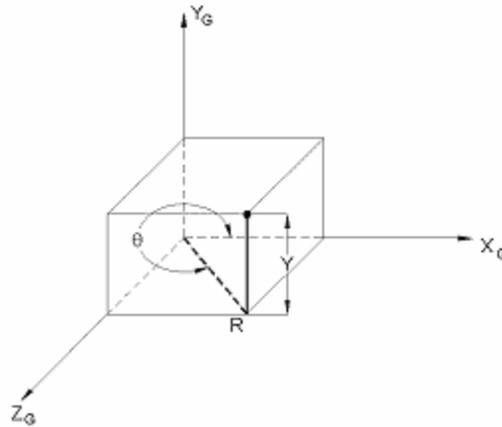
Figure 1-3: Cylindrical Coordinate System



Reverse Cylindrical Coordinate System

This is a cylindrical type coordinate system where the $R-\phi$ plane corresponds to the $X-Z$ plane of the Cartesian system. The right hand rule is followed to determine the positive direction of the Y axis.

Figure 1-4: Reverse Cylindrical Coordinate System



1.5.2 Local Coordinate System

A local coordinate system is associated with each member. Each axis of the local orthogonal coordinate system is also based on the right hand rule. Fig. 1.5 shows a beam member with start joint 'i' and end joint 'j'. The positive direction of the local x -axis is determined by joining 'i' to 'j' and projecting it in the same direction. The right hand rule may be applied to obtain the positive directions of the local y and z axes. The local y and z -axes coincide with the axes of the two principal moments of inertia. Note that the local coordinate system is always rectangular.

Section 1 General Description

1.5 Structure Geometry and Coordinate Systems

A wide range of cross-sectional shapes may be specified for analysis. These include rolled steel shapes, user specified prismatic shapes etc.. Fig. 1.6 shows local axis system(s) for these shapes.

Figure 1-5: When Global-Y is Vertical

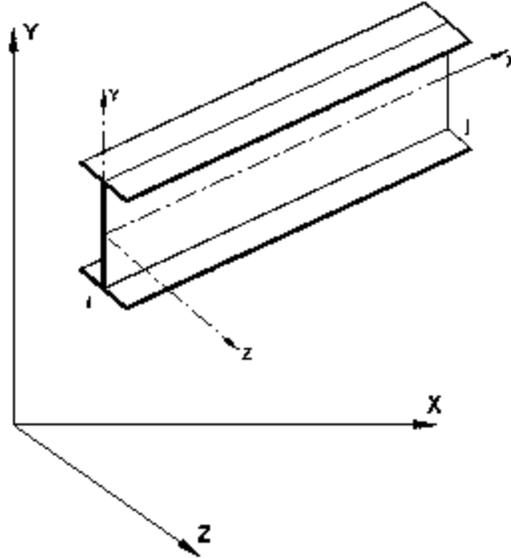


Figure 1-6: When Global-Z is Vertical (that is, SET Z UP is specified)

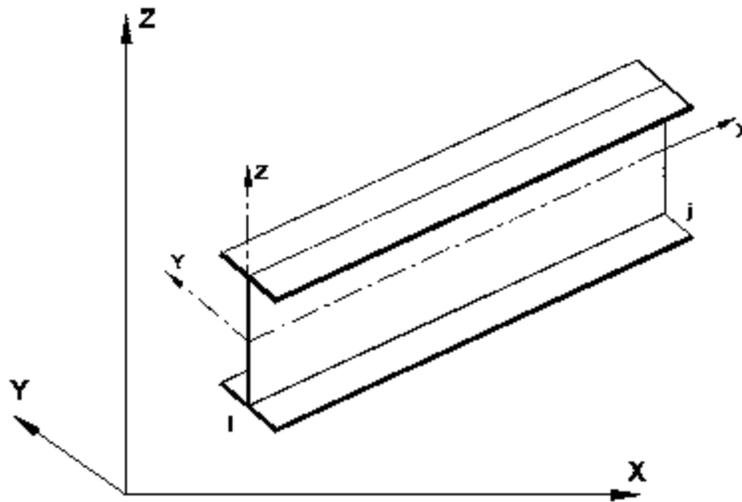


Table 1-1: Local axis system for various cross sections when global Y axis is vertical

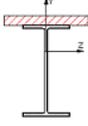
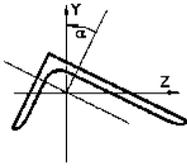
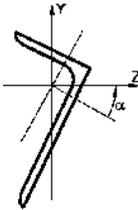
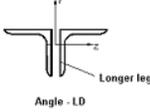
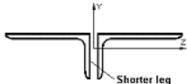
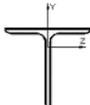
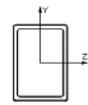
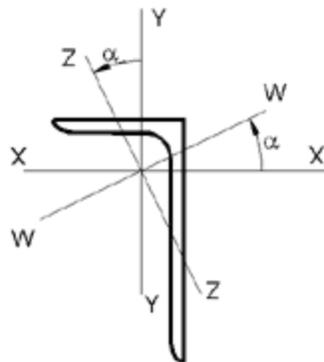
		
Wide Flange - ST	Wide Flange - TB	Wide Flange - CM
		
Angle - ST	Angle - RA	Angle - LD (Long legs back-to-back)
		
Angle - SD (Short legs back-to-back)	Wide Flange - T	Channel - ST
		
Channel - D	Prismatic	Tube - ST

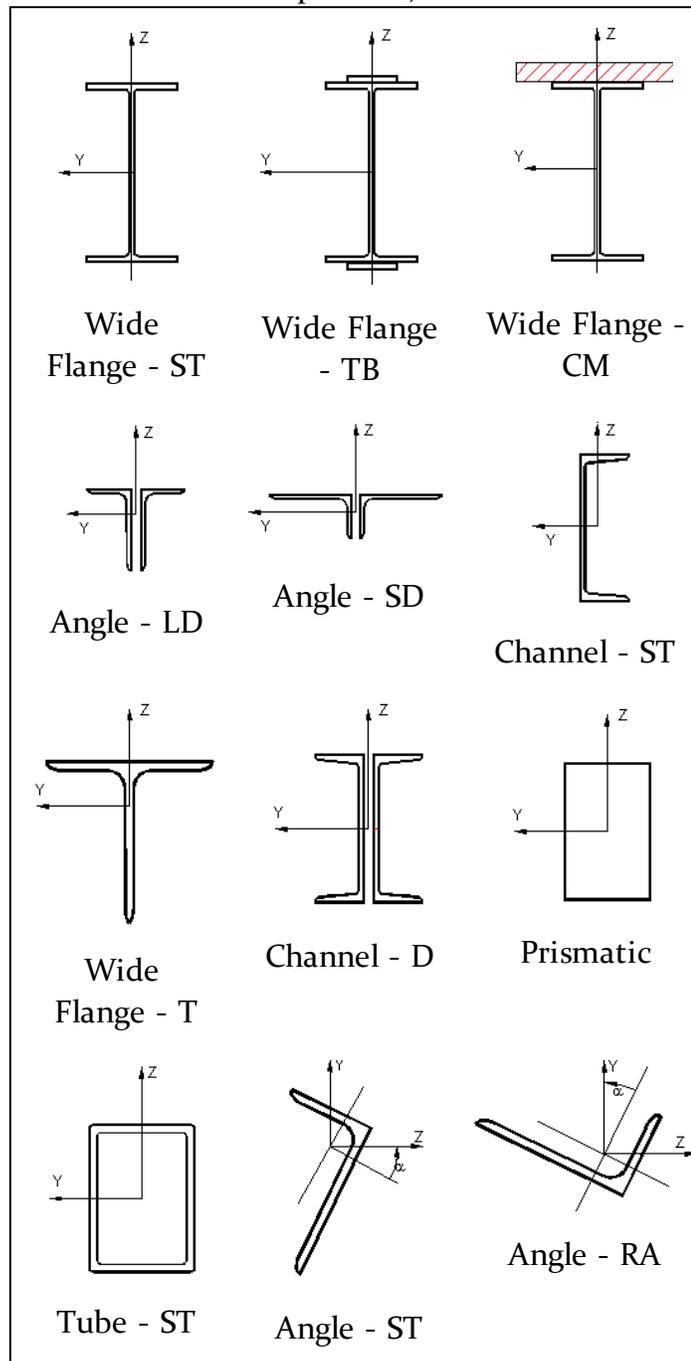
Figure 1-7: Local axis system for a single angle as defined in standard publications, which differs from the local axis of STAAD Angle - ST or Angle - RA sections.



Section 1 General Description

1.5 Structure Geometry and Coordinate Systems

Table 1-2: Local axis system for various cross sections when global Z axis is vertical (**SET Z UP** is specified).



Note: The local x-axis of the above sections is going into the paper

1.5.3 Relationship Between Global & Local Coordinates

Since the input for member loads can be provided in the local and global coordinate system and the output for member-end-forces is printed in the local coordinate system, it is important to know the relationship between the local and global coordinate systems. This relationship is defined by an angle measured in the following specified way. This angle will be defined as the beta (β) angle. For offset members the beta angle/reference point specifications are based on the offset position of the local axis, not the joint positions.

Beta Angle

When the local x-axis is parallel to the global Vertical axis, as in the case of a column in a structure, the beta angle is the angle through which the local z-axis (or local Y for **SET Z UP**) has been rotated about the local x-axis from a position of being parallel and in the same positive direction of the global Z-axis (global Y axis for **SET Z UP**).

When the local x-axis is not parallel to the global Vertical axis, the beta angle is the angle through which the local coordinate system has been rotated about the local x-axis from a position of having the local z-axis (or local Y for **SET Z UP**) parallel to the global X-Z plane (or global X-Y plane for **SET Z UP**) and the local y-axis (or local z for **SET Z UP**) in the same positive direction as the global vertical axis. Figure 1.7 details the positions for beta equals 0 degrees or 90 degrees. When providing member loads in the local member axis, it is helpful to refer to this figure for a quick determination of the local axis system.

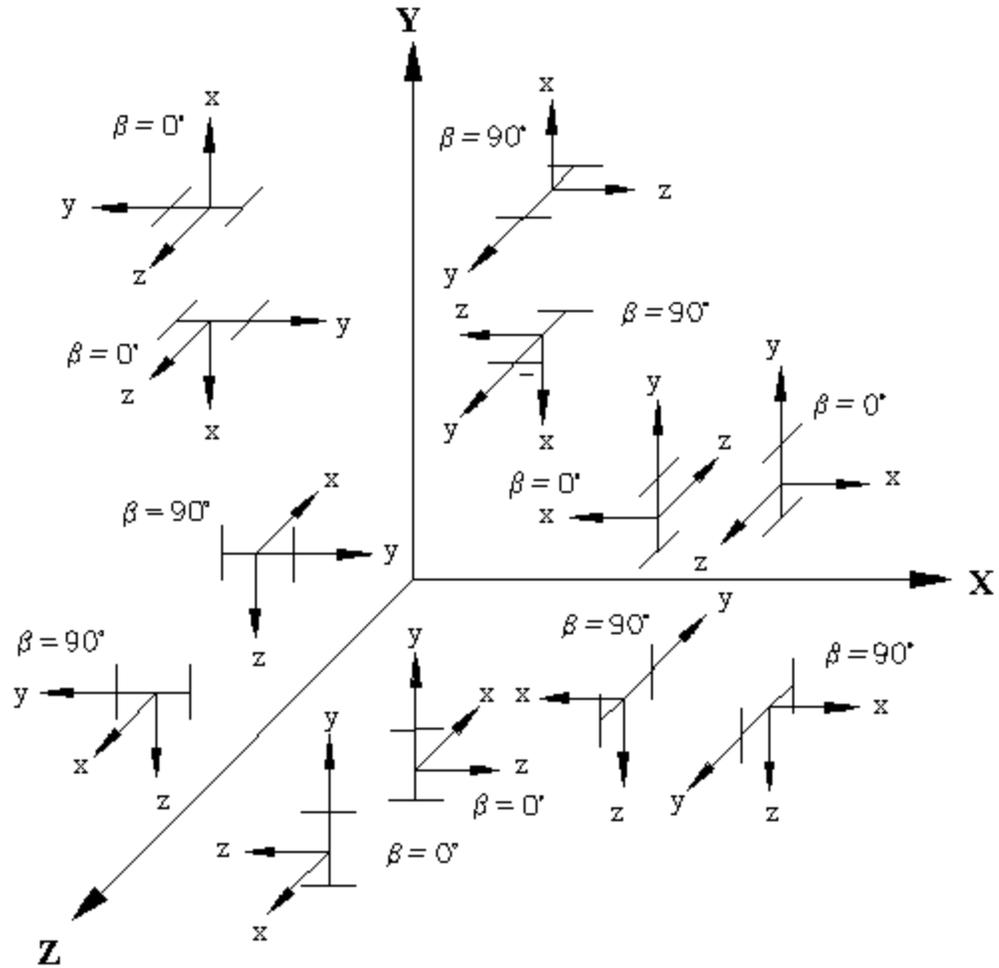
Reference Point

An alternative to providing the member orientation is to input the coordinates (or a joint number) which will be a reference point located in the member x-y plane (x-z plane for **SET Z UP**) but not on the axis of the member. From the location of the reference point, the program automatically calculates the orientation of the member x-y plane (x-z plane for **SET Z UP**).

Figure 1-8: Relationship between Global and Local axes

Section 1 General Description

1.5 Structure Geometry and Coordinate Systems



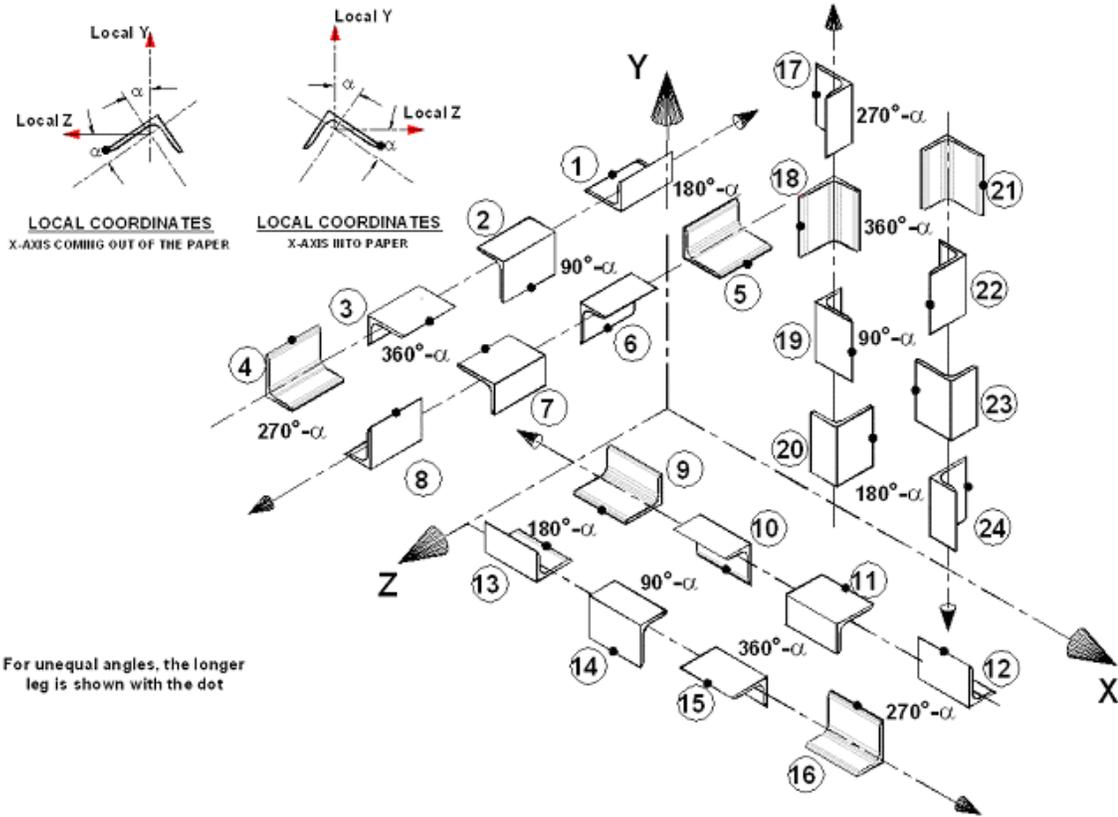
Reference Vector

This is yet another way to specify the member orientation. In the reference point method described above, the X,Y,Z coordinates of the point are in the global axis system. In a reference vector, the X,Y,Z coordinates are specified with respect to the local axis system of the member corresponding to the BETA 0 condition.

A direction vector is created by the program as explained in section 5.26.2 of this manual. The program then calculates the Beta Angle using this vector.

Figure 1-9: Beta rotation of equal & unequal legged 'ST' angles

1.5 Structure Geometry and Coordinate Systems



Note: The order of the joint numbers in the **MEMBER INCIDENCES** command determines the direction of the member's local x-axis.

Figure 1-10: Beta rotation of equal & unequal legged 'RA' angles

Section 1 General Description

1.5 Structure Geometry and Coordinate Systems

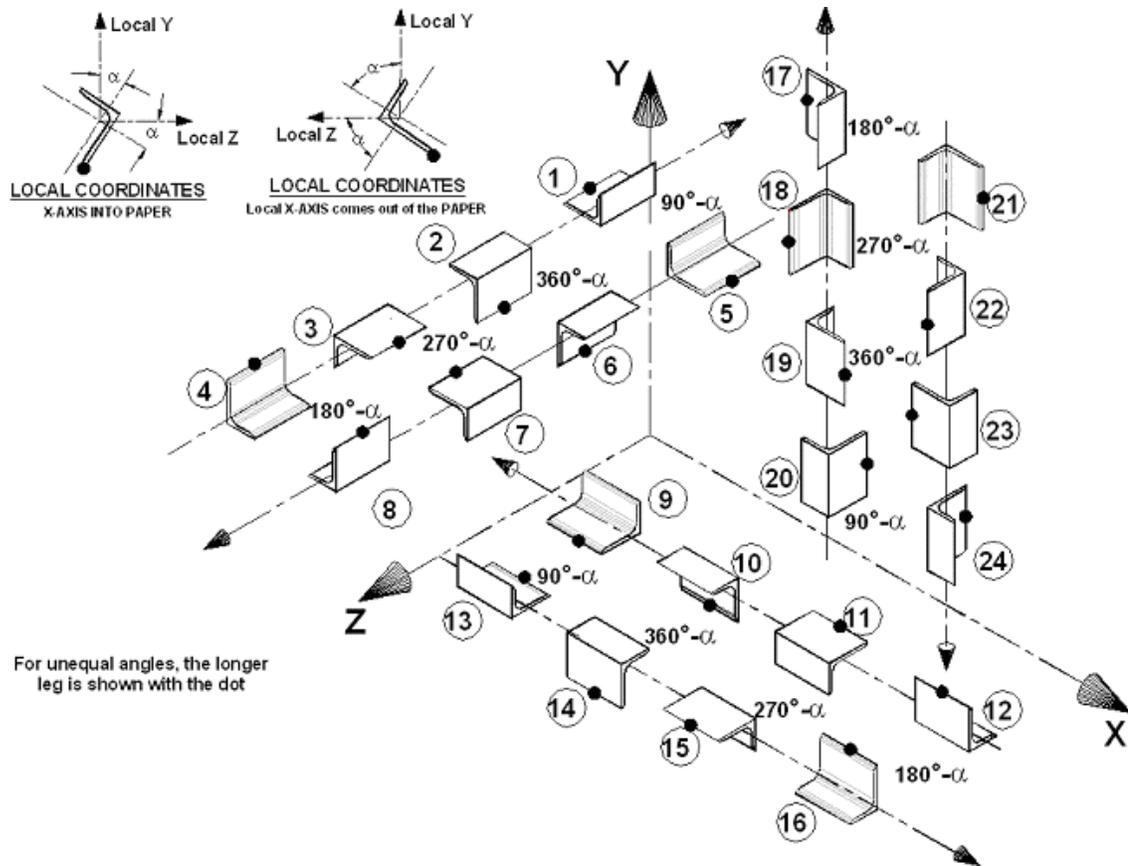


Figure 1-11: Member orientation for various Beta angles when Global-Y axis is vertical

1.5 Structure Geometry and Coordinate Systems

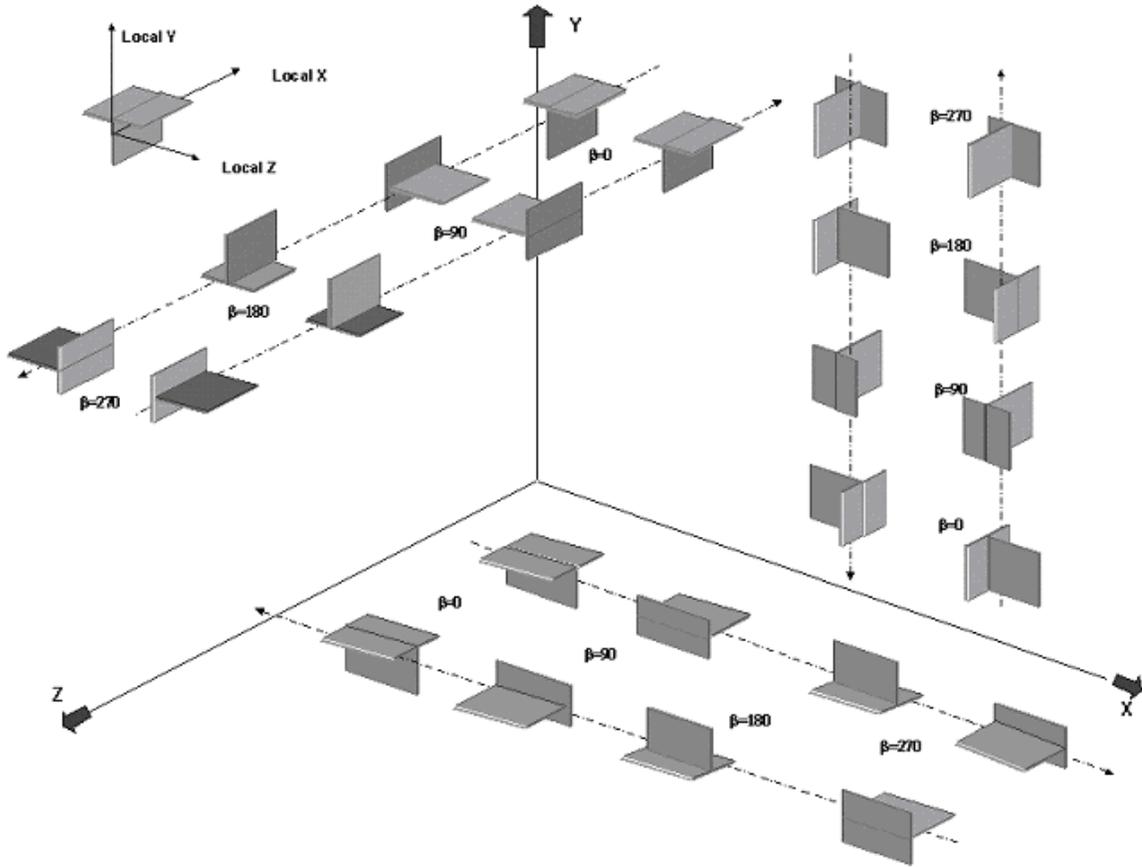


Figure 1-12: Member orientation for various Beta angles when Global-Z axis is vertical (that is, **SET Z UP** is specified)

Section 1 General Description

1.5 Structure Geometry and Coordinate Systems

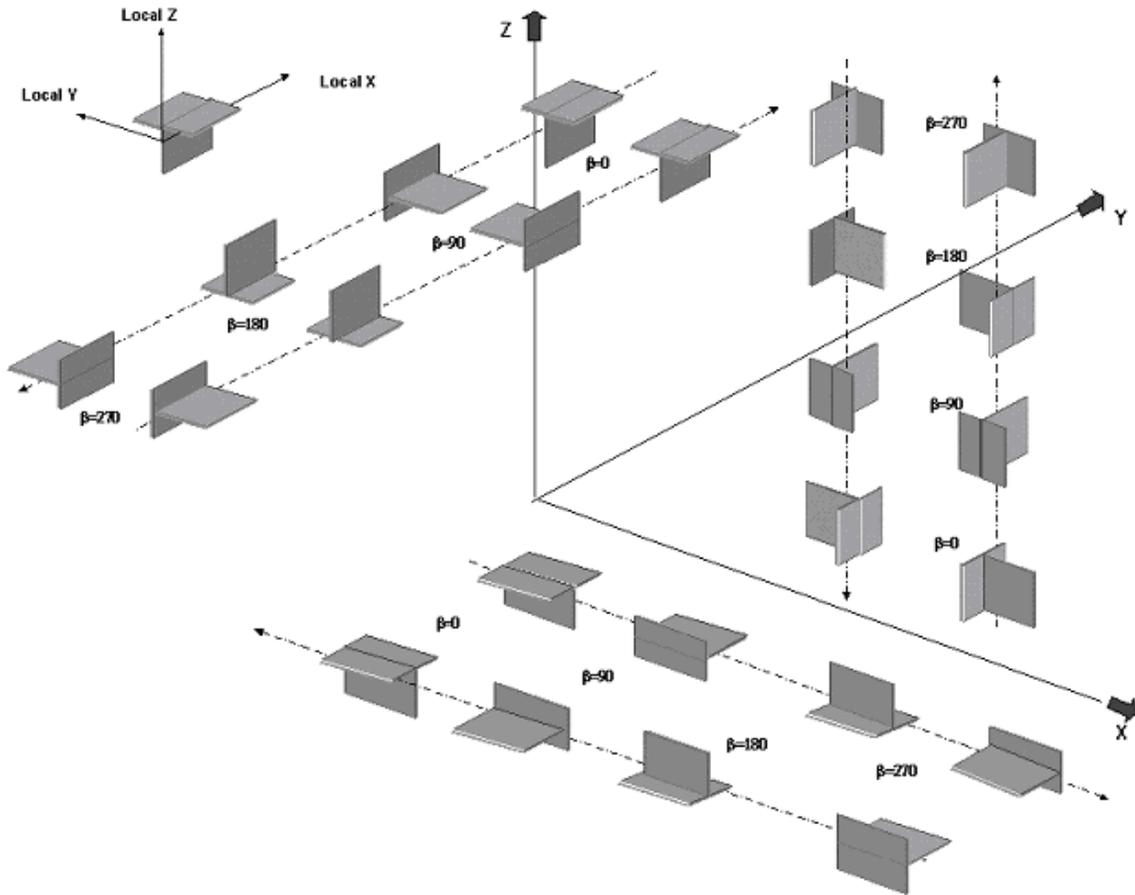
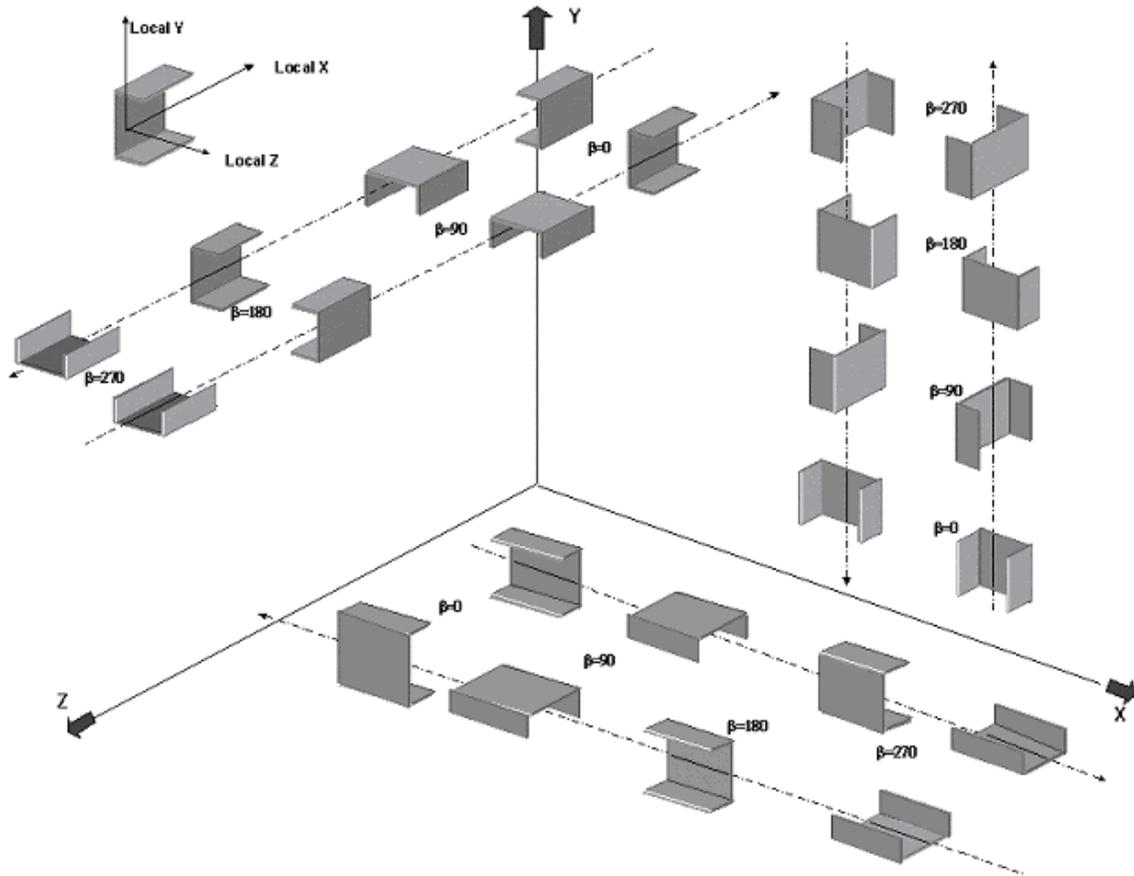


Figure 1-13: Member orientation for various Beta angles when Global-Y axis is vertical



1.6 Finite Element Information

STAAD.Pro is equipped with a plate/shell finite element, solid finite element and an entity called the surface element. The features of each is explained in the following sections.

1.6.1 Plate and Shell Element

The Plate/Shell finite element is based on the hybrid element formulation. The element can be 3-noded (triangular) or 4-noded (quadrilateral). If all the four nodes of a quadrilateral element do not lie on one plane, it is advisable to model them as triangular elements. The thickness of the element may be different from one node to another.

"Surface structures" such as walls, slabs, plates and shells may be modeled using finite elements. For convenience in generation of a finer mesh of plate/shell elements within a large area, a **MESH GENERATION** facility is available. See "Plate Element Mesh Generation" on page 309 for details.

Section 1 General Description

1.6 Finite Element Information

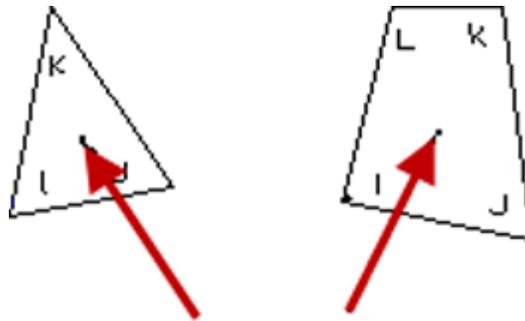
You may also use the element for **PLANE STRESS** action only (i.e., membrane/in-plane stiffness only). The **ELEMENT PLANE STRESS** command should be used for this purpose.

Geometry Modeling Considerations

The following geometry related modeling rules should be remembered while using the plate/shell element.

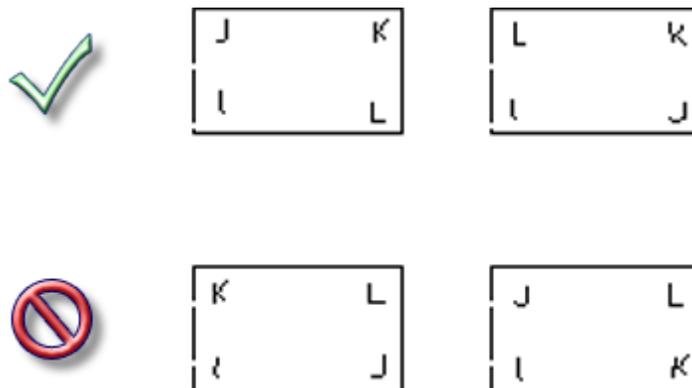
1. The program automatically generates a fictitious, center node "O" (see the following figure) at the element center.

Figure 1-14: Fictitious center node (in the case of triangular elements, a fourth node; in the case of rectangular elements, a fifth node)



2. While assigning nodes to an element in the input data, it is essential that the nodes be specified either clockwise or counter clockwise (see the following figure). For better efficiency, similar elements should be numbered sequentially.

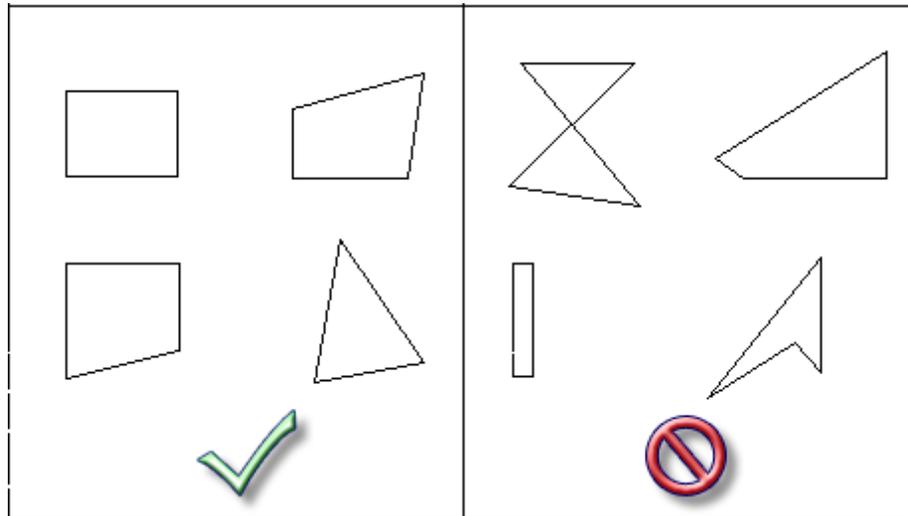
Figure 1-15: Examples of correct and incorrect numbering sequences



3. Element aspect ratio should not be excessive. They should be on the order of 1:1, and preferably less than 4:1.
4. Individual elements should not be distorted. Angles between two adjacent

element sides should not be much larger than 90 and never larger than 180.

Figure 1-16: Some examples of good and bad elements in terms of the angles



Load Specification for Plate Elements

Following load specifications are available:

1. Joint loads at element nodes in global directions.
2. Concentrated loads at any user specified point within the element in global or local directions.
3. Uniform pressure on element surface in global or local directions.
4. Partial uniform pressure on user specified portion of element surface in global or local directions.
5. Linearly varying pressure on element surface in local directions.
6. Temperature load due to uniform increase or decrease of temperature.
7. Temperature load due to difference in temperature between top and bottom surfaces of the element.

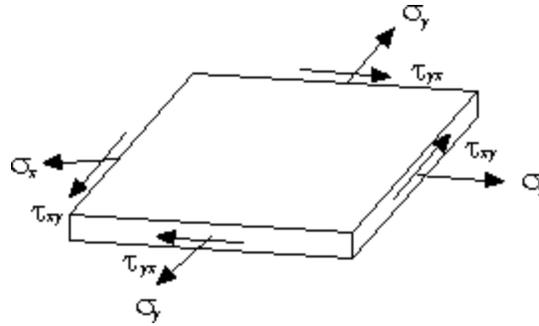
Theoretical Basis

The STAAD plate finite element is based on hybrid finite element formulations. An incomplete quadratic stress distribution is assumed. For plane stress action, the assumed stress distribution is as follows.

Figure 1-17: Assumed stress distribution

Section 1 General Description

1.6 Finite Element Information



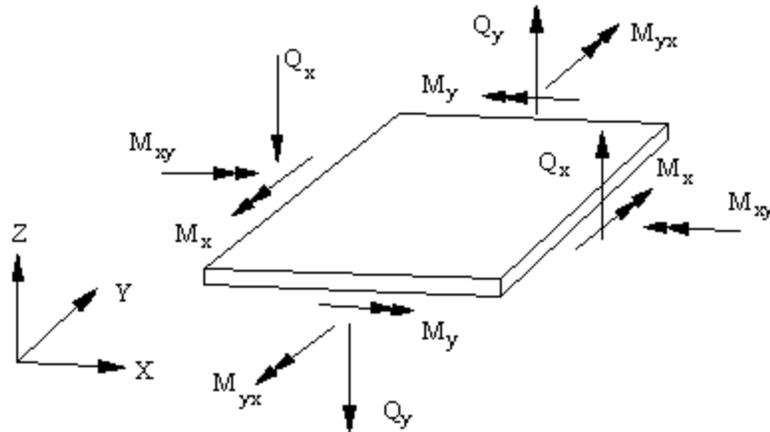
The incomplete quadratic assumed stress distribution:

$$\begin{pmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{pmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 & 0 & x^2 & 2xy & 0 \\ 0 & 0 & 0 & 1 & x & y & 0 & y^2 & 0 & 2xy \\ 0 & -y & 0 & 0 & 0 & -x & 1 & -2xy & -y^2 & -x^2 \end{bmatrix} \begin{pmatrix} a_1 \\ a_2 \\ a_3 \\ \dots \\ a_{10} \end{pmatrix}$$

a_1 through a_{10} = constants of stress polynomials

The following quadratic stress distribution is assumed for plate bending action:

Figure 1-18: Quadratic stress distribution assumed for bending



The incomplete quadratic assumed stress distribution:

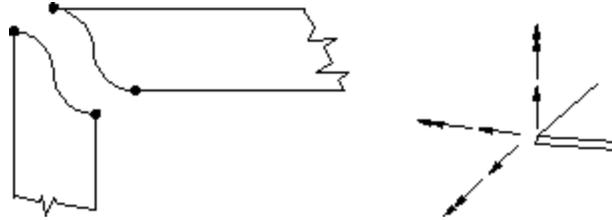
$$\begin{pmatrix} M_x \\ M_y \\ M_{xy} \\ Q_x \\ Q_y \end{pmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 & 0 & 0 & 0 & x^2 & xy & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y & 0 & 0 & 0 & 0 & 0 & xy & y^2 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & -xy & 0 & 0 & -xy \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & 0 & -xy \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & 1 & 0 & -y & 0 & x & y \end{bmatrix} \begin{pmatrix} a_1 \\ a_2 \\ a_3 \\ \dots \\ a_{12} \\ a_{13} \end{pmatrix}$$

a_1 through a_{13} = constants of stress polynomials

The distinguishing features of this finite element are:

1. Displacement compatibility between the plane stress component of one element and the plate bending component of an adjacent element which is at an angle to the first (see the following figure) is achieved by the elements. This compatibility requirement is usually ignored in most flat shell/plate elements.

Figure 1-19: Adjacent elements at some angle



2. The out of plane rotational stiffness from the plane stress portion of each element is usefully incorporated and not treated as a dummy as is usually done in most commonly available commercial software.
3. Despite the incorporation of the rotational stiffness mentioned previously, the elements satisfy the patch test absolutely.
4. These elements are available as triangles and quadrilaterals, with corner nodes only, with each node having six degrees of freedom.
5. These elements are the simplest forms of flat shell/plate elements possible with corner nodes only and six degrees of freedom per node. Yet solutions to sample problems converge rapidly to accurate answers even with a large mesh size.
6. These elements may be connected to plane/space frame members with full displacement compatibility. No additional restraints/releases are required.
7. Out of plane shear strain energy is incorporated in the formulation of the plate bending component. As a result, the elements respond to Poisson boundary conditions which are considered to be more accurate than the customary Kirchoff boundary conditions.
8. The plate bending portion can handle thick and thin plates, thus extending the usefulness of the plate elements into a multiplicity of problems. In addition, the thickness of the plate is taken into consideration in calculating the out of plane shear.
9. The plane stress triangle behaves almost on par with the well known linear stress triangle. The triangles of most similar flat shell elements incorporate the constant stress triangle which has very slow rates of convergence. Thus

Section 1 General Description

1.6 Finite Element Information

the triangular shell element is very useful in problems with double curvature where the quadrilateral element may not be suitable.

10. Stress retrieval at nodes and at any point within the element.

Plate Element Local Coordinate System

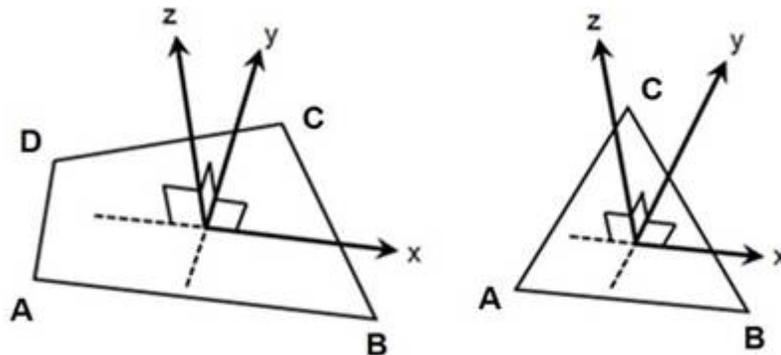
The orientation of local coordinates is determined as follows:

1. The vector pointing from I to J is defined to be parallel to the local x- axis.
2. For triangles: the cross-product of vectors IJ and JK defines a vector parallel to the local z-axis, i.e., $z = IJ \times JK$.

For quads: the cross-product of vectors IJ and JL defines a vector parallel to the local z-axis, i.e., $z = IJ \times JL$.

3. The cross-product of vectors z and x defines a vector parallel to the local y-axis, i.e., $y = z \times x$.
4. The origin of the axes is at the center (average) of the four joint locations (three joint locations for a triangle).

Figure 1-20: Element origin



Output of Plate Element Stresses and Moments

For the sign convention of output stress and moments, please see figures below.

ELEMENT stress and moment output is available at the following locations:

- A. Center point of the element.
- B. All corner nodes of the element.
- C. At any user specified point within the element.

Following are the items included in the **ELEMENT STRESS** output.

Table 1-3: Items included in the Stress Element output

Title	Description
SQX, SQY	Shear stresses (Force/ unit length/ thickness)
SX, SY	Membrane stresses (Force/unit length/ thickness)
SXY	Inplane Shear Stress (Force/unit length/ thickness)
MX, MY, MXY	Moments per unit width (Force x Length/length) (For Mx, the unit width is a unit distance parallel to the local Y axis. For My, the unit width is a unit distance parallel to the local X axis. Mx and My cause bending, while Mxy causes the element to twist out-of-plane.)
SMAX, SMIN	Principal stresses in the plane of the element (Force/unit area). The 3rd principal stress is 0.0
TMAX	Maximum 2D shear stress in the plane of the element (Force/unit area)
VONT, VONB	3D Von Mises stress at the top and bottom surfaces, where: $VM = 0.707[(S_{MAX} - S_{MIN})^2 + S_{MAX}^2 + S_{MIN}^2]^{1/2}$
TRESMAT, TRESMAB	Tresca stress, where TRESMA = MAX[(Smax-Smin) , (Smax) , (Smin)]

Notes

1. All element stress output is in the local coordinate system. The direction and sense of the element stresses are explained in the following section.
2. To obtain element stresses at a specified point within the element, you

Section 1 General Description

1.6 Finite Element Information

must provide the location (local X, local Y) in the coordinate system for the element. The origin of the local coordinate system coincides with the center of the element.

3. The 2 nonzero Principal stresses at the surface (SMAX & SMIN), the maximum 2D shear stress (TMAX), the 2D orientation of the principal plane (ANGLE), the 3D Von Mises stress (VONT & VONB), and the 3D Tresca stress (TRESCAT & TRESCAB) are also printed for the top and bottom surfaces of the elements. The top and the bottom surfaces are determined on the basis of the direction of the local z-axis.
4. The third principal stress is assumed to be zero at the surfaces for use in Von Mises and Tresca stress calculations. However, the TMAX and ANGLE are based only on the 2D inplane stresses (SMAX & SMIN) at the surface. The 3D maximum shear stress at the surface is not calculated but would be equal to the 3D Tresca stress divided by 2.0.

Sign Convention of Plate Element Stresses and Moments

See "Print Specifications " on page 714 for definitions of the nomenclature used in the following figures.

Figure 1-21: Sign conventions for plate stresses and moments

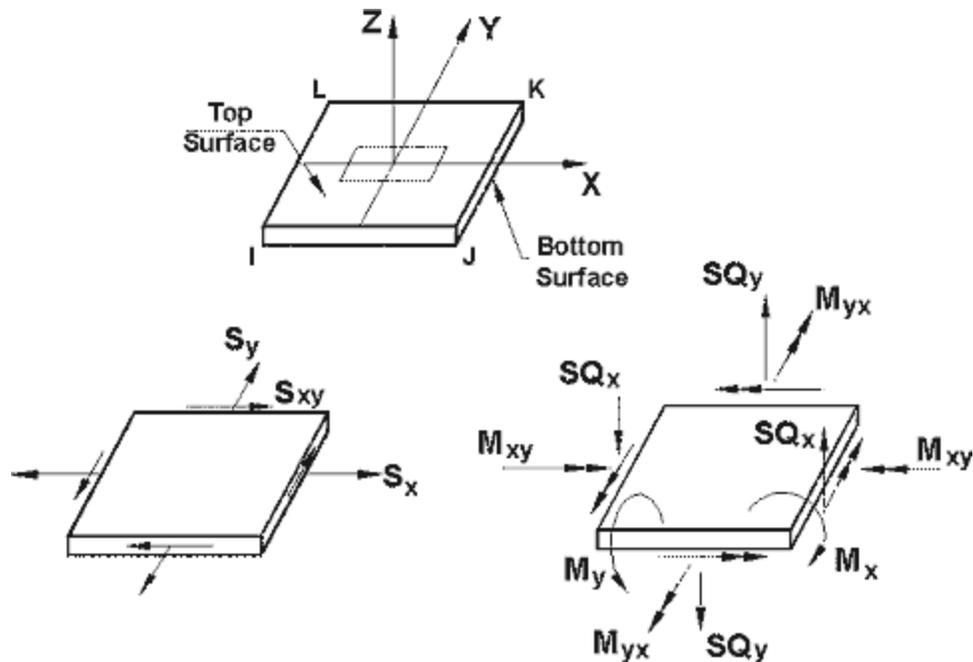
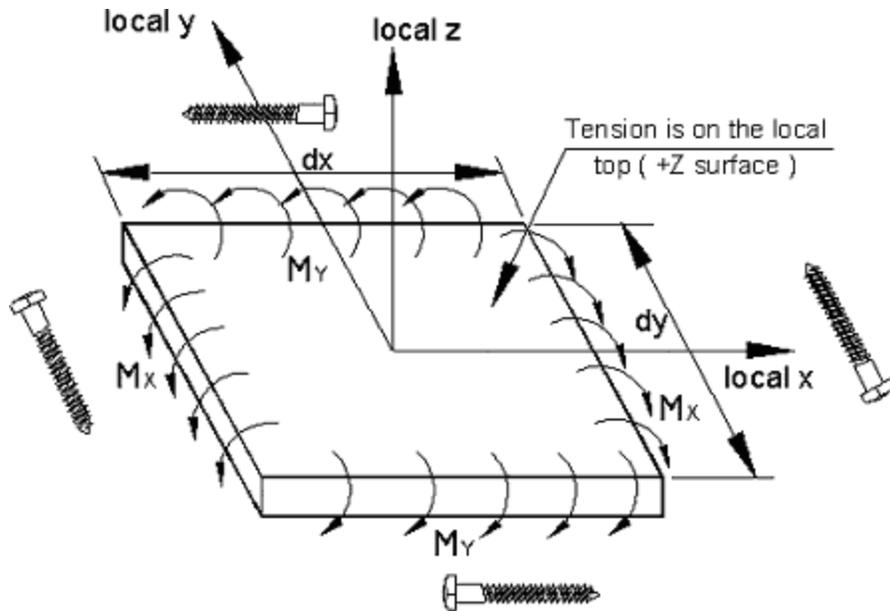


Figure 1-22: Sign convention for plate bending



M_x is the Bending Moment on the local *x* face and the local *x*-face is the face perpendicular to the local *x*-axis.

M_y is the Bending Moment on the local *y* face and the local *y*-face is the face perpendicular to the local *y*-axis.

Figure 1-23: Stress caused by M_x

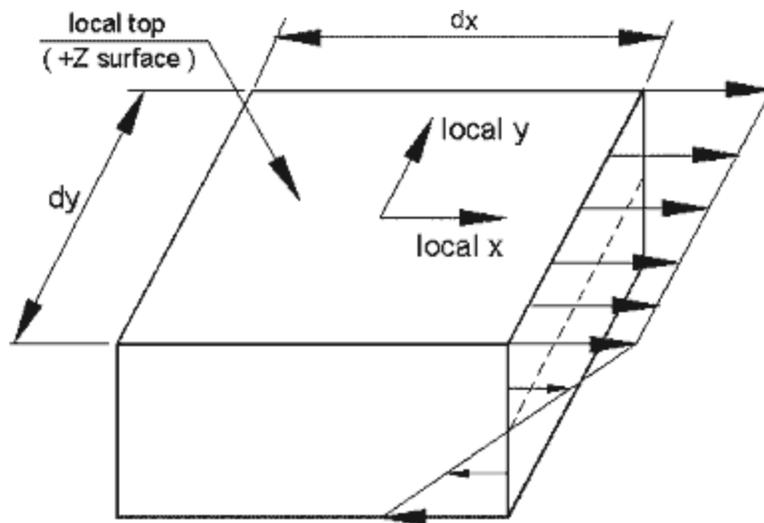


Figure 1-24: Stress caused by M_y

Section 1 General Description

1.6 Finite Element Information

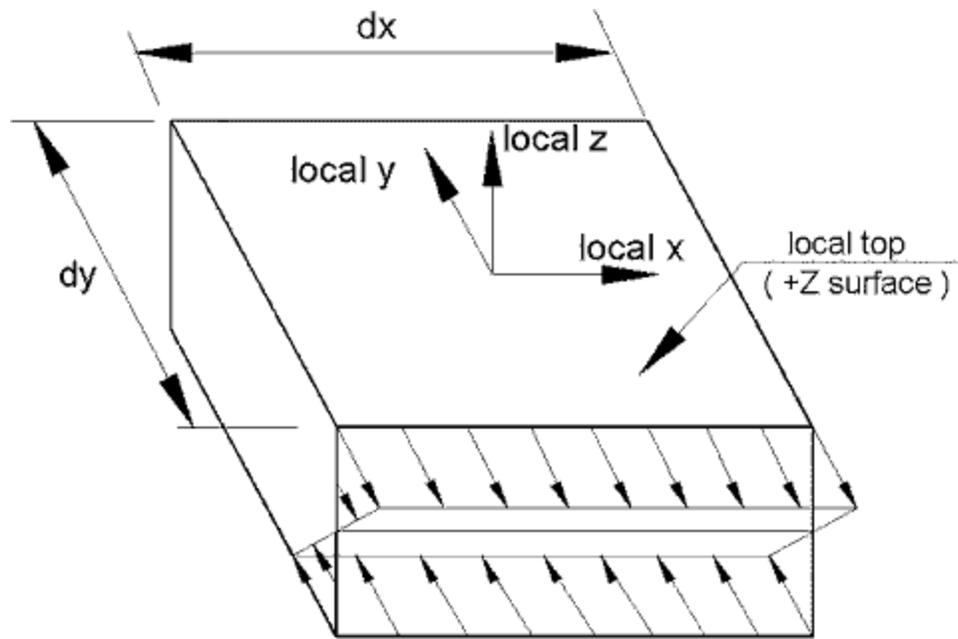


Figure 1-25: Torsion

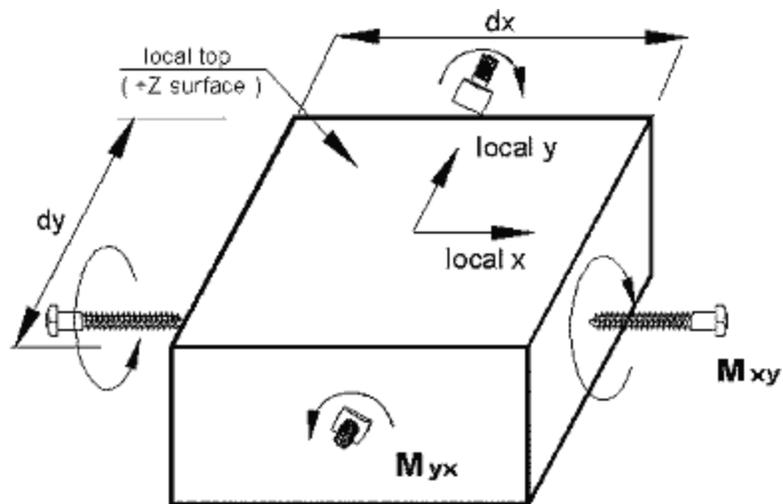


Figure 1-26: Membrane stress S_x and S_y

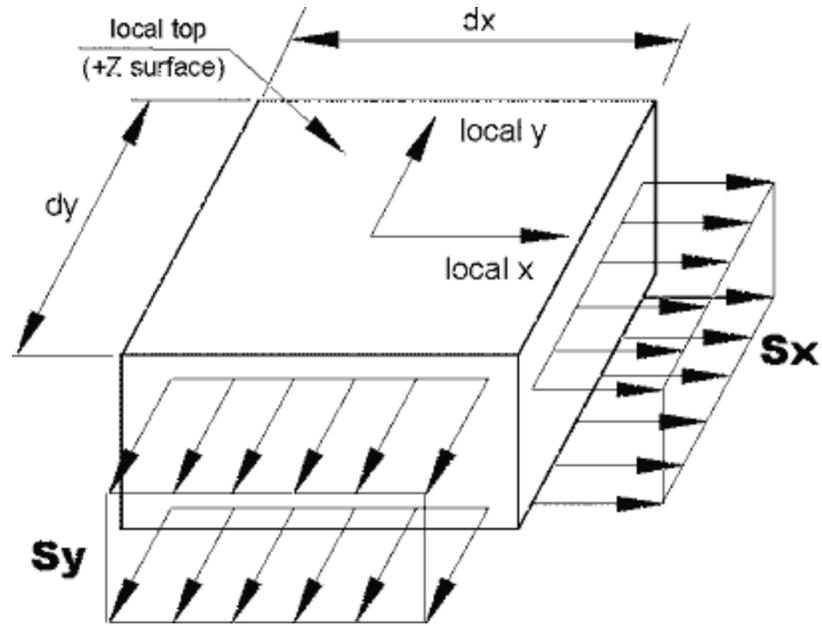


Figure 1-27: In-plane shear stresses S_{xy} and S_{yx}

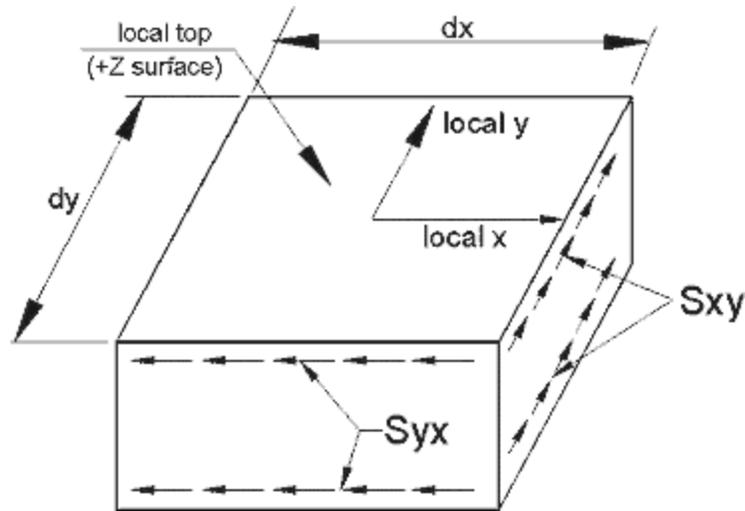
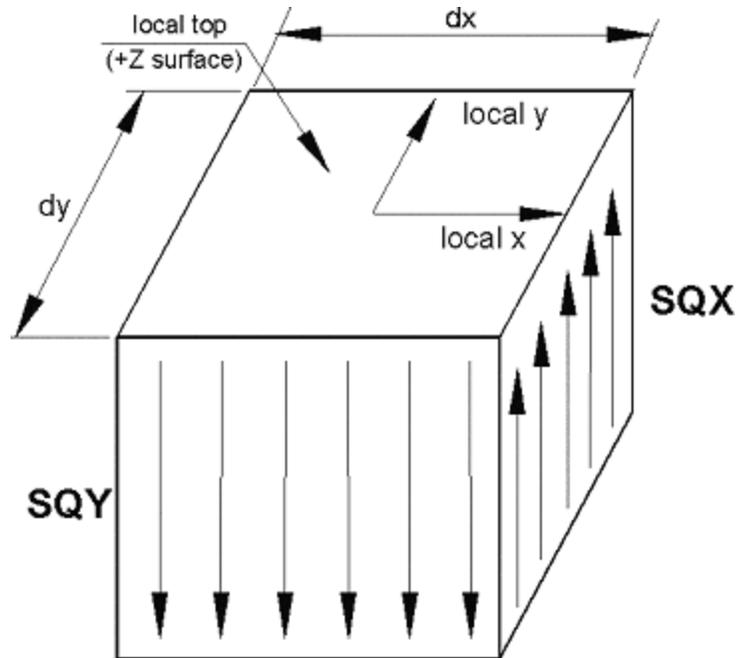


Figure 1-28: Out-of-plane shear stresses S_{QX} and S_{QY}

Section 1 General Description

1.6 Finite Element Information



Members, plate elements, solid elements and surface elements can all be part of a single STAAD model. The **MEMBER INCIDENCES** input must precede the **INCIDENCE** input for plates, solids or surfaces. All **INCIDENCES** must precede other input such as properties, constants, releases, loads, etc. The selfweight of the finite elements is converted to joint loads at the connected nodes and is not used as an element pressure load.

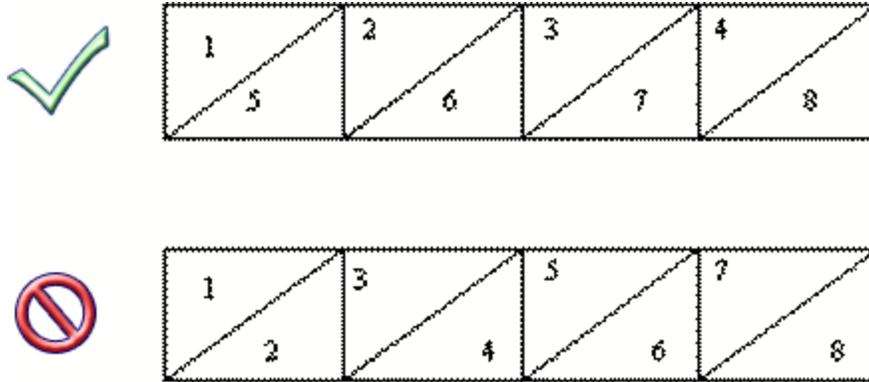
Plate Element Numbering

During the generation of element stiffness matrix, the program verifies whether the element is same as the previous one or not. If it is same, repetitive calculations are not performed. The sequence in which the element stiffness matrix is generated is the same as the sequence in which elements are input in element incidences.

Therefore, to save some computing time, similar elements should be numbered sequentially. The following figure shows examples of efficient and non-efficient element numbering.

However, you have to decide between adopting a numbering system which reduces the computation time versus a numbering system which increases the ease of defining the structure geometry.

Figure 1-29: Examples of efficient and inefficient element numbering



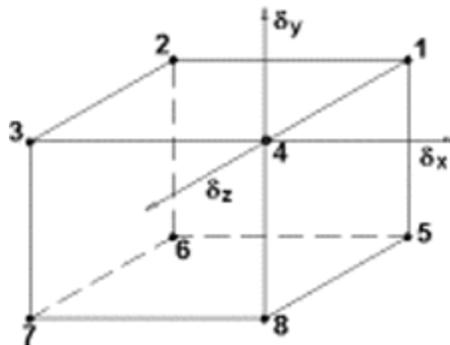
1.6.2 Solid Element

Solid elements enable the solution of structural problems involving general three dimensional stresses. There is a class of problems such as stress distribution in concrete dams, soil and rock strata where finite element analysis using solid elements provides a powerful tool.

Theoretical Basis

The solid element used in STAAD is of eight-noded, isoparametric type. These elements have three translational degrees-of-freedom per node.

Figure 1-30: eight-noded, isoparametric solid element

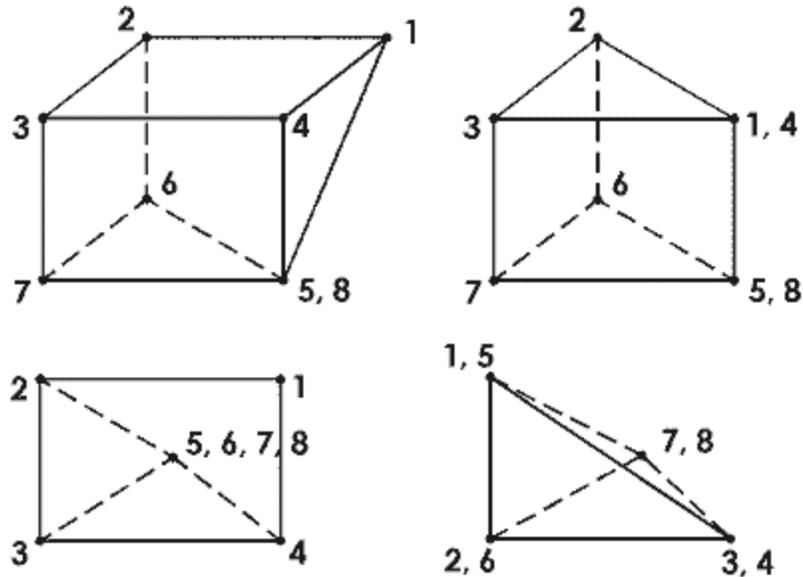


By collapsing various nodes together, an eight noded solid element can be degenerated to the following forms with four to seven nodes. Joints 1, 2, and 3 must be retained as a triangle.

Figure 1-31: Forms of a collapsed eight-noded solid element

Section 1 General Description

1.6 Finite Element Information



The stiffness matrix of the solid element is evaluated by numerical integration with eight Gauss-Legendre points. To facilitate the numerical integration, the geometry of the element is expressed by interpolating functions using natural coordinate system, (r,s,t) of the element with its origin at the center of gravity. The interpolating functions are shown below:

$$X = \sum_{i=1}^8 h_i x_i, \quad y = \sum_{i=1}^8 h_i y_i, \quad z = \sum_{i=1}^8 h_i z_i$$

where x, y and z are the coordinates of any point in the element and $x_i, y_i, z_i, i=1, \dots, 8$ are the coordinates of nodes defined in the global coordinate system. The interpolation functions, h_i are defined in the natural coordinate system, (r,s,t). Each of r,s and t varies between -1 and +1. The fundamental property of the unknown interpolation functions h_i is that their values in natural coordinate system is unity at node, i, and zero at all other nodes of the element. The element displacements are also interpreted the same way as the geometry. For completeness, the functions are given below:

$$u = \sum_{i=1}^8 h_i u_i, \quad v = \sum_{i=1}^8 h_i v_i, \quad w = \sum_{i=1}^8 h_i w_i$$

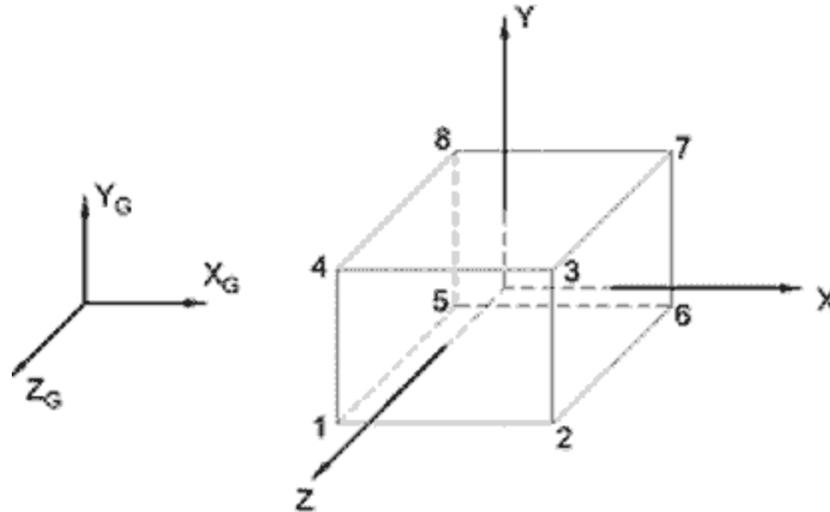
where u, v and w are displacements at any point in the element and $u_i, v_i, w_i, i=1, \dots, 8$ are corresponding nodal displacements in the coordinate system used to describe the geometry.

Three additional displacement "bubble" functions which have zero displacements at the surfaces are added in each direction for improved shear performance to form a 33x33 matrix. Static condensation is used to reduce this matrix to a 24x24 matrix at the corner joints.

Local Coordinate System

The local coordinate system used in solid element is the same as the global system.

Figure 1-32: Local coordinate system for a solid element



Properties and Constants

Unlike members and shell (plate) elements, no properties are required for solid elements. However, the constants such as modulus of elasticity and Poisson's ratio are to be specified. Also, density needs to be provided if selfweight is included in any load case.

Output of Element Stresses

Element stresses may be obtained at the center and at the joints of the solid element. The items that are printed are :

- Normal Stresses : S_{XX} , S_{YY} and S_{ZZ}
- Shear Stresses : S_{XY} , S_{YZ} and S_{ZX}
- Principal stresses : S_1 , S_2 and S_3
- Von Mises stresses: $S_{IGE} = 0.707[(S_1 - S_2)^2 + (S_2 - S_3)^2 + (S_3 - S_1)^2]^{1/2}$

Direction cosines : six direction cosines are printed, following the expression DC, corresponding to the first two principal stress directions.

Section 1 General Description

1.6 Finite Element Information

1.6.3 Surface Element

For any panel type of structural component, modeling requires breaking it down into a series of plate elements for analysis purposes. This is what is known in stress analysis parlance as meshing. When you choose to model the panel component using plate elements, you are then taking on the responsibility of meshing. Thus, what the program sees is a series of elements. It is your responsibility to ensure that meshing is done properly. Examples of these are available in example problems 9, 10, 23, 27, etc. (Refer to the Examples manual) where individual plate elements are specified.

By using the Surface type of entity, the burden of meshing is shifted from you to the program to some degree. The entire wall or slab is hence represented by just a few "Surface" entities, instead of hundreds of elements. When the program goes through the analysis phase, it will then automatically subdivide the surface into elements. Therefore, you do not have to instruct the program in what manner to carry out the meshing.

The attributes associated with the surface element, and the sections of this manual where the information may be obtained, are listed below:

Attributes	Related Sections
Surfaces Incidences	5.13.3
Openings in surface	5.13.3
Local Coordinates system for surfaces	1.6.3
Specifying sections for stress/force output	5.13.3
Property for surfaces	5.21.2
Material constants	5.26.3
Surface loading	5.32.3.4
Stress/Force output printing	5.42
Shear Wall Design	3.8.2, 5.55

Local Coordinate system for surfaces

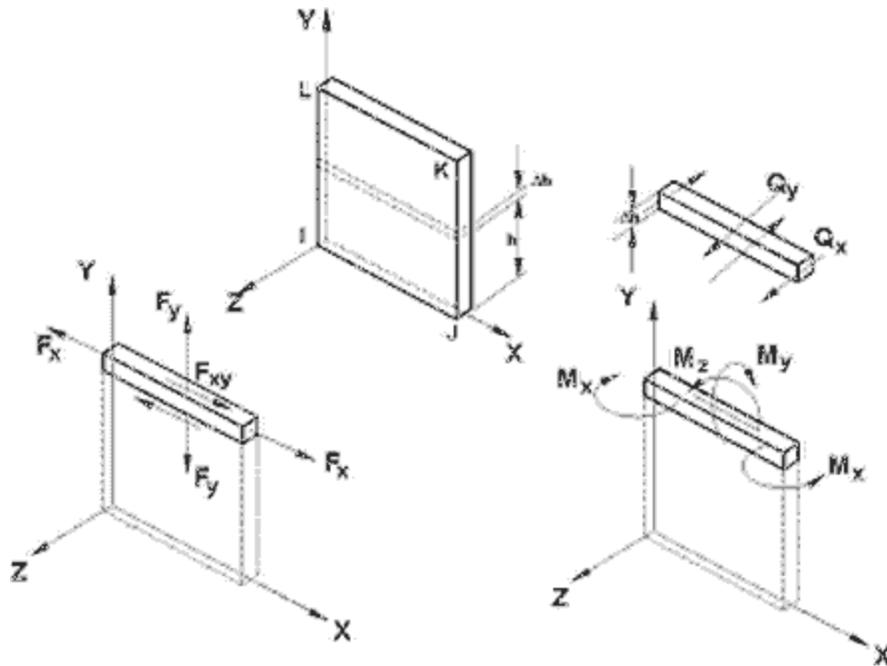
The origin and orientation of the local coordinate system of a surface element depends on the order in which the boundary nodal points are listed and position of the surface element in relation to the global coordinate system.

Let X , Y , and Z represent the local and GX , GY , and GZ the global axis vectors, respectively. The following principles apply.

- Origin of X - Y - Z is located at the first node specified.
- Direction of Z may be established by the right hand corkscrew rule, where the thumb indicates the positive Z direction, and the fingers point along the circumference of the element from the first to the last node listed.
- X is a vector product of GY and Z ($X = GY \times Z$). If GY and Z are parallel, X is taken as a vector parallel to GX .
- Finally, Y is a vector product of Z and X ($Y = Z \times X$).

The diagram below shows directions and sign convention of local axes and forces.

Figure 1-33: Local axis & sign convention of surface forces



1.7 Member Properties

The following types of member property specifications are available in STAAD:

Section 1 General Description

1.7 Member Properties

Shear Area for members refers to the shear stiffness effective area. Shear stiffness effective area is used to calculate shear stiffness for the member stiffness matrix.

As an example: for a rectangular cross section, the shear stiffness effective area is usually taken as 0.83 (Roark) to 0.85 (Cowper) times the cross sectional area. A shear area of less than the cross sectional area will reduce the stiffness. A typical shearing stiffness term is

$$(12EI/L^3)/(1+\Phi)$$

Where:

$$\Phi = (12 EI) / (GA_s L^2)$$

A_s = the shear stiffness effective area.

Phi (Φ) is usually ignored in basic beam theory. STAAD will include the PHI term unless the **SET SHEAR** command is entered.

Shear stress effective area is a different quantity that is used to calculate shear stress and in code checking. For a rectangular cross section, the shear stress effective area is usually taken as two-thirds (0.67x) of the cross sectional area.

Shear stress in STAAD may be from one of three methods.

1. (Shear Force)/(Shear stress effective area)

This is the case where STAAD computes the area based on the cross section parameters.

2. (Shear Force)/(Shear stiffness effective area)

This is the case where STAAD uses the shear area entered.

3. (V Q)/(I t)

In some codes and for some cross sections, STAAD uses this method.

The values that STAAD uses for shear area for shear deformation calculation can be obtained by specifying the command **PRINT MEMBER PROPERTIES**.

The output for this will provide this information in all circumstances: when AY and AZ are not provided, when AY and AZ are set to zero, when AY and AZ are set to very large numbers, when properties are specified using **PRISMATIC**, when properties are specified through a user table, when properties are specified through from the built-in-table, etc.

1.7.1 Prismatic Properties

The following prismatic properties are required for analysis:

- AX = Cross sectional area
- IX = Torsional constant
- IY = Moment of inertia about y-axis.
- IZ = Moment of inertia about z-axis.

In addition, the user may choose to specify the following properties:

- AY = Effective shear area for shear force parallel to local y-axis.
- AZ = Effective shear area for shear force parallel to local z-axis.
- YD = Depth of section parallel to local y-axis.
- ZD = Depth of section parallel to local z-axis.

For T-beams, YD , ZD , YB & ZB must be specified. These terms, which are shown in the next figure are:

- YD = Total depth of section (top fiber of flange to bottom fiber of web)
- ZD = Width of flange
- YB = Depth of stem
- ZB = Width of stem

For Trapezoidal beams, YD , ZD & ZB must be specified. These terms, which too are shown in the next figure are:

- YD = Total depth of section
- ZD = Width of section at top fiber
- ZB = Width of section at bottom fiber

Top & bottom are defined as positive side of the local Z axis, and negative side of the local Z axis respectively.

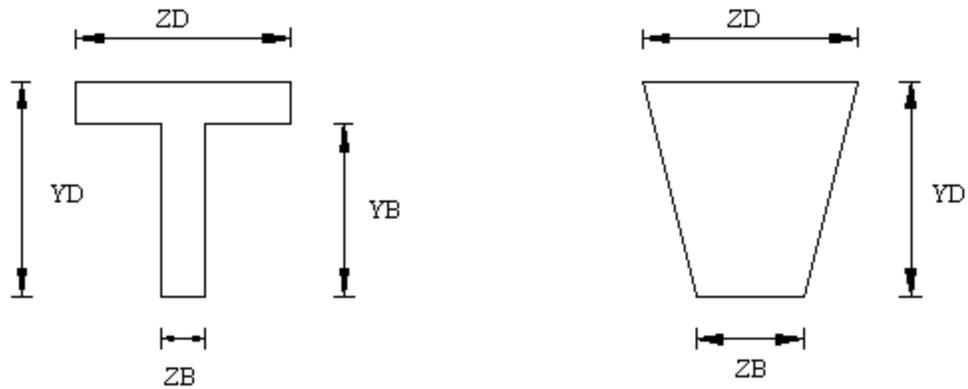
STAAD automatically considers the additional deflection of members due to pure shear (in addition to deflection due to ordinary bending theory). To ignore the shear deflection, enter a **SET SHEAR** command before the joint coordinates. This will bring results close to textbook results.

The depths in the two major directions (YD and ZD) are used in the program to calculate the section moduli. These are needed only to calculate member stresses or to perform concrete design. You can omit the YD & ZD values if stresses or design of these members are of no interest. The default value is 253.75 mm (9.99 inches) for YD and ZD . All the prismatic properties are input in the local member coordinates.

Section 1 General Description

1.7 Member Properties

Figure 1-34: Prismatic property nomenclature for a T and Trapezoidal section



To define a concrete member, you must not provide AX, but instead, provide YD and ZD for a rectangular section and just YD for a circular section. If no moment of inertia or shear areas are provided, the program will automatically calculate these from YD and ZD.

Table 1.1 is offered to assist the user in specifying the necessary section values. It lists, by structural type, the required section properties for any analysis. For the PLANE or FLOOR type analyses, the choice of the required moment of inertia depends upon the beta angle. If **BETA** equals zero, the required property is IZ.

Table 1-4: Required Section Properties

Structure Type	Required Properties
TRUSS structure	AX
PLANE structure	AX, IZ, or IY
FLOOR structure	IX, IZ or IY
SPACE structure	AX, IX, IY, IZ

1.7.2 Built-In Steel Section Library

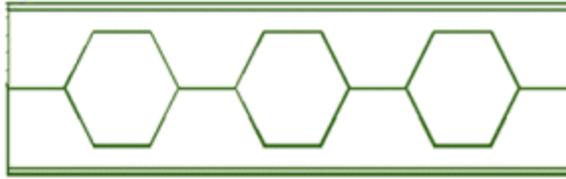
This feature of the program allows you to specify section names of standard steel shapes manufactured in different countries. Information pertaining to the American steel shapes is available in section 2.

For information on steel shapes for other countries, please refer to the International Codes manual.

STAAD.Pro comes with the non-composite castellated beam tables supplied by the steel products manufacturer SMI Steel Products. Details of the manufacture

and design of these sections may be found at
<http://www.smisteelproducts.com/English/About/design.html>

Figure 1-35: Castellated beam elevation



Since the shear areas of the sections are built into the tables, shear deformation is always considered for these sections.

1.7.3 User Provided Steel Table

You can provide a customized steel table with designated names and proper corresponding properties. The program can then find member properties from those tables. Member selection may also be performed with the program selecting members from the provided tables only.

These tables can be provided as a part of a STAAD input or as separately created files from which the program can read the properties. If you do not use standard rolled shapes or only use a limited number of specific shapes, you can create permanent member property files. Analysis and design can be limited to the sections in these files.

See "User Steel Table Specification" on page 326 and See "Property Specification from User Provided Table" on page 350 for additional details.

1.7.4 Tapered Sections

Properties of tapered I-sections may be provided through **MEMBER PROPERTY** specifications. Given key section dimensions, the program is capable of calculating cross-sectional properties which are subsequently used in analysis.

Tapered I-sections have constant flange dimensions and a linearly varying web depth along the length of the member.

See "Tapered Member Specification" on page 349 for details.

1.7.5 Assign Command

If you want to avoid the trouble of defining a specific section name but rather leave it to the program to assign a section name, the **ASSIGN** command may be used. The section types that may be assigned include **BEAM**, **COLUMN**, **CHANNEL**, **ANGLE** and **DOUBLE ANGLE**.

Section 1 General Description

1.7 Member Properties

When the keyword **BEAM** is specified, the program will assign an I-shaped beam section (Wide Flange for AISC, UB section for British).

For the keyword **COLUMN** also, the program will assign an I-shaped beam section (Wide Flange for AISC, UC section for British).

If steel design-member selection is requested, a similar type section will be selected.

See "Assign Profile Specification" on page 350 for details.

1.7.6 Steel Joist and Joist Girders

STAAD.Pro includes facilities for specifying steel joists and joist girders. The basis for this implementation is the information contained in the 1994 publication of the American Steel Joist Institute called "Fortieth edition standard specifications, load tables and weight tables for steel joist and joist girders". The following are the salient features of the implementation.

Member properties can be assigned by specifying a joist designation contained in tables supplied with the program. The following joists and joist girder types have been implemented:

- Open web steel joists – K series and KCS joists
- Longspan steel joists – LH series
- Deep Longspan steel joists – DLH series
- Joist Girders – G series

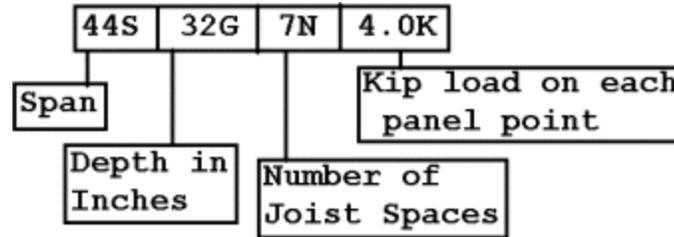
The pages in the Steel Joist Institute publication where these sections are listed are shown in the following table.

Table 1-5: SJI joist types

Joist type	Beginning page number
K series	24
KCS	30
LH series	54
DLH series	57
Joist girders	74

The designation for the G series Joist Girders is as shown in page 73 of the Steel Joist Institute publication. STAAD.Pro incorporates the span length also in the name, as shown in the next figure.

Figure 1-36: STAAD nomenclature for SJI joist girders



Theoretical basis for modeling the joist

Steel joists are prefabricated, welded steel trusses used at closely spaced intervals to support floor or roof decking. Thus, from an analysis standpoint, a joist is not a single member in the same sense as beams and columns of portal frames that one is familiar with. Instead, it is a truss assembly of members. In general, individual manufacturers of the joists decide on the cross section details of the members used for the top and bottom chords, and webs of the joists. So, joist tables rarely contain any information on the cross-section properties of the individual components of a joist girder. The manufacturer's responsibility is to guarantee that, no matter what the cross section details of the members are, the joist simply has to ensure that it provides the capacity corresponding to its rating.

The absence of the section details makes it difficult to incorporate the true truss configuration of the joist in the analysis model of the overall structure. In STAAD, selfweight and any other member load applied on the joist is transferred to its end nodes through simply supported action. Also, in STAAD, the joist makes no contribution to the stiffness of the overall structure.

As a result of the above assumption, the following points must be noted with respect to modeling joists:

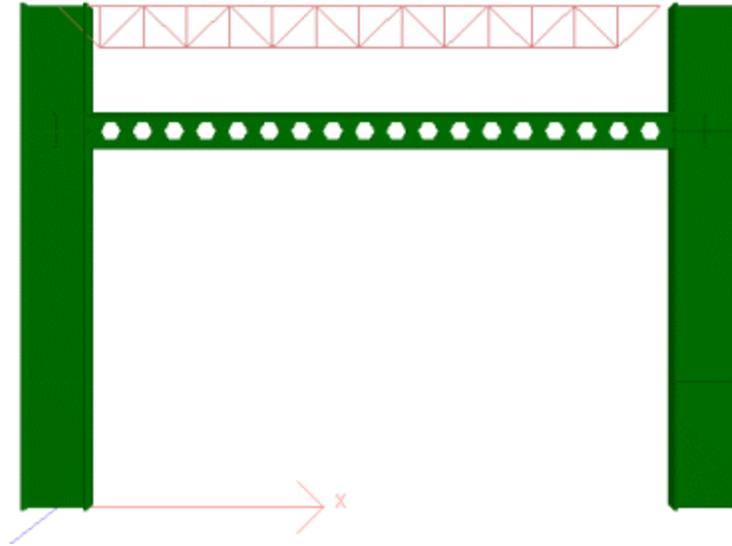
1. The entire joist is represented in the STAAD input file by a single member. Graphically it will be drawn using a single line.
2. After creating the member, the properties should be assigned from the joist database.
3. The 3D Rendering feature of the program will display those members using a representative Warren type truss.

Section 1 General Description

1.7 Member Properties

4. The intermediate span-point displacements of the joist cannot be determined.

Figure 1-37: Example rendering of a joist member in STAAD.Pro



Assigning the joists

The procedure for assigning the joists is explained in the Graphical User Interface manual.

The STAAD joists database includes the weight per length of the joists. So, for selfweight computations in the model, the weight of the joist is automatically considered.

Example

An example of a structure with joist (command file input data) is shown below.

```
STAAD SPACE EXAMPLE FOR JOIST GIRDER
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0
3 30 10 0; 4 30 0 0
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
MEMBER PROPERTY AMERICAN
```

```

1 3 TABLE ST W21X50
MEMBER PROPERTY SJIJOIST
2 TABLE ST 22K6
CONSTANTS
E STEEL ALL
DENSITY STEEL ALL
POISSON STEEL ALL
SUPPORTS
1 4 FIXED
UNIT POUND FEET
LOAD 1
SELFWEIGHT Y -1
LOAD 2
MEMBER LOAD
2 UNI GY -250
LOAD COMB 3
1 1 2 1
PERF ANALY PRINT STAT CHECK
PRINT SUPP REAC
FINISH

```

1.7.7 Composite Beams and Composite Decks

There are two methods in STAAD for specifying composite beams. Composite beams are members whose property is comprised of an I-shaped steel cross section (like an American W shape) with a concrete slab on top. The steel section and concrete slab act monolithically. The two methods are:

1. The EXPLICIT definition method – In this method, the member geometry is first defined as a line. It is then assigned a property from the steel database, with the help of the 'CM' attribute. See "Assigning Properties from Steel Tables" on page 341 for details on using this method. Additional parameters like CT (thickness of the slab), FC (concrete strength), CW (effective width of slab), CD (concrete density), etc., some optional and some mandatory,

Section 1 General Description

1.8 Member/ Element Release

are also provided.

Hence, the responsibility of determining the attributes of the composite member, like concrete slab width, lies upon you, the user. If you wish to obtain a design, additional terms like rib height, rib width, etc. must also be separately assigned with the aid of design parameters. Hence, some amount of effort is involved in gathering all the data and assigning them.

2. The composite deck generation method – The laboriousness of the previous procedure can be alleviated to some extent by using the program's composite deck definition facilities. The program then internally converts the deck into individual composite members (calculating attributes like effective width in the process) during the analysis and design phase. The deck is defined best using the graphical tools of the program since a database of deck data from different manufacturers is accessible from easy-to-use dialogs. Since all the members which make up the deck are identified as part of a single object, load assignment and alterations to the deck can be done to just the deck object, and not the individual members of the deck.

See "Composite Decks" on page 351 for details.

1.7.8 Curved Members

Members can be defined as being curved. Tapered sections are not permitted. The cross-section should be uniform throughout the length.

The design of curved members is not supported.

See "Curved Member Specification" on page 355 for details.

1.8 Member/ Element Release

STAAD allows releases for members and plate elements.

One or both ends of a member or element can be released. Members/Elements are assumed to be rigidly framed into joints in accordance with the structural type specified. When this full rigidity is not applicable, individual force components at either end of the member can be set to zero with member release statements. By specifying release components, individual degrees of freedom are removed from the analysis. Release components are given in the local coordinate system for each member. Note that **PARTIAL** moment release is also allowed.

Only one of sections 1.8 and 1.9 properties can be assigned to a given member. The last one entered will be used. In other words, a **MEMBER RELEASE** should not

1.9 Truss and Tension- or Compression-Only Members

be applied on a member which is declared **TRUSS**, **TENSION ONLY**, or **COMPRESSION ONLY**.

See "Member/Element Releases" on page 370 for details

1.9 Truss and Tension- or Compression-Only Members

For analyses which involve members that carry axial loads only (i.e., truss members) there are two methods for specifying this condition. When all the members in the structure are truss members, the type of structure is declared as **TRUSS** whereas, when only some of the members are truss members (e.g., bracings of a building), the **MEMBER TRUSS** command can be used where those members will be identified separately.

In STAAD, the **MEMBER TENSION** or **MEMBER COMPRESSION** command can be used to limit the axial load type the member may carry.

See "Member Tension/Compression Specification" on page 377 for details.

1.10 Tension, Compression - Only Springs

In STAAD, the **SPRING TENSION** or **SPRING COMPRESSION** command can be used to limit the load direction the support spring may carry. The analysis will be performed accordingly.

See "Axial Member Specifications" on page 375 for details.

1.11 Cable Members

STAAD supports two types of analysis for cable members:

1.11.1 Linearized Cable Members

Cable members may be specified by using the **MEMBER CABLE** command. While specifying cable members, the initial tension in the cable must be provided. The following paragraph explains how cable stiffness is calculated.

The increase in length of a loaded cable is a combination of two effects. The first component is the elastic stretch, and is governed by the familiar spring relationship:

$$F = Kx$$

Where:

$$K_{\text{elastic}} = EA/L$$

The second component of the lengthening is due to a change in geometry (as a cable is pulled taut, sag is reduced). This relationship can be described by

Section 1 General Description

1.11 Cable Members

$$F = Kx$$

but here,

$$K_{sag} = \frac{12T^3}{w^2L^3\left(\frac{1}{\cos^2\alpha}\right)}$$

Where:

w = weight per unit length of cable

T = tension in cable

α = angle that the axis of the cable makes with a horizontal plane (= 0, cable is horizontal; = 90, cable is vertical).

Therefore, the "stiffness" of a cable depends on the initial installed tension (or sag). These two effects may be combined as follows:

$$K_{comb} = \frac{1}{\left(\frac{1}{K_{sag}} + \frac{1}{K_{elastic}}\right)} = \frac{(EA/L)}{\left(1 + \frac{w^2L^2EA(\cos^2\alpha)}{12T^3}\right)}$$

Note: When T = infinity, $K_{comb} = EA/L$ and that when T = 0, $K_{comb} = 0$. It should also be noted that as the tension increases (sag decreases) the combined stiffness approaches that of the pure elastic situation.

The following points need to be considered when using the cable member in STAAD :

1. The linear cable member is only a truss member whose properties accommodate the sag factor and initial tension. The behavior of the cable member is identical to that of the truss member. It can carry axial loads only. As a result, the fundamental rules involved in modeling truss members have to be followed when modeling cable members. For example, when two cable members meet at a common joint, if there isn't a support or a 3rd member connected to that joint, it is a point of potential instability.
2. Due to the reasons specified in 1) above, applying a transverse load on a cable member is not advisable. The load will be converted to two concentrated loads at the 2 ends of the cable and the true deflection pattern of the cable will never be realized.
3. A tension only cable member offers no resistance to a compressive force applied at its ends. When the end joints of the member are subjected to a compressive force, they "give in" thereby causing the cable to sag. Under

these circumstances, the cable member has zero stiffness and this situation has to be accounted for in the stiffness matrix and the displacements have to be recalculated. But in STAAD, merely declaring the member to be a cable member does not guarantee that this behavior will be accounted for. It is also important that you declare the member to be a tension only member by using the **MEMBER TENSION** command, after the **CABLE** command. This will ensure that the program will test the nature of the force in the member after the analysis and if it is compressive, the member is switched off and the stiffness matrix re-calculated.

4. Due to potential instability problems explained in item 1 above, you should also avoid modeling a catenary by breaking it down into a number of straight line segments. The cable member in STAAD cannot be used to simulate the behavior of a catenary. By catenary, we are referring to those structural components which have a curved profile and develop axial forces due their self weight. This behavior is in reality a non-linear behavior where the axial force is caused because of either a change in the profile of the member or induced by large displacements, neither of which are valid assumptions in an elastic analysis. A typical example of a catenary is the main U shaped cable used in suspension bridges.
5. The increase of stiffness of the cable as the tension in it increases under applied loading is updated after each iteration if the cable members are also declared to be **MEMBER TENSION**. However, iteration stops when all tension members are in tension or slack; not when the cable tension converges.

See "Second Order Analysis" on page 67, See "Axial Member Specifications" on page 375, and See "Analysis Specification" on page 676 for details.

1.11.2 Nonlinear Cable and Truss Members

Cable members for the Nonlinear Cable Analysis may be specified by using the **MEMBER CABLE** command. While specifying cable members, the initial tension in the cable or the unstressed length of the cable must be provided. you should ensure that all cables will be in sufficient tension for all load cases to converge. Use selfweight in every load case and temperature if appropriate; i.e., don't enter component cases (e.g., wind only).

The nonlinear cable may have large motions and the sag is checked on every load step and every equilibrium iteration.

In addition there is a nonlinear truss which is specified in the Member Truss command. The nonlinear truss is simply any truss with pretension specified. It is

Section 1 General Description

1.12 Member Offsets

essentially the same as a cable without sag. This member takes compression. If all cables are taut for all load cases, then the nonlinear truss may be used to simulate cables. The reason for using this substitution is that the truss solution is more reliable.

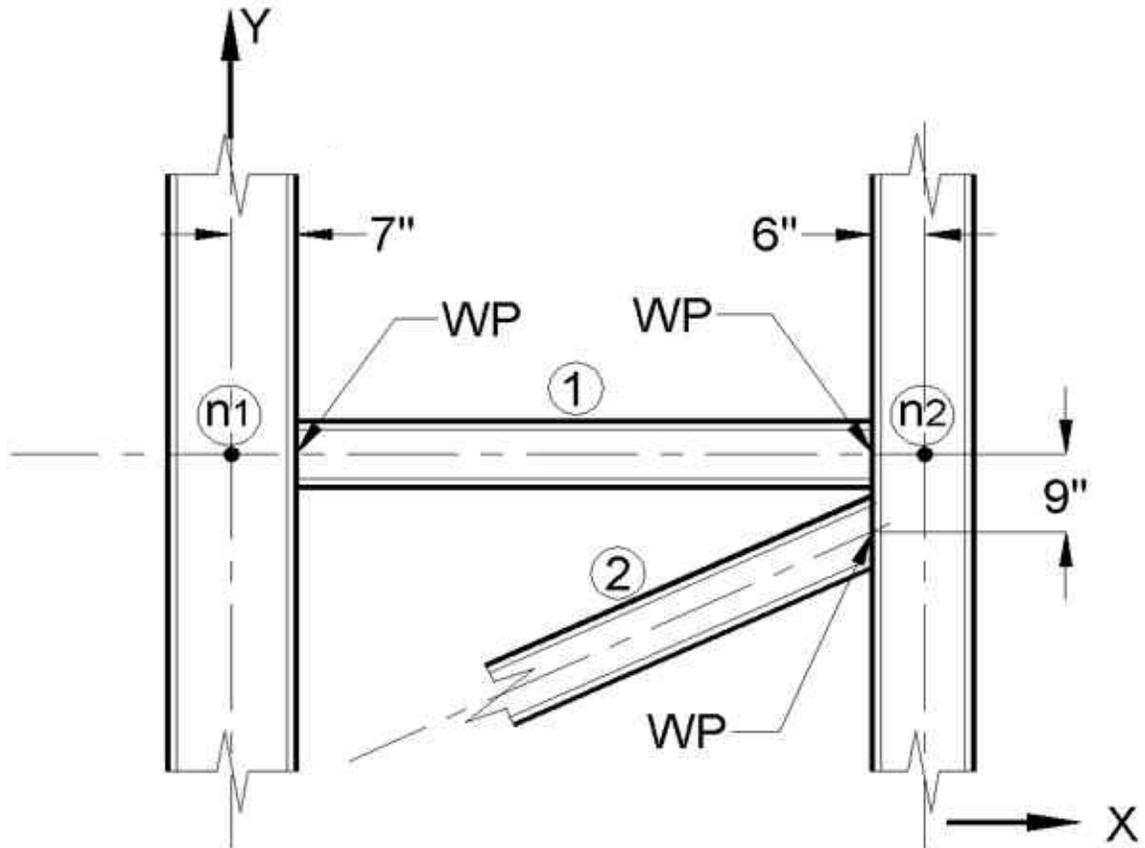
Points 1, 2, and 4 in the previous section will not apply to nonlinear cable analysis if sufficient pretension is applied, so joints may be entered along the shape of a cable (in some cases a stabilizing stiffness may be required and entered for the first loadstep). Point 3 above: The Member Tension command is unnecessary and ignored for the nonlinear cable analysis. Point 5 above: The cable tensions are iterated to convergence in the nonlinear cable analysis.

See "Nonlinear Cable/Truss Analysis" on page 78, See "Axial Member Specifications" on page 375, and See "Nonlinear Cable Analysis" on page 683 for details.

1.12 Member Offsets

Some members of a structure may not be concurrent with the incident joints thereby creating offsets. This offset distance is specified in terms of global or local coordinate system (i.e., X, Y and Z distances from the incident joint). Secondary forces induced, due to this offset connection, are taken into account in analyzing the structure and also to calculate the individual member forces. The new offset centroid of the member can be at the start or end incidences and the new working point will also be the new start or end of the member. Therefore, any reference from the start or end of that member will always be from the new offset points.

Figure 1-38: Example of working points (WP)



In the figure above, WP refers to the location of the centroid of the starting or ending point of the member.

Example

MEMBER OFFSET

1 START 7

1 END -6

2 END -6 -9

MEMBER OFFSET

1 START 7

1 END -6

2 END -6 -9

See "Member Offset Specification" on page 383 for details.

Section 1 General Description

1.13 Material Constants

1.13 Material Constants

The material constants are: modulus of elasticity (**E**); weight density (**DEN**); Poisson's ratio (**POISS**); co-efficient of thermal expansion (**ALPHA**), Composite Damping Ratio, and beta angle (**BETA**) or coordinates for any reference (**REF**) point.

E value for members must be provided or the analysis will not be performed. Weight density (**DEN**) is used only when selfweight of the structure is to be taken into account. Poisson's ratio (**POISS**) is used to calculate the shear modulus (commonly known as G) by the formula,

$$G = 0.5 \cdot E / (1 + \text{POISS})$$

If Poisson's ratio is not provided, STAAD will assume a value for this quantity based on the value of E. Coefficient of thermal expansion (**ALPHA**) is used to calculate the expansion of the members if temperature loads are applied. The temperature unit for temperature load and **ALPHA** has to be the same.

Composite damping ratio is used to compute the damping ratio for each mode in a dynamic solution. This is only useful if there are several materials with different damping ratios.

BETA angle and **REF**erence point are discussed in Section 1.5.3 and are input as part of the member constants.

Note: Poisson's Ratio must always be defined after the Modulus of Elasticity for a given member/element.

1.14 Supports

STAAD allows specifications of supports that are parallel as well as inclined to the global axes.

Supports are specified as **PINNED**, **FIXED**, or **FIXED** with different releases. A pinned support has restraints against all translational movement and none against rotational movement. In other words, a pinned support will have reactions for all forces but will resist no moments. A fixed support has restraints against all directions of movement.

The restraints of a fixed support can also be released in any desired direction as specified. See "Global Support Specification" on page 405 for details.

Translational and rotational springs can also be specified. The springs are represented in terms of their spring constants. A translational spring constant is

defined as the force to displace a support joint one length unit in the specified global direction. Similarly, a rotational spring constant is defined as the force to rotate the support joint one degree around the specified global direction. See "Multilinear Spring Support Specification" on page 414 for details.

For static analysis, Multi-linear spring supports can be used to model the varying, non-linear resistance of a support (e.g., soil). See "Automatic Spring Support Generator for Foundations" on page 409 for descriptions of the elastic footing and elastic foundation mat facilities.

The Support command is also used to specify joints and directions where support displacements will be enforced.

1.15 Master/Slave Joints

The master/slave option is provided to enable the user to model rigid links in the structural system. This facility can be used to model special structural elements like a rigid floor diaphragm. Several slave joints may be provided which will be assigned same displacements as the master joint. The user is also allowed the flexibility to choose the specific degrees of freedom for which the displacement constraints will be imposed on the slaved joints. If all degrees of freedom (F_x , F_y , F_z , M_x , M_y and M_z) are provided as constraints, the joints will be assumed to be rigidly connected.

See "Master/Slave Specification" on page 420 for details.

1.16 Loads

Loads in a structure can be specified as joint load, member load, temperature load and fixed-end member load. STAAD can also generate the self-weight of the structure and use it as uniformly distributed member loads in analysis. Any fraction of this self-weight can also be applied in any desired direction.

1.16.1 Joint Load

Joint loads, both forces and moments, may be applied to any free joint of a structure. These loads act in the global coordinate system of the structure. Positive forces act in the positive coordinate directions. Any number of loads may be applied on a single joint, in which case the loads will be additive on that joint.

See "Joint Load Specification" on page 540 for details.

Section 1 General Description

1.16 Loads

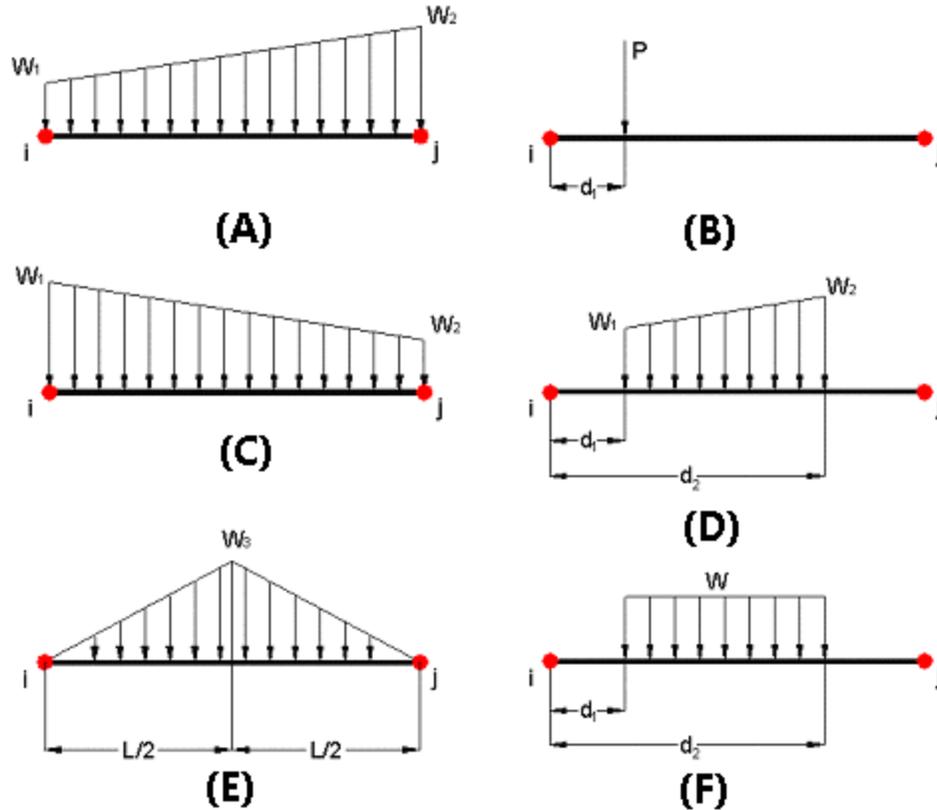
1.16.2 Member Load

Three types of member loads may be applied directly to a member of a structure. These loads are uniformly distributed loads, concentrated loads, and linearly varying loads (including trapezoidal). Uniform loads act on the full or partial length of a member. Concentrated loads act at any intermediate, specified point. Linearly varying loads act over the full length of a member. Trapezoidal linearly varying loads act over the full or partial length of a member. Trapezoidal loads are converted into a uniform load and several concentrated loads.

Any number of loads may be specified to act upon a member in any independent loading condition. Member loads can be specified in the member coordinate system or the global coordinate system. Uniformly distributed member loads provided in the global coordinate system may be specified to act along the full or projected member length. See "Global Coordinate System" on page 10 to find the relation of the member to the global coordinate systems for specifying member loads. Positive forces act in the positive coordinate directions, local or global, as the case may be.

Uniform moment may not be applied to tapered members. Only uniform load over the entire length is available for curved members.

Figure 1-39: Member Load Configurations for A) linear loads, B) concentrated loads, C) linear loads, D) trapezoidal load, E) triangular (linear) loads, and E) uniform load



See "Member Load Specification" on page 541 for details.

1.16.3 Area Load, One-way, and Floor Loads

Often a floor is subjected to a uniform pressure. It could require a lot of work to calculate the equivalent member load for individual members in that floor. However, with the **AREA**, **ONEWAY** or **FLOOR LOAD** facilities, you can specify the pressure (load per unit square area). The program will calculate the tributary area for these members and calculate the appropriate member loads. The Area Load and Oneway load are used for one way distribution and the Floor Load is used for two way distribution.

The following assumptions are made while transferring the area/floor load to member load:

- The member load is assumed to be a linearly varying load for which the start and the end values may be of different magnitude.
- Tributary area of a member with an area load is calculated based on half the spacing to the nearest approximately parallel members on both sides. If the spacing is more than or equal to the length of the member, the area load will be ignored. Oneway load does not have this limitation.

Section 1 General Description

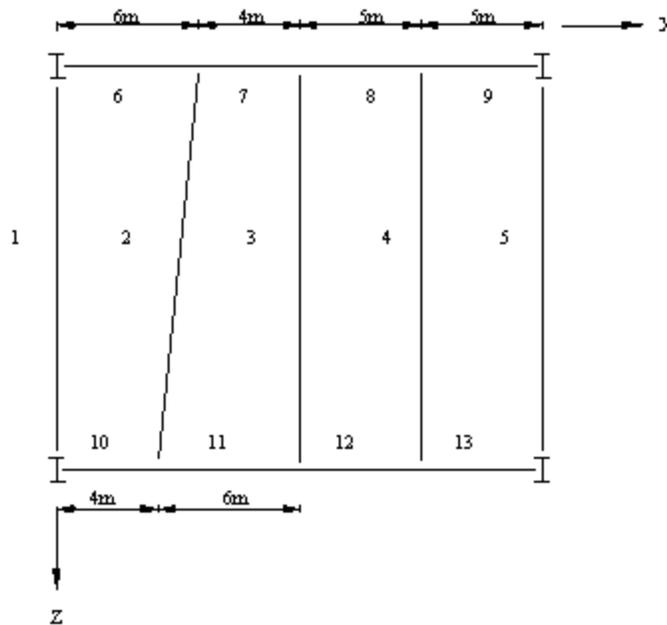
1.16 Loads

- c. These loading types should not be specified on members declared as **MEMBER CABLE**, **MEMBER TRUSS**, **MEMBER TENSION**, **MEMBER COMPRESSION**, or **CURVED**.

Note: Floor Loads and One-way Loads can be reduced when included in a load case defined as “Reducible” according to the UBC/IBC rules.

An example:

Figure 1-40: Example floor structure with area load specification of 0.1



Member 1 will have a linear load of 0.3 at one end and 0.2 at the other end. Members 2 and 4 will have a uniform load of 0.5 over the full length. Member 3 will have a linear load of 0.45 and 0.55 at respective ends. Member 5 will have a uniform load of 0.25. The rest of the members, 6 through 13, will have no contributory area load since the nearest parallel members are more than each of the member lengths apart. However, the reactions from the members to the girder will be considered.

Only member loads are generated from the Area, Oneway and Floor load input. Thus, load types specific to plates, solids or surface are not generated. That is because, the basic assumption is that, a floor load or area load is used in situations where the basic entity (plate, solid or surface) which acts as the medium for application of that load, is not part of the structural model.

See "Area, One-way, and Floor Load Specifications" on page 554 for details.

1.16.4 Fixed End Member Load

Load effects on a member may also be specified in terms of its fixed end loads. These loads are given in terms of the member coordinate system and the directions are opposite to the actual load on the member. Each end of a member can have six forces: axial; shear y; shear z; torsion; moment y, and moment z.

See "Fixed-End Load Specification" on page 579 for details.

1.16.5 Prestress and Poststress Member Load

Members in a structure may be subjected to prestress load for which the load distribution in the structure may be investigated. The prestressing load in a member may be applied axially or eccentrically. The eccentricities can be provided at the start joint, at the middle, and at the end joint. These eccentricities are only in the local y-axis. A positive eccentricity will be in the positive local y-direction. Since eccentricities are only provided in the local y-axis, care should be taken when providing prismatic properties or in specifying the correct **BETA** angle when rotating the member coordinates, if necessary. Two types of prestress load specification are available; **PRESTRESS**, where the effects of the load are transmitted to the rest of the structure, and **POSTSTRESS**, where the effects of the load are experienced exclusively by the members on which it is applied.

1. The cable is assumed to have a generalized parabolic profile. The equation of the parabola is assumed to be

$$y = ax^2 + bx + c$$

Where:

$$a = 1/L^2 (2es - 4em + 2ee)$$

$$b = 1/L (4em - ee - 3es)$$

$$c = es$$

es = eccentricity of the cable at the start of the member (in local y-axis)

em = eccentricity of the cable at the middle of the member (in local y-axis)

ee = eccentricity of the cable at the end of the member (in local y-axis)

L = Length of the member.

Section 1 General Description

1.16 Loads

- The angle of inclination of the cable with respect to the local x-axis (a straight line joining the start and end joints of the member) at the start and end points is small which gives rise to the assumption that

$$\sin\theta = \theta = dy/dx$$

Hence, if the axial force in the cable is P, the vertical component of the force at the ends is P(dy/dx) and the horizontal component of the cable force is,

$$P[1 - (dy/dx)^2]^{1/2}$$

Users are advised to ensure that their cable profile meets this requirement. An angle under 5 degrees is recommended.

- The member is analyzed for the prestressing / poststressing effects using the equivalent load method. This method is well documented in most reputed books on Analysis and Design of Prestressed concrete. The magnitude of the uniformly distributed load is calculated as

$$udl = 8 \cdot Pe/L^2$$

Where:

P = Axial force in the cable

e = (es + ee)/2 - em

L = Length of the member

- The force in the cable is assumed to be same throughout the member length. No reduction is made in the cable forces to account for friction or other losses.
- The term **MEMBER PRESTRESS** as used in STAAD signifies the following condition. The structure is constructed first. Then, the prestressing force is applied on the relevant members. As a result, the members deform and depending on their end conditions, forces are transmitted to other members in the structure. In other words, "PRE" refers to the time of placement of the member in the structure relative to the time of stressing.
- The term **MEMBER POSTSTRESS** as used in STAAD signifies the following condition. The members on which such load is applied are first cast in the factory. Following this, the prestressing force is applied on them. Meanwhile, the rest of the structure is constructed at the construction site. Then, the prestressed members are brought and placed in position on the partially built structure. Due to this sequence, the effects of prestressing are

"experienced" by only the prestressed members and not transmitted to the rest of the structure. In other words, "POST" refers to the time of placement of the member in the structure relative to the time of stressing.

7. As may be evident from Item (6) above, it is not possible to compute the displacements of the ends of the **POSTSTRESSED** members for the effects of poststressing, and hence are assumed to be zero. As a result, displacements of intermediate sections (See SECTION DISPLACEMENT command) are measured relative to the straight line joining the start and end joints of the members as defined by their initial **JOINT COORDINATES**.

See "Prestress Load Specification" on page 573 for details.

1.16.6 Temperature and Strain Load

Uniform temperature difference throughout members and elements may be specified. Temperature differences across both faces of members and through the thickness of plates may also be specified (uniform temperature only for solids).. The program calculates the axial strain (elongation and shrinkage) due to the temperature difference for members. From this it calculates the induced forces in the member and the analysis is done accordingly. The strain intervals of elongation and shrinkage can be input directly.

See "Temperature Load Specification for Members, Plates, and Solids" on page 578 for details.

1.16.7 Support Displacement Load

Static Loads can be applied to the structure in terms of the displacement of the supports. Displacement can be translational or rotational. Translational displacements are provided in the specified length while the rotational displacements are always in degrees. Note that displacements can be specified only in directions in which the support has an "enforced" specification in the Support command.

See "Support Joint Displacement Specification" on page 580 for details.

1.16.8 Loading on Elements

On Plate/Shell elements, the types of loading that are permissible are:

1. Pressure loading which consists of loads which act perpendicular to the surface of the element. The pressure loads can be of uniform intensity or trapezoidally varying intensity over a small portion or over the entire

Section 1 General Description

1.17 Load Generator

surface of the element.

2. Joint loads which are forces or moments that are applied at the joints in the direction of the global axes.
3. Temperature loads which may be constant throughout the plate element (causing only elongation / shortening) or may vary across the depth of a plate element causing bending on the plate element.. The coefficient of thermal expansion for the material of the element must be provided in order to facilitate computation of these effects.
4. The self-weight of the elements can be applied using the **SELFWEIGHT** loading condition. The density of the elements has to be provided in order to facilitate computation of the self-weight.

On Solid elements, the loading types available are:

1. The self-weight of the solid elements can be applied using the **SELFWEIGHT** loading condition. The density of the elements has to be provided in order to facilitate computation of the self-weight.
2. Joint loads which are forces or moments that are applied at the joints in the direction of the global axes.
3. Temperature loads which may be constant throughout the solid elements (causing only elongation / shortening). The coefficient of thermal expansion for the material of the element must be provided in order to facilitate computation of these effects.
4. Pressure on the faces of solids.

Only translational stiffness is supported in solid elements. Thus, at joints where there are only solid elements, moments may not be applied. For efficiency, rotational supports should be used at these joints.

See "Element Load Specifications" on page 544 for details.

1.17 Load Generator

Load generation is the process of taking a load causing unit such as wind pressure, ground movement or a truck on a bridge, and converting it to a form such as member load or a joint load which can be then be used in the analysis.

For seismic loads, a static analysis method or a dynamic analysis method can be adopted. The static analysis method, which is the one referred to here, is based on codes such as UBC, IBC, AIJ, IS1893 etc. For dynamic analysis, see the sections in this chapter on response spectrum and time history analysis.

Input for the load generation facility consists of two parts:

1. Definition of the load system(s).
2. Generation of primary load cases using previously defined load system(s).

1.17.1 Moving Load Generator

This feature enables the user to generate static loads on members due to vehicles moving on a structure. Moving load system(s) consisting of concentrated loads at fixed specified distances in both directions on a plane can be defined by the user. A user specified number of primary load cases will be subsequently generated by the program and taken into consideration in analysis. American Association of State Highway and Transportation Officials (AASHTO) vehicles are available within the program and can be specified using standard AASHTO designations.

See "Definition of Moving Load System" on page 429 and See "Generation of Moving Loads" on page 651 for details.

1.17.2 Seismic Load Generator

The STAAD seismic load generator follows the procedure of equivalent lateral load analysis explained in UBC, IBC and several other codes. It is assumed that the lateral loads will be exerted in X and Z (or X and Y if Z is up) directions (horizontal) and Y (or Z if Z is up) will be the direction of the gravity loads. Thus, for a building model, Y (or Z if Z is up) axis will be perpendicular to the floors and point upward (all Y (or Z if Z is up) joint coordinates positive). The user is required to set up his model accordingly. Total lateral seismic force or base shear is automatically calculated by STAAD using the appropriate equation from the code. IBC 2003, IBC 2000, UBC 1997, 1994, or 1985, IS:1893, Japanese, Colombian and other specifications may be used.

For load generation per the codes, the user is required to provide seismic zone coefficients, importance factors, soil characteristic parameters, etc. See "UBC 1997 Load Definition" on page 434 for the detailed input required for each code.

Instead of using the approximate code based formulas to estimate the building period in a certain direction, the program calculates the period using Rayleigh quotient technique. This period is then utilized to calculate seismic coefficient C .

After the base shear is calculated from the appropriate equation, it is distributed among the various levels and roof per UBC specifications. The distributed base shears are subsequently applied as lateral loads on the structure. These loads may then be utilized as normal load cases for analysis and design.

See "Generation of Seismic Loads" on page 653 for details.

Section 1 General Description

1.17 Load Generator

1.17.3 Wind Load Generator

The Wind Load Generator is a utility which takes as input wind pressure and height ranges over which these pressures act and generates nodal point and member loads.

This facility is available for two types of structures.

- Panel type or Closed structures
- Open structures

Closed structures are ones like office buildings where non-structural entities like a glass facade, aluminum sheets, timber panels or non-load bearing walls act as an obstruction to the wind. If these entities are not included in the structural model, the load generated as a result of wind blowing against them needs to be computed. So, the steps involved in load generation for such structures are:

- i. Identify the panels – regions circumscribed by members so that a polygonal closed area is formed. The area may also be formed between the ground level along one edge and members along the other.
- ii. Calculate the panel area and multiply it by the wind pressure.
- iii. Convert the resulting force into nodal point loads.

Plates and solids are not considered in the calculation of the panel area. Openings within the panels may be modeled with the help of exposure factors. An exposure factor is associated with each joint of the panel and is a fractional number by which the area affecting a joint of the panel can be reduced or increased.

The automated load generator should only be used for vertical panels. Panels not parallel to the global Y axis (for Y UP) should be loaded separately.

Open structures are those like transmission towers, in which the region between members is “open” allowing the wind to blow through. The procedure for load generation for open structures is i) Calculate the exposed area of the individual members of the model. ii) Multiply that exposed area by the wind pressure to arrive at the force and apply the force on individual members as a uniformly distributed load. It is assumed that all members of the structure within the specified ranges are subjected to the pressure and hence, they will all receive the load. The concept of members on the windward side shielding the members in the inside regions of the structure does not exist for open structures. Members loaded as an open structure need not be vertical.

See "Definition of Wind Load" on page 503 and See "Generation of Wind Loads" on page 660 for details.

1.17.4 Snow Load

STAAD.Pro is capable of generating snow loading on a structure in accordance with the provisions of the ASCE-7-02 code. The feature is currently implemented for structures with flat or sloping roofs. Snow load generation for members of open lattice structures like electrical transmission towers is currently not part of this facility. Hence, the feature is based on panel areas, not the exposed width of individual members.

See "Definition of Snow Load" on page 530 and See "Generation of Snow Loads" on page 664 for details.

1.18 Analysis Facilities

Salient features of each type of analysis are discussed in the following sections. Detailed theoretical treatments of these features are available in standard structural engineering textbooks.

1.18.1 Stiffness Analysis

The stiffness analysis implemented in STAAD is based on the matrix displacement method. In the matrix analysis of structures by the displacement method, the structure is first idealized into an assembly of discrete structural components.

Structural systems such as slabs, plates, spread footings, etc., which transmit loads in two directions (frame members or finite elements). Each component has an assumed form of displacement in a manner which satisfies the force equilibrium and displacement compatibility at the joints. have to be discretized into a number of three or four noded finite elements connected to each other at their nodes. Loads may be applied in the form of distributed loads on the element surfaces or as concentrated loads at the joints. The plane stress effects as well as the plate bending effects are taken into consideration in the analysis.

Assumptions of the Analysis

For a complete analysis of the structure, the necessary matrices are generated on the basis of the following assumptions:

1. The structure is idealized into an assembly of beam, plate and solid type elements joined together at their vertices (nodes). The assemblage is loaded and reacted by concentrated loads acting at the nodes. These loads may be

Section 1 General Description

1.18 Analysis Facilities

both forces and moments which may act in any specified direction.

2. A beam member is a longitudinal structural member having a constant, doubly symmetric or near-doubly symmetric cross section along its length. Beam members always carry axial forces. They may also be subjected to shear and bending in two arbitrary perpendicular planes, and they may also be subjected to torsion. From this point these beam members are referred to as "members" in the manual.
3. A plate element is a three or four noded planar element having variable thickness. A solid element is a four-to-eight-noded, three dimensional element. These plate and solid elements are referred to as "elements" in the manual.
4. Internal and external loads acting on each node are in equilibrium. If torsional or bending properties are defined for any member, six degrees of freedom are considered at each node (i.e., three translational and three rotational) in the generation of relevant matrices. If the member is defined as truss member (i.e., carrying only axial forces) then only the three degrees (translational) of freedom are considered at each node.
5. Two types of coordinate systems are used in the generation of the required matrices and are referred to as local and global systems.

Local coordinate axes are assigned to each individual element and are oriented such that computing effort for element stiffness matrices are generalized and minimized. Global coordinate axes are a common datum established for all idealized elements so that element forces and displacements may be related to a common frame of reference.

Basic Equation

The complete stiffness matrix of the structure is obtained by systematically summing the contributions of the various member and element stiffness. The external loads on the structure are represented as discrete concentrated loads acting only at the nodal points of the structure.

The stiffness matrix relates these loads to the displacements of the nodes by the equation:

$$A_j = a_j + S_j \cdot D_j$$

This formulation includes all the joints of the structure, whether they are free to displace or are restrained by supports. Those components of joint displacements that are free to move are called degrees of freedom. The total number of degrees of freedom represent the number of unknowns in the analysis.

Method to Solve for Displacements

There are many methods to solve the unknowns from a series of simultaneous equations.

In STAAD.Pro, the element stiffness matrices are assembled into a global stiffness matrix by standard matrix techniques used in FEA programs. The technique used by STAAD was copied from SAP IV. The global stiffness matrix is then decomposed as

$$[K] = [L^T] [D] [L]$$

which is a modified Gauss method.

$$[K] \{d\} = \{F\}$$

becomes

$$[L^T] [D] [L] \{d\} = \{F\}$$

which can be manipulated into a forward and a backward substitution step to obtain $\{d\}$. STAAD can detect singular matrices and solve then via a technique copied from Stardyne.

Basic Solver

An approach which is particularly suited for structural analysis is called the method of decomposition. This method has been selected for use in STAAD. Since the stiffness matrices of all linearly elastic structures are always symmetric, an especially efficient form of the decomposition called Modified Cholesky's method may be applied to these problems. This method is reasonably accurate and well suited for the Gaussian elimination process in solving the simultaneous equations.

Advanced Solver

(Available effective 2007 Build 01): An approach is used that is mathematically equivalent to the modified Choleski method. However the order of operations, memory use, and file use is highly optimized. Run times are often 10 to 100 (even 1000) times faster.

Consideration of Bandwidth

For the Basic Solver only. The method of decomposition is particularly efficient when applied to a symmetrically banded matrix. For this type of matrix fewer calculations are required due to the fact that elements outside the band are all equal to zero.

Section 1 General Description

1.18 Analysis Facilities

STAAD takes full advantage of this bandwidth during solution, as it is important to have the least bandwidth to obtain the most efficient solution. For this purpose, STAAD offers features by which the program can internally rearrange the joint numbers to provide a better bandwidth.

For the Advanced Solver only. Internal storage order is automatically calculated to minimize time and memory.

Multiple Structures & Structural Integrity

The integrity of the structure is a very important requirement that must be satisfied by all models. You must make sure that the model developed represents one or more properly connected structures.

An "integral" structure may be defined as a system in which proper "stiffness connections" exist between the members/elements. The entire model functions as one or more integrated load resisting systems. STAAD checks structural integrity using a sophisticated algorithm and reports detection of multiple structures within the model. If you did not intend for there to be multiple structures, then you can fix it before any analysis. There are several additional model checking options within the **Tools** and **Geometry** menus.

Modeling and Numerical Instability Problems

Instability problems can occur due to two primary reasons.

1. Modeling problem

There are a variety of modeling problems which can give rise to instability conditions. They can be classified into two groups.

- a. Local instability - A local instability is a condition where the fixity conditions at the end(s) of a member are such as to cause an instability in the member about one or more degrees of freedom. Examples of local instability are:
 - i. Member Release: Members released at both ends for any of the following degrees of freedom (FX, FY, FZ and MX) will be subjected to this problem.
 - ii. A framed structure with columns and beams where the columns are defined as "TRUSS" members. Such a column has no capacity to transfer shears or moments from the superstructure to the supports.

- b. Global Instability - These are caused when the supports of the structure are such that they cannot offer any resistance to sliding or overturning of the structure in one or more directions. For example, a 2D structure (frame in the XY plane) which is defined as a SPACE FRAME with pinned supports and subjected to a force in the Z direction will topple over about the X-axis. Another example is that of a space frame with all the supports released for FX, FY or FZ.
2. Math precision

A math precision error is caused when numerical instabilities occur in the matrix inversion process. One of the terms of the equilibrium equation takes the form $1/(1-A)$, where $A=k_1/(k_1+k_2)$; k_1 and k_2 being the stiffness coefficients of two adjacent members. When a very "stiff" member is adjacent to a very "flexible" member, viz., when $k_1 \gg k_2$, or $k_1+k_2 \approx k_1$, $A=1$ and hence, $1/(1-A) = 1/0$. Thus, huge variations in stiffnesses of adjacent members are not permitted. Artificially high E or I values should be reduced when this occurs.

Math precision errors are also caused when the units of length and force are not defined correctly for member lengths, member properties, constants etc.

Users also have to ensure that the model defined represents one single structure only, not two or more separate structures. For example, in an effort to model an expansion joint, you may end up defining separate structures within the same input file. Multiple structures defined in one input file can lead to grossly erroneous results.

See "Analysis Specification" on page 676 for details.

1.18.2 Second Order Analysis

STAAD offers the capability to perform second order stability analyses.

See "Analysis Specification" on page 676 for details.

1.18.2.1 P-Delta Analysis

Structures subjected to lateral loads often experience secondary forces due to the movement of the point of application of vertical loads. This secondary effect, commonly known as the P-Delta effect, plays an important role in the analysis of the structure.

In textbooks this secondary effect is typically referred to as stress stiffening for members in tension (or softening for compression). The stiffness changes due to

Section 1 General Description

1.18 Analysis Facilities

P-Delta are known as geometric stiffness, [Kg]. There are two types of P-Delta effects for members. $P-\Delta$ which is due to the displacement of one end of a member relative to the other end (e.g., story drift of column members). A second effect is $P-\delta$ which is due to the bending of the member.

$P-\delta$ due to the bending of the member not only affects the local & global stiffness, nodal displacements, and member end forces; it also has an additional effect on the section displacements and section moments. The (axial compressive member force) times (the local relative to the ends section displacement) gives a section moment in addition to the flexural moment. This additional section moment will cause an additional sectional displacement; and so on. Normally this process will converge after 5-20 iterations if the member buckling load is not exceeded. STAAD uses up to 20 iterations unless convergence or divergence occurs.

$P-\delta$ due to the bending of the member can also occur with tension if the member has sufficient bending. STAAD only iterates once for tension.

STAAD does not include the effects of geometric stiffness for solids. If the part of the structure that deforms involves non-trivial motions of solids, then the results will be erroneous for P-Delta analysis (as well as for buckling analysis).

1.18.2.1.1 Large Delta and Small Delta

In STAAD, a procedure has been adopted to incorporate the P-Delta effect into the analysis without re-forming and factorizing the global stiffness matrix on each iteration. Actually, only the global stiffness matrix is formed and factorized; which must be done for any analysis. Only the relatively fast forward and backward substitution step for typically five to 25 iterations must be performed. This step is done simultaneously for however many cases are being solved. See section 1.18.2.1.2 for an alternate formulation of P-Delta that may be used in dynamics.

Note: This feature is available in STAAD.Pro 2007 Build 01 and greater.

If a structure is heavily loaded it may become unstable for some load cases. It may take 10 to 30 iterations for this instability to become obvious by the maximum displacements or bending moment envelope values becoming very large or infinite or reported as NaN.

The procedure consists of the following steps:

1. First, the primary deflections are calculated based on the provided external loading.

2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. By default the small delta effects are calculated. To include only the large delta effects, enter the **LARGEDELTA** option on the **PDELTA** command. These forces and stresses are used to calculate geometric stiffness terms. These terms times the displacement results from the prior iteration create the P-Delta secondary loading. This secondary loading is then combined with the originally applied loading to create the effective load vector for the next iteration.

The lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. The **REPEAT LOAD** facility (see Section 5.32.11) has been created with this requirement in mind. This facility allows you to combine previously defined primary load cases to create a new primary load case.

3. The revised load vector is used with the static triangular factorized matrix to generate new deflections.
4. Element/Member forces and support reactions are calculated based on the new deflections.

Repeat steps 2 to 4 for several iterations. Three to 30 iterations are recommended. This procedure yields reasonably accurate results with small displacement problems. You are allowed to specify the number of iterations. If the Converged option is used, then set the displacement convergence tolerance by entering a **SET PDELTATOL i_g** command before the Joint Coordinates. If all changes in displacement dof from one iteration to the next is less than the specified tolerance value, i_g , then that case is converged.

The P-Delta analysis is recommended by several design codes such as ACI 318, LRFD, IS456-1978, etc. in lieu of the moment magnification method for the calculation of more realistic forces and moments.

P-Delta effects are calculated for frame members and plate elements only. They are not calculated for solid elements. P-Delta has the most effect in structures where there are vertical and horizontal loads in the same load case.

The maximum displacement should be reviewed for P-Delta analyses because this analysis type permits large buckling displacements if the loads make the structure unstable. You may need to repeat the analysis with only one to five iterations in order to get a pre-collapse solution in order to view the large displacement areas.

Section 1 General Description

1.18 Analysis Facilities

The section moment due to tension and the section displacements due to shear/bending are added to the moment diagram, if small delta is selected. This is no iteration performed for this step.

1.18.2.1.2 P-Delta Kg Analysis

In STAAD, an alternate procedure has been adopted to incorporate the P-Delta effect into the analysis by combining the global stiffness matrix and the global geometric stiffness matrix $[K+Kg]$.

Note: This feature is available in STAAD.Pro 2007 Build 01 and greater.

1. First, the primary deflections are calculated by linear static analysis based on the provided external loading.
2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These terms are the terms of the Kg matrix which are added to the global stiffness matrix K.

The lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. The **REPEAT LOAD** facility (see Section 5.32.11) has been created with this requirement in mind. This facility allows the user to combine previously defined primary load cases to create a new primary load case.

This procedure yields reasonably accurate results with small displacement problems. STAAD allows the user to specify multiple iterations of this P-Delta-KG procedure; however one iteration is almost always sufficient.

The P-Delta analysis is recommended by several design codes such as ACI 318, LRFD, IS456-1978, etc. in lieu of the moment magnification method for the calculation of more realistic forces and moments.

P-Delta effects are calculated for frame members and plate elements only. They are not calculated for solid elements.

The maximum displacement should be reviewed for P-Delta analyses because this analysis type permits buckling. You may need to repeat the analysis with only one to five iterations or as a static case in order to get a pre-collapse solution in order to view the large displacement areas.

Buckling may also cause the analysis to fail with a negative definite matrix failure. In this case, a message is printed and the results of the case are set to

zero. (In this case, repeat the analysis using **PDELTA 30 ANALYSIS SMALLDELTA** instead).

1.18.2.1.3 P-Delta K+Kg Dynamic Analysis

In STAAD, an alternate procedure has been adopted to incorporate the P-Delta effect into dynamic analysis by combining the global stiffness matrix and the global geometric stiffness matrix [K+Kg].

Note: This feature is available in STAAD.Pro 2007 Build 01 and greater.

This method uses the resulting [K+Kg] matrices from the last static case before the **PDELTA KG** command in the dynamic cases that precede the **PDELTA KG** command.

LOAD n

Static case input

LOAD n+1

Dynamic case input

PDELTA KG ANALYSIS

1. First, the primary deflections are calculated by linear static analysis based on the provided external loading for case n.
2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These terms are the terms of the Kg matrix which are added to the global stiffness matrix K.

The final triangular factorization for case n is then used in the dynamic case n+1 along with the masses specified in case n+1 to solve the dynamic analysis.

Lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. The **REPEAT LOAD** facility (See "Repeat Load Specification" on page 647) has been created with this requirement in mind. This facility allows the user to combine previously defined primary load cases to create a new primary load case.

P-Delta effects are calculated for frame members and plate elements only. They are not calculated for solid elements. P-Delta is restricted to structures where members and plate elements carry the vertical load from one structure level to the next.

Section 1 General Description

1.18 Analysis Facilities

The maximum displacement should be reviewed for P-Delta analyses because this analysis type permits buckling. You may need to repeat the analysis with only one to five iterations or as a static case in order to get a pre-collapse solution in order to view the large displacement areas.

Buckling may also cause the analysis to fail with a negative definite matrix failure. In this case a message is printed and the results of the case are set to zero. The dynamic results should be ignored if this type of failure should occur.

1.18.2.1.4 AISC 360-05 Direct Analysis

From STAAD.Pro 2007 Build 03 onwards, the ANSI/AISC 360-05 Direct Analysis procedure has been adopted to incorporate the P-Delta effect into a static analysis by combining the global stiffness matrix and the global geometric stiffness matrix [K+Kg]; plus flexural stiffness reduction; plus axial stiffness reduction; plus an additional flexure reduction if member axial compression forces are above 50% of yield; plus the addition of notional loads.

Note: This feature is available in STAAD.Pro 2007 Build 03 and greater.

1. First, the primary deflections are calculated by linear static analysis based on the provided external loading for case n. The stiffness reductions and notional loads are included here.
2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These forces and stresses are used to calculate geometric stiffness terms. These terms times the displacement results from the prior iteration create the P-Delta secondary loading. This secondary loading is then combined with the originally applied loading to create the effective load vector for the next iteration.
3. The final triangular factorization for case n is then used to calculate displacements and member forces.

Lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. The **REPEAT LOAD** facility (See "Repeat Load Specification" on page 647) has been created with this requirement in mind. This facility allows the user to combine previously defined primary load cases to create a new primary load case.

4. The axial force is compared to yield force to calculate τ_b (see Appendix 7 of

AISC 360-05). Flexure stiffness of selected members is set to $(0.8 * \tau_b * EI)$

5. Steps 2 to 4 are repeated until convergence or the iteration limit is reached.

See "Notional Loads" on page 665 for details on adding notional loads for a direct analysis.

1.18.2.2 Buckling Analysis

In STAAD, two procedures have been adopted to incorporate the calculation of the Buckling Factor for a load case. The buckling factor is the amount by which all of the loadings in a load case must be factored to cause global buckling of the structure.

Note: This feature is available in STAAD.Pro 2007 Build 01 and greater.

STAAD does not include the effects of geometric stiffness for solids. If the part of the structure that deforms during buckling involves non-trivial motions of solids, then the results will be erroneous for buckling (as well as for P-Delta analysis).

1.18.2.2.1 Basic Solver

In STAAD, a simple procedure has been adopted to incorporate the calculation of the Buckling Factor for any number of primary load cases. The buckling factor is the amount by which all of the loadings in a load case must be factored to cause global buckling of the structure.

1. First, the primary deflections are calculated by linear static analysis based on the provided external loading.
2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These terms are the terms of the Kg matrix which are multiplied by the estimated BF (buckling factor) and then added to the global stiffness matrix K.

Buckling Kg matrix effects are calculated for frame members and plate elements only. They are not calculated for solid elements. So buckling analysis is restricted to structures where members and plate elements carry the vertical load from one structure level to the next.

3. For compressive cases, the Kg matrix is negative definite. If the buckling factor is large enough, then $[[K]+BF*[Kg]]$ will also be negative definite

Section 1 General Description

1.18 Analysis Facilities

which indicates that BF times the applied loads is greater than the loading necessary to cause buckling.

4. STAAD starts an iterative procedure with a BF estimate of 1.0. If that BF causes buckling, then a new, lower BF estimate is used in the next trial. If the BF does not cause buckling, then a higher BF estimate is used. On the first iteration, if the determinant of the K matrix is positive and lower than the determinant of the K+Kg matrix, then the loads are in the wrong direction to cause buckling; and STAAD will stop the buckling calculation for that case.
5. After a few iterations, STAAD will have the largest BF that did not cause buckling (lower bound) and the lowest BF that did cause buckling (upper bound). Then each trial will use a BF estimate that is halfway between the current upper and lower bounds for BF.
6. After the default iteration limit is reached or the user specified iteration limit, MAXSTEPS, is reached or when two consecutive BF estimates are within 0.1% of each other; then the iteration is terminated.
7. Results for this load case are based on the last lower bound BF calculated.
 - Only primary load cases may be solved
 - Any number of buckling cases may be solved.
 - Only the first buckling mode (lowest BF) is calculated.
 - The buckling shape may not be as expected even though the buckling factor is OK. To enhance the mode shape result, apply small loads in the locations and directions where you expect the large displacements.

1.18.2.2.2 Advanced Solver

In STAAD, a second procedure has been adopted to incorporate the calculation of the Buckling Factor for one primary load case. The buckling factor is the amount by which all of the loadings in a load case must be factored to cause global buckling of the structure. This procedure is an eigenvalue calculation to get buckling factors and buckling shapes.

1. First, the primary deflections are calculated by linear static analysis based on the provided external loading.
2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate

geometric stiffness terms. Both the large delta effects and the small delta effects for members are calculated. These terms are the terms of the K_g matrix.

3. An eigenvalue problem is formed. $| [K] - BF_i * [K_g] | = 0$

There will be up to 4 buckling factors (BF) and associated buckling mode shapes calculated. The buckling factor is the amount by which the static load case needs to be multiplied by to just cause buckling (Euler buckling). BF less than 1.0 means that the load causes buckling; greater than 1.0 means buckling has not occurred. If BF is negative, then the static loads are in the opposite direction of the buckling load.

Notes

- Solid elements do not contribute to K_g in STAAD.
- Buckling shapes for the last buckling case only may be displayed in the postprocessor. If there are several buckling cases, then all will have their buckling factors printed.
- The displacement and member/element results are not calculated for the load case times the buckling factor.

1.18.2.3 Static Geometrically Nonlinear Analysis

In STAAD, a procedure has been adopted to incorporate the geometric nonlinearities into the analysis by updating the global stiffness matrix and the global geometric stiffness matrix $[K+K_g]$ on every step based on the deformed position. The deformations significantly alter the location or distribution of loads, such that equilibrium equations must be written with respect to the deformed geometry, which is not known in advance.

Note: This feature is available in STAAD.Pro 2007 Build 05 and greater.

1. First, the primary deflections are calculated by linear static analysis based on the provided external loading.
2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These terms are the terms of the K_g matrix which are added to the global stiffness matrix K .

Section 1 General Description

1.18 Analysis Facilities

3. Next the deflections are re-calculated. Now equilibrium is computed in the deformed position to get out of balance forces. The tangential stiffness matrix is determined from each members new position; the Kg matrix is updated; and the out of balance forces are applied to get the next iteration result.
4. Repeat until converged. If displacements are much too large, then try using **ARC 5** to limit displacements on the first linear static step to 5 inches or some suitable value. The **STEP 10** parameter may help by loading the structure over many steps.
5. The options for Newton-Raphson, Kg, Steps = 1 are usually taken; but these options are available for some difficult cases.
6. Offset beams, curved beams, cables are not permitted. Tension/compression is not permitted.

Nonlinear effects are calculated for springs, frame members and plate elements only. They are not calculated for solid elements.

The maximum displacement should be reviewed for Nonlinear analyses because this analysis type may result in buckling or large displacements.

The following limitations should be noted regarding static, geometrically nonlinear analyses:

- Large rotations in one step should be avoided by using more steps.
- Very large displacements, unstable structures, and/or post-buckling should be avoided.
- Geometrically nonlinear only. No tension/compression or contact is considered. No yield, plastic moment hinges or bilinear behavior is considered.
- Solids cannot be used for this analysis method.

Note: The nonlinear analysis command requires the Advanced Analysis Engine package.

See "Geometric Nonlinear Analysis" on page 703 for details.

1.18.2.4 Imperfection Analysis

Structures subjected to vertical and lateral loads often experience secondary forces due to curvature imperfections in the columns and beams. This secondary effect is similar to the P-Delta effect. In STAAD the procedure consists of the following steps:

1. First, the deflections and the axial forces in the selected imperfect members are calculated based on the provided external loading.
2. The axial forces and the input imperfections are then used to compute an additional loading on the selected imperfect members that are in compression. These additional loads are combined with the originally applied loading.
3. The static analysis is performed with the combined loading to obtain the final result.

The section moment due to tension and the section displacements due to shear/bending are added to the moment diagram, if small delta is selected. This is no iteration performed for this step.

See "Imperfection Analysis" on page 706 for details.

1.18.2.5 Multilinear Analysis

When soil is to be modeled as spring supports, the varying resistance it offers to external loads can be modeled using this facility, such as when its behavior in tension differs from its behavior in compression. Stiffness-Displacement characteristics of soil can be represented by a multi-linear curve. Amplitude of this curve will represent the spring characteristic of the soil at different displacement values. The load cases in a multi-linear spring analysis must be separated by the **CHANGE** command and **PERFORM ANALYSIS** command. The **SET NL** command must be provided to specify the total number of primary load cases. There may not be any **PDELTA**, dynamic, or **TENSION/ COMPRESSION** member cases. The multi-linear spring command will initiate an iterative analysis which continues to convergence.

1.18.2.6 Tension / Compression Only Analysis

When some members or support springs are linear but carry only tension (or only compression), then this analysis may be used. This analysis is automatically selected if any member or spring has been given the tension or compression only characteristic. This analysis is an iterative analysis which continues to convergence. Any member/ spring that fails its criteria will be inactive (omitted) on the next iteration. Iteration continues until all such members have the proper load direction or are inactive (default iteration limit is 10).

This is a simple method that may not work in some cases because members are removed on interim iterations that are needed for stability. If instability messages appear on the second and subsequent iterations that did not appear on

Section 1 General Description

1.18 Analysis Facilities

the first cycle, then do not use the solution. If this occurs on cases where only springs are the tension/compression entities, then use multi-linear spring analysis.

The load cases in a tension/compression analysis must be separated by the **CHANGE** command and **PERFORM ANALYSIS** command. The **SET NL** command must be provided to specify the total number of primary load cases. There may not be any Multi-linear springs, **NONLINEAR**, or dynamic cases.

1.18.2.7 Nonlinear Cable/Truss Analysis

Note: This feature is available in limited form.

When all of the members, elements and support springs are linear except for cable and/or preloaded truss members, then this analysis type may be used. This analysis is based on applying the load in steps with equilibrium iterations to convergence at each step. The step sizes start small and gradually increase (15-20 steps is the default). Iteration continues at each step until the change in deformations is small before proceeding to the next step. If not converged, then the solution is stopped. You can then select more steps or modify the structure and rerun.

Structures can be artificially stabilized during the first few load steps in case the structure is initially unstable (in the linear, small displacement, static theory sense).

The user has control of the number of steps, the maximum number of iterations per step, the convergence tolerance, the artificial stabilizing stiffness, and the minimum amount of stiffness remaining after a cable sags.

This method assumes small displacement theory for all members/trusses/elements other than cables & preloaded trusses. The cables and preloaded trusses can have large displacement and moderate/large strain. Cables and preloaded trusses may carry tension and compression but cables have a reduced E modulus if not fully taut. Pretension is the force necessary to stretch the cable/truss from its unstressed length to enable it to fit between the two end joints. Alternatively, you may enter the unstressed length for cables.

The current nonlinear cable analysis procedure can result in compressive forces in the final cable results. The procedure was developed for structures, loadings, and pretensioning loads that will result in sufficient tension in every cable for all loading conditions. The possibility of compression was considered acceptable in the initial implementation because most design codes strongly recommend cables to be in tension to avoid the undesirable dynamic effects of a slack cable such as

galloping, singing, or pounding. The engineer must specify initial preloading tensions which will ensure that all cable results are in tension. In addition this procedure is much more reliable and efficient than general nonlinear algorithms. To minimize the compression the **SAGMIN** input variable can be set to a small value such as 0.01, however that can lead to a failure to converge unless many more steps are specified and a higher equilibrium iteration limit is specified. SAGMIN values below 0.70 generally requires some adjustments of the other input parameters to get convergence.

Currently the cable and truss are not automatically loaded by selfweight, but the user should ensure that selfweight is applied in every load case. Do not enter component load cases such as wind only; every case must be realistic. Member loads will be lumped at the ends for cables and trusses. Temperature load may also be applied to the cables and trusses. It is OK to break up the cable/truss into several members and apply forces to the intermediate joints. Y-up is assumed and required.

The member force printed for the cable is F_x and is along the chord line between the displaced positions of the end joints.

The analysis sequence is as follows:

1. Compute the unstressed length of the nonlinear members based on joint coordinates, pretension, and temperature.
2. Member/Element/Cable stiffness is formed. Cable stiffness is from EA/L and the sag formula plus a geometric stiffness based on current tension.
3. Assemble and solve the global matrix with the percentage of the total applied load used for this load step.
4. Perform equilibrium iterations to adjust the change in directions of the forces in the nonlinear cables, so that the structure is in static equilibrium in the deformed position. If force changes are too large or convergence criteria not met within 15 iterations then stop the analysis.
5. Go to step 2 and repeat with a greater percentage of the applied load. The nonlinear members will have an updated orientation with new tension and sag effects.
6. After 100% of the applied load has converged then proceed to compute member forces, reactions, and static check. Note that the static check is not exactly in balance due to the displacements of the applied static equivalent joint loads.

The load cases in a nonlinear cable analysis must be separated by the **CHANGE** command and **PERFORM CABLE ANALYSIS** command. The **SET NL** command

Section 1 General Description

1.18 Analysis Facilities

must be provided to specify the total number of primary load cases. There may not be any Multi-linear springs, compression only, PDelta, NONLINEAR, or dynamic cases.

Also for cables and preloaded trusses:

1. Do not use Member Offsets.
2. Do not include the end joints in Master/Slave command.
3. Do not connect to inclined support joints.
4. Y direction must be up.
5. Do not impose displacements.
6. Do not use Support springs in the model.
7. Applied loads do not change global directions due to displacements.
8. Do not apply Prestress load, Fixed end load.
9. Do not use Load Combination command to combine cable analysis results. Use a primary case with Repeat Load instead.

1.18.3 Dynamic Analysis

Available dynamic analysis facilities include solution of the free vibration problem (eigenproblem), response spectrum analysis and forced vibration analysis.

1.18.3.1 Solution of the Eigenproblem

The eigenproblem is solved for structure frequencies and mode shapes considering a diagonal, lumped mass matrix, with masses possible at all active d.o.f. included. Two solution methods may be used: the subspace iteration method for all problem sizes (default for all problem sizes), and the optional determinant search method for small problems.

1.18.3.2 Mass Modeling

The natural frequencies and mode shapes of a structure are the primary parameters that affect the response of a structure under dynamic loading. The free vibration problem is solved to extract these values. Since no external forcing function is involved, the natural frequencies and mode shapes are direct functions of the stiffness and mass distribution in the structure. Results of the frequency and mode shape calculations may vary significantly depending upon the mass modeling. This variation, in turn, affects the response spectrum and forced

vibration analysis results. Thus, extreme caution should be exercised in mass modeling in a dynamic analysis problem.

In STAAD, all masses that are capable of moving should be modeled as loads applied in all possible directions of movement. Even if the loading is known to be only in one direction there is usually mass motion in other directions at some or all joints and these mass directions (applied as loads, in weight units) must be entered to be correct. Joint moments that are entered will be considered to be weight moment of inertias (force-length² units).

Please enter selfweight, joint, and element loadings in global directions with the same sign as much as possible so that the representative masses do not cancel each other.

Member/Element loadings may also be used to generate joint translational masses. Member end joint moments that are generated by the member loading (including concentrated moments) are discarded as irrelevant to dynamics. Enter mass moments of inertia, if needed, at the joints as joint moments.

STAAD uses a diagonal mass matrix of six lumped mass equations per joint. The selfweight or uniformly loaded member is lumped 50% to each end joint without rotational mass moments of inertia. The other element types are integrated but—roughly speaking—the weight is distributed equally amongst the joints of the element.

The members/elements of finite element theory are simple mathematical representations of deformation meant to apply over a small region. The finite element analysis (FEA) procedures will converge if you subdivide the elements and rerun; then subdivide the elements that have significantly changed results and rerun; and so on, until the key results are converged to the accuracy needed.

An example of a simple beam problem that needs to subdivide physical members to better represent the mass distribution (as well as the dynamic response and the force distribution response along members) is a simple floor beam between two columns will put all of the mass on the column joints. In this example, a vertical ground motion will not bend the beam even if there is a concentrated force (mass) at mid span.

Masses that are assigned to slave degrees of freedom (dof) are moved to the master node with a rotatory mass moment of inertia applied at the master. This will be an approximation if the master node is not at the center of gravity (CG, i.e., center of mass) of the slave masses.

In addition, the dynamic results will not reflect the location of a mass within a member (i.e., the masses are lumped at the joints). This means that the motion, of a large mass in the middle of a member relative to the ends of the member, is

Section 1 General Description

1.18 Analysis Facilities

not considered. This may affect the frequencies and mode shapes. If this is important to the solution, split the member into two. Another effect of moving the masses to the joints is that the resulting shear/moment distribution is based as if the masses were not within the member.

Note: If one end of a member is a support, then half of the member mass is lumped at the support and will not move during the dynamic response. Use **ENFORCED** supports to minimize this limitation.

1.18.3.3 Damping Modeling

Damping may be specified by entering values for each mode, or using a formula based on the first two frequencies, or by using composite modal damping. Composite modal damping permits computing the damping of a mode from the different damping ratios for different materials (steel, concrete, soil). Modes that deform mostly the steel would have steel damping ratio, whereas modes that mostly deform the soil, would have the soil damping ratio.

Composite Damping

Composite modal damping is based on a weighted average of strain energies in each material, (or element), for each mode (eigenvector). The critical composite damping term D_j for mode J is computed by:

$$D_i = \frac{\sum_{i=1}^n \{\phi_i\}^T b_i \cdot [k] \cdot \{\phi_i\}}{\{\phi_j\}^T \cdot [K] \cdot \{\phi_j\}}$$

Where:

n = total number of degrees of freedom

b_i = equivalent percent of critical damping associated with component i

$\{\phi_j\}$ = mode shape vector for mode J

$[k]_i$ = stiffness associated with component i

$[K]$ = stiffness matrix for the system

1.18.3.4 Response Spectrum

This capability allows the user to analyze the structure for seismic loading. For any supplied response spectrum (either acceleration vs. period or displacement

vs. period), joint displacements, member forces, and support reactions may be calculated. Modal responses may be combined using one of the square root of the sum of squares (SRSS), the complete quadratic combination (CQC), the ASCE4-98 (ASCE), the Ten Percent (TEN) or the absolute (ABS) methods to obtain the resultant responses. Results of the response spectrum analysis may be combined with the results of the static analysis to perform subsequent design. To account for reversibility of seismic activity, load combinations can be created to include either the positive or negative contribution of seismic results.

1.18.3.5 Response Time History

STAAD is equipped with a facility to perform a response history analysis on a structure subjected to time varying forcing function loads at the joints and/or a ground motion at its base. This analysis is performed using the modal superposition method. Hence, all the active masses should be modeled as loads in order to facilitate determination of the mode shapes and frequencies. Please refer to the previous section on "mass modeling" for additional information on this topic. In the mode superposition analysis, it is assumed that the structural response can be obtained from the "p" lowest modes. The equilibrium equations are written as

$$[m]\{x''\} + [c]\{x'\} + [k]\{x\} = \{P(t)\}$$

Note: The double-prime notation (") designates the second derivative (i.e., acceleration) and a prime notation (') designates the first derivative (i.e., velocity).

Using the transformation

$$\{x\} = \sum_{i=1}^p \{\phi\}_i q_i$$

The equation for $\{P(t)\}$ reduces to "p" separate uncoupled equations of the form

$$q_i'' + 2 \xi_i \omega_i q_i' + \omega_i^2 q_i = R_i(t)$$

where:

ξ = the modal damping ration

ω = the natural frequency for the i^{th} mode.

These are solved by the Wilson- θ method which is an unconditionally stable step by step scheme. The time step for the response is entered by you or set to a default value, if not entered. The q_i s are substituted in equation 2 to obtain the displacements $\{x\}$ at each time step.

Section 1 General Description

1.18 Analysis Facilities

Time History Analysis for a Structure Subjected to a Harmonic Loading

A Harmonic loading is one in which can be described using the following equation

$$F(t) = F_0 \sin(\omega t + \phi)$$

Where:

$F(t)$ = Value of the forcing function at any instant of time "t"

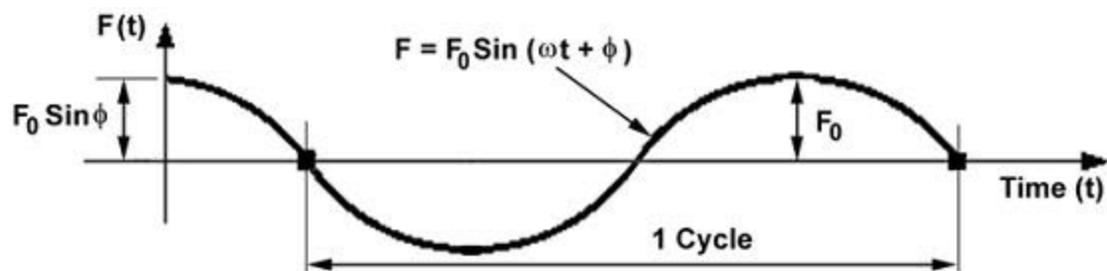
F_0 = Peak value of the forcing function

ω = Frequency of the forcing function

ϕ = Phase angle

A plot of the above equation is shown in the figure below.

Figure 1-41: Harmonic loading function



The results are the maximums over the entire time period, including start-up transients. So, they do not match steady-state response.

Definition of Input in STAAD for the above Forcing Function

As can be seen from its definition, a forcing function is a continuous function. However, in STAAD, a set of discrete time-force pairs is generated from the forcing function and an analysis is performed using these discrete time-forcing pairs. What that means is that based on the number of cycles that you specify for the loading, STAAD will generate a table consisting of the magnitude of the force at various points of time. The time values are chosen from this time 'o' to $n \cdot t_c$ in steps of "STEP" where n is the number of cycles and t_c is the duration of one cycle. **STEP** is a value that you may provide or may choose the default value that is built into the program. STAAD will adjust **STEP** so that a $\frac{1}{4}$ cycle will be evenly divided into one or more steps. See "Definition of Time History Load" on page 507 for a list of input parameters that need to be specified for a Time History Analysis on a structure subjected to a Harmonic loading.

The relationship between variables that appear in the STAAD input and the corresponding terms in the equation shown above is explained below.

F_o = Amplitude

ω = Frequency

ϕ = Phase

Forces applied at slave dof will be ignored; apply them at the master instead.

1.18.3.6 Steady State and Harmonic Response

A structure [subjected only to harmonic loading, all at a given forcing frequency and with non-zero damping] will reach a steady state of vibration that will repeat every forcing cycle. This steady state response can be computed without calculating the transient time history response prior to the steady state condition.

$$R(t) = R_o \sin(\omega t + \phi)$$

The result, R , has a maximum value of R_o and a phase angle ϕ . These two values for displacement, velocity, and acceleration at each joint may be printed or displayed.

This analysis is performed using the modal superposition method. Hence, all the active masses should be modeled as loads in order to facilitate determination of the mode shapes and frequencies. See "Mass Modeling" on page 80 for additional information on this topic. In the mode superposition analysis, it is assumed that the structural response can be obtained from the "p" lowest modes.

A Harmonic loading is one in which can be described using the following equation

$$F(t) = F_o \sin(\omega t + \phi)$$

Where:

$F(t)$ = Value of the forcing function at any instant of time "t"

F_o = Peak value of the forcing function

ω = Frequency of the forcing function

ϕ = Phase angle

A plot of the above equation is shown in Section 1.18.3.5.

Section 1 General Description

1.18 Analysis Facilities

The results are the steady-state response which is the absolute maximum of displacement (and other output quantities) and the corresponding phase angle after the steady state condition has been reached.

In addition, a Harmonic response can be calculated. This response consists of a series of Steady State responses for a list of frequencies. The joint displacement, velocity, or acceleration can be displayed as the response value versus frequency. Load case results are the maximums over all of the frequencies.

All results are positive as in the Response Spectrum and Time history analyses. This means section results should be ignored (BEAM o.o in Parameters for code checking). Because of this, you may want to add the steady state response to Dead & Live loads for one combination case and subtract the steady state response from those loads for another combination case.

Ground motion or a joint force distribution may be specified. Each global direction may be at a different phase angle.

Output frequency points are selected automatically for modal frequencies and for a set number of frequencies between modal frequencies. There is an option to change the number of points between frequencies and an option to add frequencies to the list of output frequencies.

The load case that defines the mass distribution must be the case just before the **PERFORM STEADY STATE ANALYSIS** command. Immediately after that command is a set of data starting with **BEGIN STEADY** and ending with **END STEADY**. The list of additional frequencies and the steady state load cases with joint loads or ground accelerations and phasing data are entered here. The optional print command for the maximum displacement and associated phase angle for selected joints must be at the end of this block of input.

Note: Stardyne-Dynrez data beginning with **START2** and ending with **ALL DONE** may substitute for the **BEGIN** to **END STEADY** data if the **STRESS** data is omitted.

Note: A license for the advanced analysis module is required to access this feature.

1.18.3.7 Pushover Analysis

Pushover analysis is a static, nonlinear procedure using simplified nonlinear technique to estimate seismic structural deformations. It is an incremental static

analysis used to determine the force-displacement relationship, or the capacity curve, for a structure or structural element.

In STAAD, the basis for this analysis is the information published in the documents FEMA 356 : 2000 and ATC 40.

Please contact Bentley's Technical Support Group for a separate document containing the details of the implementation.

Note: A license for the advanced analysis module is required to access this feature.

1.19 Member End Forces

Member end forces and moments in the member result from loads applied to the structure. These forces are in the local member coordinate system. The following figures show the member end actions with their directions.

Figure 1-42: Member end forces when Global Y is vertical

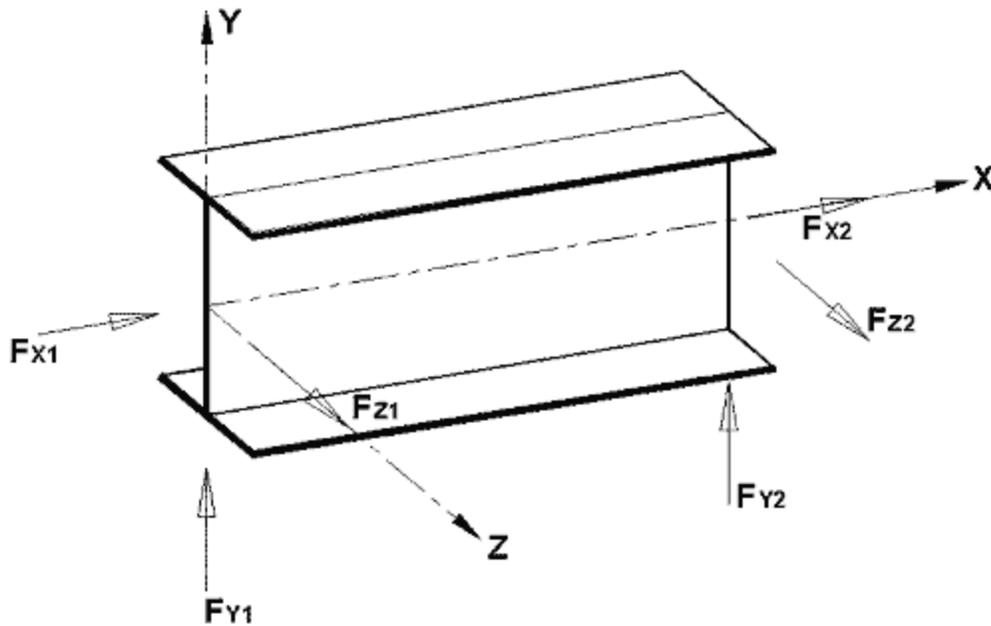


Figure 1-43: Member end moments when Global Y is vertical

Section 1 General Description

1.19 Member End Forces

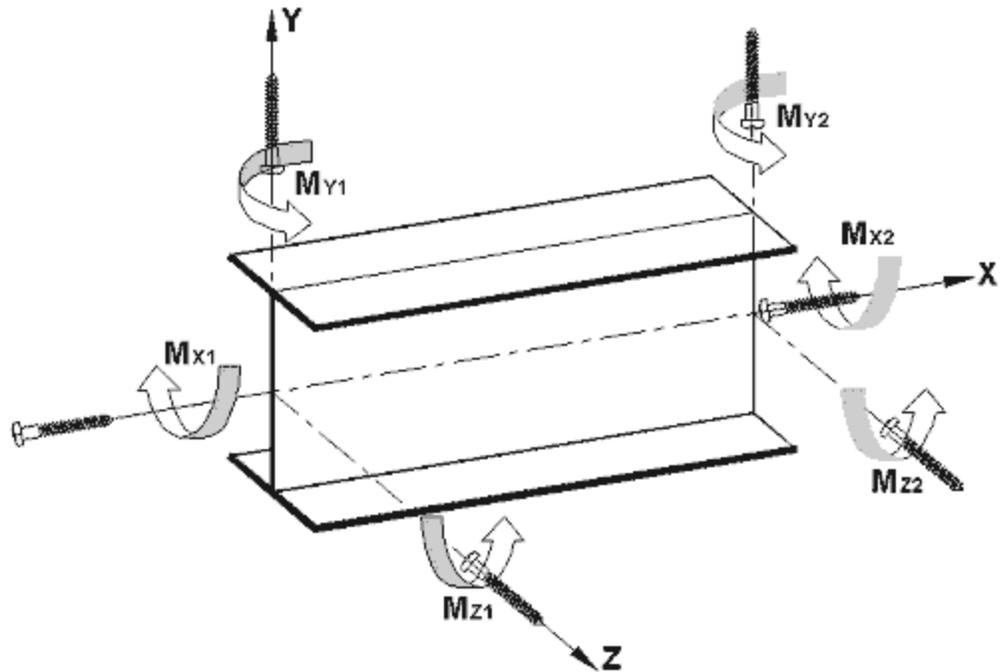


Figure 1-44: Member end forces when Global Z is vertical (that is, **SET Z UP** is specified)

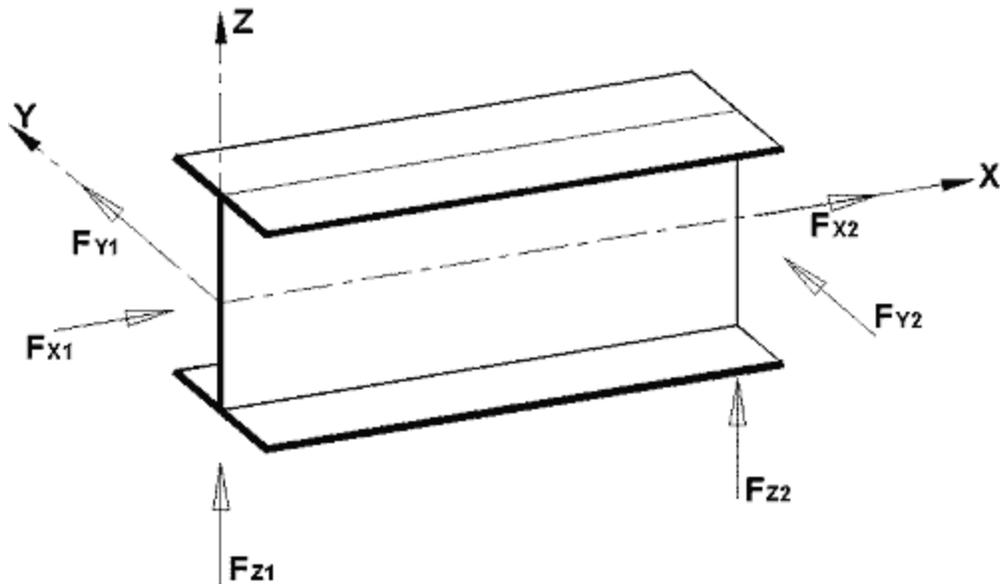
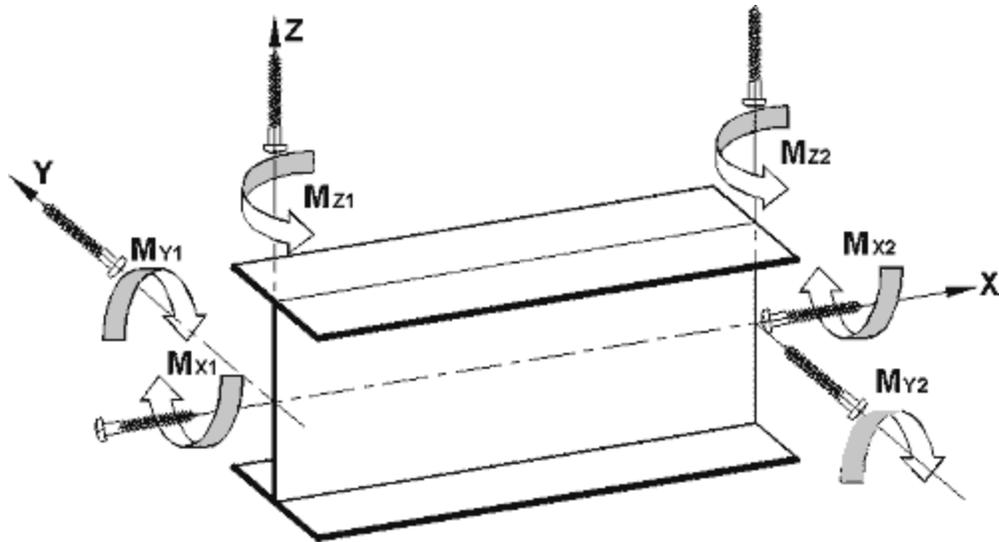
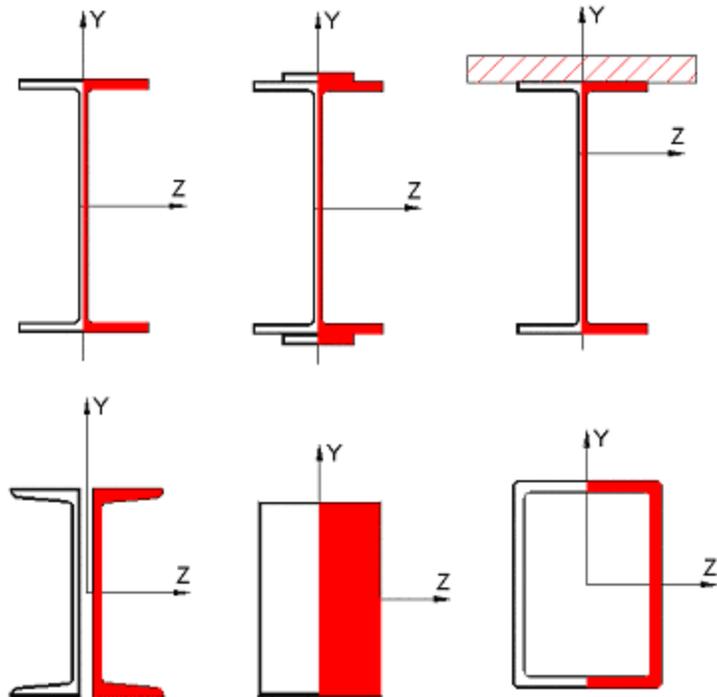


Figure 1-45: Member end forces when Global Z is vertical (that is, **SET Z UP** is specified)



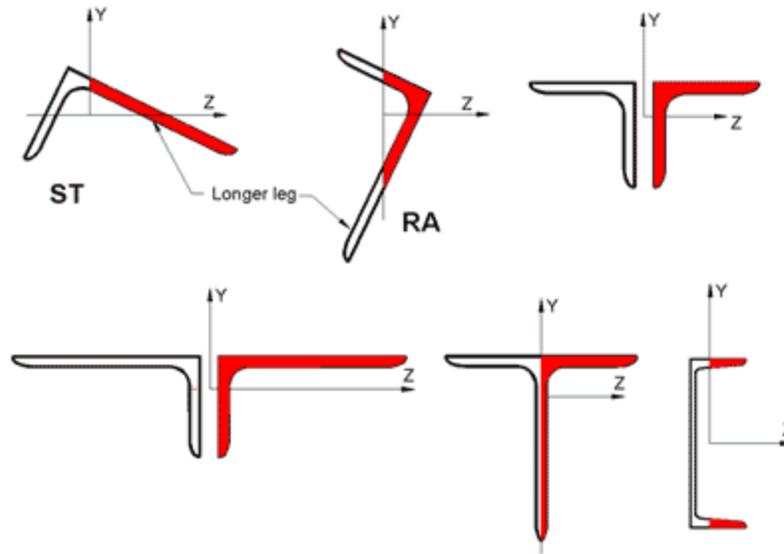
Stress Zones Due to Bending

Figure 1-46: Stress zones due to bending about the Y axis (M_Y) for various section types



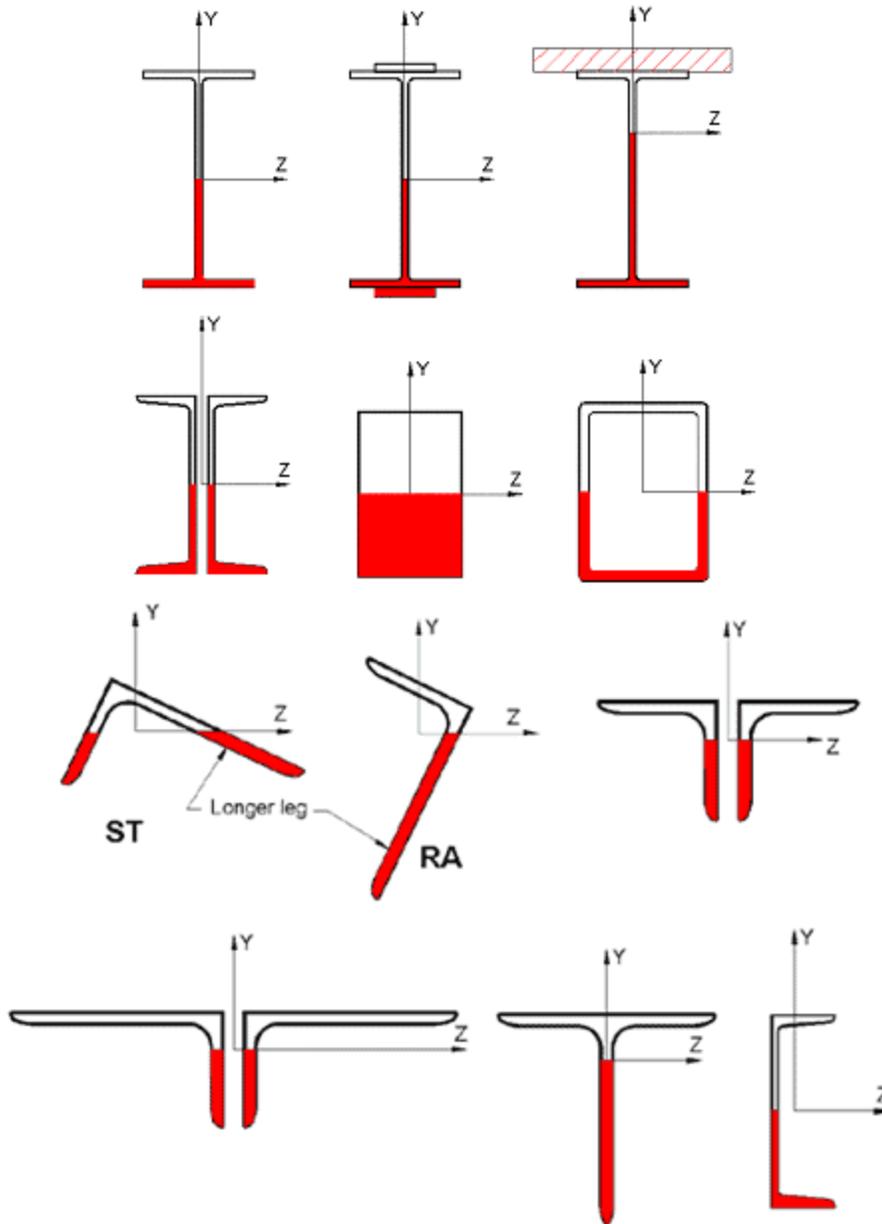
Section 1 General Description

1.19 Member End Forces



Note: Local X axis goes into the page; Global Y is vertically upwards; Shaded area indicates zone under compression; Non-shaded area indicates zone under tension

Figure 1-47: Stress zones due to bending about the Z axis (MZ) for various section types



Note: Local X axis goes into the page; Global Y is vertically upwards; Shaded area indicates zone under compression; Non-shaded area indicates zone under tension.

See "Section Specification" on page 713 for details.

Section 1 General Description

1.19 Member End Forces

1.19.1 Secondary Analysis

Solution of the stiffness equations yield displacements and forces at the joints or end points of the member. STAAD is equipped with the following secondary analysis capabilities to obtain results at intermediate points within a member.

1.19.1.1 Member Forces at Intermediate Sections

With the **SECTION** command, you may choose any intermediate section of a member where forces and moments need to be calculated. These forces and moments may also be used in design of the members. The maximum number of sections specified may not exceed five, including one at the start and one at the end of a member. If no intermediate sections are requested, the program will consider the start and end member forces for design. However, of the sections provided, they are the only ones to be considered for design.

1.19.1.2 Member Displacements at Intermediate Sections

Like forces, displacements of intermediate sections of members can be printed or plotted. This command may not be used for truss or cable members.

1.19.1.3 Member Stresses at Specified Sections

Member stresses can be printed at specified intermediate sections as well as at the start and end joints. These stresses include:

- a. Axial stress, which is calculated by dividing the axial force by the cross sectional area,
- b. Bending-y stress, which is calculated by dividing the moment in local-y direction by the section modulus in the same direction,
- c. Bending-z stress, which is the same as above except in local-z direction,
- d. Shear stresses (in y and z directions), and
- e. Combined stress, which is the sum of axial, bending-y and bending-z stresses.

All the stresses are calculated as the absolute value.

1.19.1.4 Force Envelopes

Force envelopes of the member forces FX (axial force), FY (Shear-y), and MZ (moment around local z-axis, i.e., strong axis) can be printed for any number of

intermediate sections. The force values include maximum and minimum numbers representing maximum positive and maximum negative values. The following is the sign convention for the maximum and minimum values:

- FX A positive value is compression, and negative tension.
- FY A positive value is shear in the positive y-direction, and negative in the negative y-direction.
- FZ Same as above, except in local z-direction.
- MZ A positive moment will mean a moment causing tension at the top of the member. Conversely, a negative moment will cause tension at the bottom of the member. The top of a member is defined as the side towards positive local y-axis.
- MY Same as above, except about local z axis.

1.20 Multiple Analyses

Structural analysis/design may require multiple analyses in the same run. STAAD allows you to change input such as member properties, support conditions etc. in an input file to facilitate multiple analyses in the same run. Results from different analyses may be combined for design purposes.

For structures with bracing, it may be necessary to make certain members inactive for a particular load case and subsequently activate them for another. STAAD provides an **INACTIVE** facility for this type of analysis.

Inactive Members

With the **INACTIVE** command, members can be made inactive. These inactive members will not be considered in the stiffness analysis or in any printout. The members made inactive by the **INACTIVE** command are made active again with the **CHANGE** command. This can be useful in an analysis where stage construction is modeled due to which, a set of members should be inactive for certain load cases. This can be accomplished by:

- a. making the desired members inactive
- b. providing the relevant load cases for which the members are inactive
- c. performing the analysis
- d. using the **CHANGE** command to make all the inactive members active
- e. making another set of members inactive and providing the proper load

Section 1 General Description

1.21 Steel, Concrete, and Timber Design

cases for which the members are meant to be inactive, performing the analysis and repeating the procedure as necessary.

1.21 Steel, Concrete, and Timber Design

Detailed information on the extensive design capabilities in STAAD for steel, concrete and timber is presented in Sections 2, 3 and 4 respectively.

Detailed information on the extensive design capabilities in STAAD for American steel, concrete, and timber codes is presented in the Technical Reference Manual. Detailed information on codes from other countries and regions, as well as specialized American codes and Aluminum design is presented in the International Design Codes manual.

1.22 Footing Design

Note: This section has been removed.

Contact Bentley's Technical Support Group for further information.

1.23 Printing Facilities

All input data and output may be printed using **PRINT** commands available in STAAD. The input is normally echoed back in the output. However, if required, the echo can be switched off.

Extensive listing facilities are provided in almost all **PRINT** commands to allow you to specify joints, members and elements for which values are required.

1.24 Plotting Facilities

Please refer to the STAAD.Pro Graphical Environment Manual for a complete description of the extensive screen and hardcopy graphical facilities available and information on using them.

1.25 Miscellaneous Facilities

STAAD offers the following miscellaneous facilities for problem solution.

Perform Rotation

This command can be used to rotate the structure shape through any desired angle about any global axis. The rotated configuration can be used for further

analysis and design. This command may be entered after the Joint Coordinates or between two Joint Coordinate commands or after all Member/Element Incidences are specified.

See "Rotation of Structure Geometry" on page 324 for details.

Substitute

Joint and member numbers may be redefined in STAAD through the use of the **SUBSTITUTE** command. After a new set of numbers are assigned, input and output values will be in accordance with the new numbering scheme. This facility allows the user to specify numbering schemes that will result in simple input specification as well as easy interpretation of data.

See "Redefinition of Joint and Member Numbers" on page 317 for details.

Calculation of Center of Gravity

STAAD is capable of calculating the center of gravity of the structure. The **PRINT CG** command may be utilized for this purpose.

See "Print Specifications " on page 714 for details.

1.26 Post Processing Facilities

All output from the STAAD engine may be utilized for further processing by the STAAD.Pro Graphical Interface. Please refer to the STAAD.Pro Graphical Environment Manual for a complete description of the extensive screen and hardcopy graphical facilities available and for information on how to use them.

Section 1 General Description

Section 2

American Steel Design

2.1 Design Operations	97
2.2 Member Properties	98
2.3 Steel Design per AISC 360 Unified Specification	102
2.4 Steel Design per AISC 9th Edition	112
2.5 Steel Design per AASHTO Specifications	175
2.6 Design per American Cold Formed Steel Code	209

2.1 Design Operations

STAAD contains a broad set of facilities for designing structural members as individual components of an analyzed structure. The member design facilities provide the user with the ability to carry out a number of different design operations. These facilities may be used selectively in accordance with the requirements of the design problem. The operations to perform a design are:

- Specify the members and the load cases to be considered in the design.
- Specify whether to perform code checking or member selection.

Section 2 American Steel Design

2.2 Member Properties

- Specify design parameter values, if different from the default values.

These operations may be repeated by the user any number of times depending upon the design requirements.

Steel Design may be performed according to several codes such as AISC-ASD (9th edition), AISC-LRFD, AISC 13th edition, AISI, AASHTO, etc. A brief description of each is presented in the following pages.

Currently, STAAD supports steel design of wide flange, S, M, HP shapes, tees, angle, double angle, channel, double channel, pipes, tubes, beams with cover plate and composite beams (I shapes with concrete slab on top).

2.2 Member Properties

For specification of member properties of standard American steel sections, the steel section library available in STAAD may be used. The syntax for specifying the names of built-in steel shapes is described in the next section.

2.2.1 Built-in Steel Section Library

The following sections describe specification of steel sections from the AISC Steel Tables.

2.2.1.1 AISC Steel Table

Almost all AISC steel shapes are available for input. Following are the descriptions of all the types of sections available:

Wide Flanges (W shapes)

All wide flange sections as listed in AISC are available the way they are written, e.g., W10X49, W21X50, etc.

```
20  TO  30  TA  ST  W10X49
33  36  TA  ST  W18X86
```

C, MC, S, M, HP Shapes

The above shapes are available as listed in AISC (9th Edition) without decimal points. For example, C8X11.5 will be input as **C8X11** and S15X42.9 will be input as **S15X42**, omitting the fractional portion of the weight past the decimal.

Note: Exception: **MC6X151** for MC6X15.1 and **MC6X153** for MC6X15.3.

10 TO 20 BY 2 TA ST C15X40

1 2 TA ST MC8X20

Double Channels

Back to back double channels, with or without spacing between them, are available. The letter D in front of the section name will specify a double channel.

21 22 24 TA D MC9X25

55 TO 60 TA D C8X18

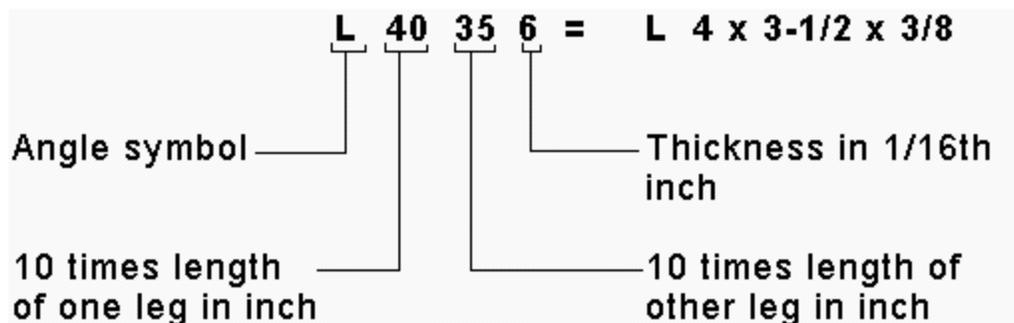
Front to front channel steel sections can be defined by using the FR (Double Channel – Front to Front) option and any spacing required between the channels are specified with the SP designation.

61 62 TABLE FR C4X5 SP 0.5

Note: The SP parameter is optional, but if it is not set, the section will not be assumed to be a closed box for torsional calculations.

Angles

Angle specifications in STAAD are different from those in the AISC manual. The following example illustrates angle specifications.



Similarly, **L505010** = L 5 x 5 x 5/8 and **L904016** = L 9 x 4 x 1

Section 2 American Steel Design

2.2 Member Properties

At present, there are two ways to define the local y and z-axes for an angle section. To make the transition from the AISC Manual to the program data easy, the standard section for an angle is specified:

```
51 52 53 TA ST L40356
```

This specification has the local z-axis (i.e., the minor axis) corresponding to the Z-Z axis specified in the steel tables. Many engineers are familiar with a convention used by some other programs in which the local y-axis is the minor axis. STAAD provides for this convention by accepting the command:

```
54 55 56 TA RA L40356
```

Note: RA denotes reverse angle

Double Angles

Short leg back to back or long leg back to back double angles can be specified by inputting the word SD or LD, respectively, in front of the angle size. In case of an equal angle either LD or SD will serve the purpose.

```
14 TO 20 TA LD L35304 SP 0.5
```

Long leg back to back L_{3-1/2}x₃x_{1/4} with 0.5 space.

```
23 27 TA SD L904012
```

Short leg back to back L 9x4x_{3/4}

Tees

Tees are not input by their actual names, as they are listed in the AISC manual, but instead by designating the beam shapes (W and S) from which they are cut. For example

```
1 2 5 8 TA T W8X24
```

Tee cut from W8x24, or a TW₄x₁₂

Pipes

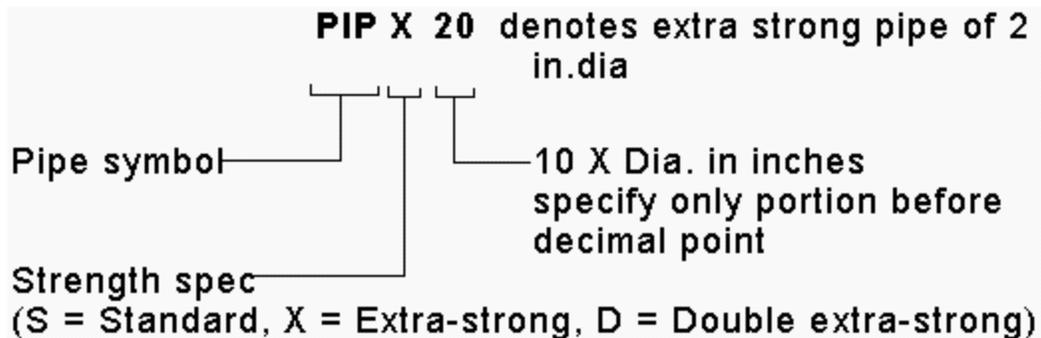
Two types of specifications can be used for pipe sections. In general pipes may be input by their outer and inner diameters. For example,

1 TO 9 TA ST PIPE OD 2.0 ID 1.875

Will mean a pipe with an outer diameter of 2.0 and inner diameter of 1.875 in current input units.

Pipe sections listed in the AISC manual can be specified as follows.

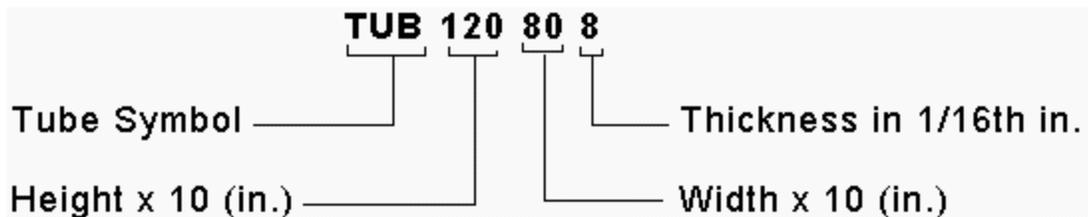
5 TO 10 TA ST PIPX20



Tubes

Tubes from the AISC tables can be specified as follows.

5 TO 10 TA ST TUB120808



Tubes, like pipes, can be input by their dimensions (Height, Width and Thickness) as follows.

6 TA ST TUBE DT 8.0 WT 6.0 TH 0.5

Section 2 American Steel Design

2.3 Steel Design per AISC 360 Unified Specification

Is a tube that has a height of 8, a width of 6, and a wall thickness of 0.5 in the current input units.

Note: Member Selection cannot be performed on tubes specified in the latter way. Only code checking can be performed on these sections.

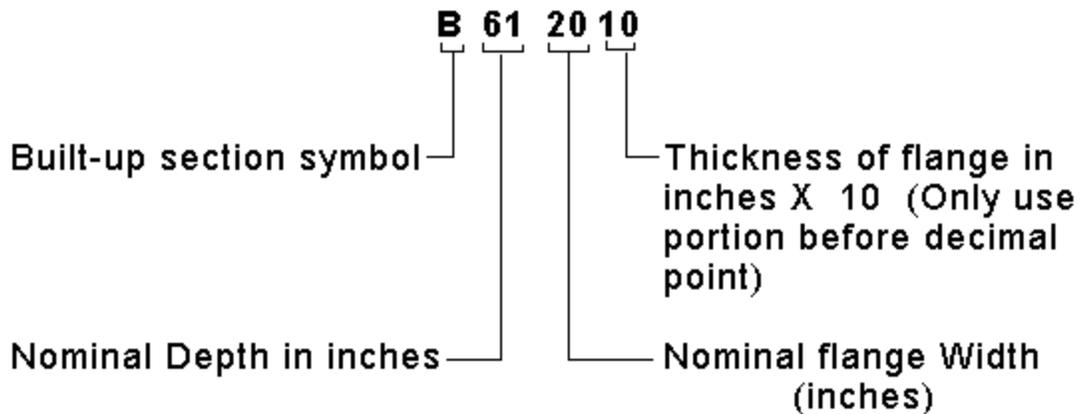
2.2.1.2 Welded Plate Girders

The AISC welded plate girder shapes (pages 2-230 and 2-231 – AISC 9th edition) are available in the Steel Section library of the program.

1 TO 10 TA ST B612010

15 16 TA ST B682210

Nomenclature



2.2.1.3 Castellated Beams

STAAD.Pro incorporates the non-composite castellated beam tables supplied by the steel products manufacturer SMI Steel Products. See "Castellated Beams" on page 151 for details.

2.3 Steel Design per AISC 360 Unified Specification

Steel member design per ANSI/AISC 360-05 and 360-10, *Specifications for Structural Steel Buildings*, has been implemented in STAAD.Pro. These specifications are published as part of the *AISC Steel Construction Manual*. Since the ASD and the LRFD method are both addressed in those specifications, they are referred to as **UNIFIED**.

To use the 2010 edition (default), specify the command:

CODE AISC UNIFIED

or

CODE AISC UNIFIED 2010

To use the 2005 edition, specify the command:

CODE AISC UNIFIED 2005

Hint: Either method may be selected in the user interface using the Steel Design - Whole Structure dialog.

Design can be performed according to the provisions for **Load and Resistance Factor Design (LRFD)** or to the provisions for **Allowable Strength Design (ASD)**, as per section B3 of the code. This selection of the design methodology can be done through the **METHOD** parameter. The full list of parameters is given in section 2.3.8.

2.3.1 General Comments on Design as per AISC Unified Code

Both Allowable Stress Design and Load Resistance Factor Design methods are implemented in STAAD. The selection of the method can be done through the **METHOD** parameter explained in the parameter list. This Unified Code allows the designer to design the member as per LRFD as well as ASD method.

Design for Strength Using Load and Resistance Factor Design (LRFD)

Design according to the provisions for Load and Resistance Factor Design (LRFD) satisfies the requirements of the AISC 360 Unified Code Specification, when the design strength of each structural component equals or exceeds the required strength determined on the basis of the LRFD load combinations.

Design shall be performed in accordance with Equation B3-1 of the Code:

$$R_u \leq \phi R_n$$

Where:

R_u = Required Strength (LRFD)

R_n = Nominal Strength,

ϕ = Resistance Factor,

ϕR_n = Design Strength

Section 2 American Steel Design

2.3 Steel Design per AISC 360 Unified Specification

Design for Strength Using Allowable Strength Design (ASD)

Design according to the provisions for Allowable Strength Design (ASD) satisfies the requirements of the AISC 360 Unified Code Specification when the allowable strength of each structural component equals or exceeds the required strength determined on the basis of the ASD load combinations.

Design shall be performed in accordance with Equation B3-2 of the Code:

$$R_a \leq R_n/\Omega$$

Where:

R_a = Required Strength (ASD)

R_n = Nominal Strength,

Ω = Safety Factor,

R_n/Ω = Allowable Strength

2.3.2 Section Classification

The LRFD specification allows inelastic deformation of section elements. Thus local buckling becomes an important criterion. Steel sections are classified as compact, non-compact or slender element sections depending upon their local buckling characteristics. This classification is a function of the geometric properties of the section. The design procedures are different depending on the section class. STAAD.Pro is capable of determining the section classification for the standard shapes and design accordingly.

The Section Classification is done as per section B4 and Table B4.1, for Stiffened and Un-Stiffened Elements of a section.

2.3.3 Limit States

2.3.3.1 Axial Tension

The criteria governing the capacity of tension members are based on:

- Tensile Yielding in Gross Section and
- Tensile Rupture of Net Section.

The limit state of yielding in the gross section is intended to prevent excessive elongation of the member, and the corresponding check is done as per section D2-(a) of the code.

The second limit state involves fracture at the section with the minimum effective net area, and the corresponding check is done as per section D2-(b) of the code.

STAAD calculates the tension capacity of a given member based on these two limit states.

The Net Section Area may be specified by the user through the use of the parameter NSF (see Table 2.1). The Effective Net Area of tension members can be determined by using the Shear Lag Factor. You can also input the shear lag factor through the use of the parameter SLF.

2.3.3.2 Axial Compression

The Design Compressive Strength (LRFD), $\phi_c P_n$, and the Allowable Compressive Strength (ASD), P_n/Ω_c , are calculated by the program.

The Nominal Compressive Strength, P_n , shall be the minimum value obtained according to the Limit States of -

- Flexural Buckling,
- Torsional Buckling, and
- Flexural-Torsional Buckling.

The Nominal Compressive Strength, P_n , for a particular member is calculated by STAAD according to the procedure outlined in Chapter E, section E3 to E5, of the unified code specifications. For slender elements, the procedure described in section E7 is used.

Effective length for calculation of compression resistance may be provided through the use of the parameters KY, KZ. If not provided, the entire member length will be taken into consideration.

In addition to the compression resistance criterion, compression members are required to satisfy slenderness limitations which are a function of the nature of use of the member (main load resisting component, bracing member, etc.). In both the member selection and code checking process, STAAD immediately does a slenderness check on appropriate members before continuing with other procedures for determining the adequacy of a given member.

2.3.3.3 Flexural Design Strength

The Design Flexural Strength (LRFD), $\phi_b M_n$, and the Allowable Flexural Strength (ASD), M_n/Ω_b , are being calculated by the program.

Section 2 American Steel Design

2.3 Steel Design per AISC 360 Unified Specification

The Nominal Flexural Strength, M_n , is determined according to Sections F2 through F12 of unified code specifications, for different types of rolled sections.

The Nominal Flexural Strength of a member is determined by the limit states of Yielding (Y), Lateral-Torsional Buckling (LTB), Flange Local Buckling (FLB), Web Local Buckling (WLB), Tension Flange Yielding (TFY), Leg Local Buckling (LLB), and Local Buckling (LB).

The program internally calculates the Lateral-Torsional Buckling Modification Factor (C_b) for non-uniform moment diagrams when both ends of the unsupported segment are braced. The purpose of this factor is to account for the influence of the moment gradient on lateral-torsional buckling.

To specify laterally unsupported length, the parameter **UNF** can be used, by default which takes the value of the member length.

2.3.3.4 Design for Shear

The Design Shear Strength (LRFD), $\phi_v \cdot V_n$, and the Allowable Shear Strength (ASD), V_n / Ω_v , are calculated by the program, as per section G2 of the unified code specifications.

The Nominal Shear Strength, V_n , of un-stiffened or stiffened webs, is calculated taking care of limit states of shear yielding and shear buckling. The sections G4 to G7 of the code specifications are used to evaluate Nominal Shear Strength, V_n for different types of rolled sections.

2.3.3.5 Design for Combined Forces and Torsion

The interaction of flexure and axial forces in singly and doubly symmetric shapes is governed by sections H1 and H3. These interaction formulas cover the general case of biaxial bending combined with axial force and torsion. They are also valid for uniaxial bending and axial force.

Note: STAAD.Pro only performs checks for torsion per formula H3 for closed HSS sections. Open sections—such as wide flanges—which may be subject warping stresses are not checked.

2.3.4 Design Parameters

Design per AISC 360-05 and 360-10 (Unified) specifications is requested by using the **CODE** parameter. Other applicable parameters are summarized in the following Table. These parameters communicate design decisions from the

engineer to the program and thus allow the user to control the design process to suit an application's specific needs.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

Table 2-1: AISC 360-05 and 360-10 Design Parameters

Parameter Name	Default Value	Description
<u>CODE</u>	AISC UNIFIED	Used to designate this code (default is the 2010 edition). CODE AISC UNIFIED 200-5 CODE AISC UNIFIED 201-0
<u>AXIS</u>	1	1 = Design single angles for bending based on principle axis. 2 = Design single angles for bending based on geometric axis.
<u>BEAM</u>	1.0	0.0 = design at ends and those locations specified by the SECTION command. 1.0 = design at ends and at every 1/12 th point along member length.
<u>CAN</u>	0	0 = deflection check based on the principle that maximum deflection occurs within the span between DJ ₁ and DJ ₂ . 1 = deflection check based on the principle that maximum deflection is of the cantilever type (see note 2 on Table 2.1)

Section 2 American Steel Design

2.3 Steel Design per AISC 360 Unified Specification

Parameter Name	Default Value	Description
<u>CB</u> **	1.0	Coefficient C_b per Chapter F. If C_b is set to 0.0, it will be calculated by the program. Any other value will be directly used in the design.
<u>DF</u>	none (mandatory for deflection check)	"Deflection Length" / Maximum allowable local deflection
<u>DJ</u> ₁	Start Joint of member	Joint No. denoting starting point for calculation of "Deflection Length" (see note 1 on Table 2.1)
<u>DJ</u> ₂	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length" (see note 1 on Table 2.1)
<u>D</u> MAX	1000.0 mm	Maximum allowable depth for member selection.
<u>D</u> MIN	0.0 mm	Minimum allowable depth for member selection.
<u>F</u> U	400 MPa	Ultimate strength of steel.
<u>F</u> YLD	250 MPa	Yield strength of steel.
<u>K</u> X	1.0	K value for flexural-torsional buckling.
<u>K</u> Y	1.0	K value in local y-axis.
<u>K</u> Z	1.0	K value in local z-axis.
<u>L</u> X	Member Length	Length for flexural-torsional buckling
<u>L</u> Y	Member Length	Length to calculate slenderness ratio for buckling about local y-axis.

Parameter Name	Default Value	Description
<u>LZ</u>	Member Length	Length to calculate slenderness ratio for buckling about local z-axis.
<u>MAIN</u>	200	Allowable slenderness limit for compression members.
<u>METHOD</u>	LRFD	Used to specify LRFD or ASD design methods.
<u>NSF</u>	1.0	Net Section Factor for tension members.
<u>PROFILE</u>		Used in member selection. Refer to Section 5.48.1 for details.
<u>RATIO</u>	1.0	Permissible ratio of actual load to allowable strength.
<u>SLF</u>	1.0	Shear Lag Factor for determination of Net Effective Area of tension members.
<u>STP</u>	1.0	Section Type to determine F_r (compression residual stress in flange) 1.0 = Rolled section ($F_r = 10$ ksi) 2.0 = Welded section ($F_r = 16.5$ ksi)
<u>TMAIN</u>	300	Allowable slenderness limit for tension members.

Section 2 American Steel Design

2.3 Steel Design per AISC 360 Unified Specification

Parameter Name	Default Value	Description
<u>TRACK</u>	0	Specifies the amount of detail included in design output 0 = Suppress all member capacities 1 = Print all member capacities 2 = Print full member design details
<u>UNB</u>	Member Length	Unsupported length of the bottom* flange for calculating flexural strength. Will be used only if compression is in the bottom flange.
<u>UNT</u>	Member Length	Unsupported length of the top* flange for calculating flexural strength. Will be used only if compression is in the top flange.

* Top and Bottom represent the positive and negative side of the local Y axis (local Z axis if **SET Z UP** is used).

** Non-default values of CB must be re-entered before every subsequent **CHECK CODE** or **SELECT** command.

Notes

1. For a description of the deflection check parameters **DFF**, **DJ1**, **DJ2** see the Notes section of Table 2.1 of this manual.
2. **NSF** is the Net Section Factor as used in most of the Steel Design Codes of STAAD. It is defined as the Ratio of 'Net cross section area' / 'Gross section area' for tension member design. The default value is 1.0. For the AISC 360 code, it is described in section D.3.2.
3. **SLF** is the Shear Lag Factor, as used in Section D.3.3 of the AISC 360-05 code. This factor is used to determine the Effective Net Area by multiplying

this factor with Net Area of the cross section. Please refer to Table D3.1 of the 360 code for a list of acceptable SLF values. In STAAD, the default value for **SLF** is 1.0. The Effective Net Area is used to determine the Tensile Strength for Tensile rupture in the Net Section, as per equation D.2.2.

4. To summarize, the “Gross Area” (A_g) is multiplied by **NSF** to get the “Net Area” (A_n) of the section. The “Net Area” (A_n) is again multiplied by **SLF** to get the “Effective Net Area” (A_e) of the section.

2.3.5 Code Checking and Member Selection

Code Checking and Member Selection options are both available in the AISC 360 Unified Code implementation in STAAD.Pro. See "Code Checking" on page 131 and See "Member Selection" on page 132 for general information on these options. See "Code Checking Specification" on page 731 and See "Member Selection Specification" on page 732 for information on specification of these commands.

2.3.6 Tabulated Results of Steel Design

Results of Code Checking and Member Selection are presented in the output file. The output is clearly marked for the selected specification (ASIC 360), edition used (2010 or 2005), and the design method (LRFD or ASD).

The following details are presented on Code Checking of any member:

- Result of Code Checking (Pass / Fail) for the member Number.
- Critical Condition which governed the design and the corresponding Ratio and Location.
- Loads corresponding to the Critical Condition at the Critical Location.
- Section Classification
- Slenderness check report
- Section Capacities in Axial Tension, Axial Compression, Bending and Shear in both the directions.

2.4 Steel Design per AISC 9th Edition

2.4.1 Working Stress Design

2.4.1.1 Allowables per AISC Code

For steel design, STAAD compares the actual stresses with the allowable stresses as defined by the American Institute of Steel Construction (AISC) Code. The ninth edition of the AISC Code, as published in 1989, is used as the basis of this design (except for tension stress). Because of the size and complexity of the AISC codes, it would not be practical to describe every aspect of the steel design in this manual. Instead, a brief description of some of the major allowable stresses are described herein.

2.4.1.1.1 Tension Stress

Allowable tensile stress on the net section is calculated as;

$$F_t = 0.60 F_y$$

2.4.1.1.2 Shear Stress

Allowable shear stress on the gross section,

$$F_v = 0.4 F_y$$

2.4.1.1.3 Stress Due To Compression

Allowable compressive stress on the gross section of axially loaded compression members is calculated based on the formula E-1 in the AISC Code, when the largest effective slenderness ratio (Kl/r) is less than C_c . If Kl/r exceeds C_c , allowable compressive stress is decreased as per formula E2-2 of the Code.

$$C_c = \sqrt{\frac{2\pi^2 E}{F_y}}$$

2.4.1.1.4 Bending Stress

Allowable bending stress for tension and compression for a symmetrical member loaded in the plane of its minor axis, as given in Section 1.5.1.4 is:

$$F_b = 0.66 F_y$$

If meeting the requirements of this section of:

- a. $b_f/2t_f \leq 65/\sqrt{F_y}$
- b. $b_f/t_f \leq 190/\sqrt{F_y}$
- c. $d/t \leq 640(1 - 3.74(f_a/F_y))/\sqrt{F_y}$ when $(f_a/F_y) < 0.16$, or than $257/\sqrt{F_y}$ if $(f_a/F_y) > 0.16$
- d. The laterally unsupported length shall not exceed $76.0 b_f/F_y$ (except for pipes or tubes), nor $20,000/(d F_y/A_p)$
- e. The diameter:thickness ratio of pipes shall not exceed $3,300/\sqrt{F_y}$

If for these symmetrical members, $b_f/2t_f$ exceeds $65/\sqrt{F_y}$, but is less than $95/\sqrt{F_y}$, $F_b = F_y(0.79 - 0.002(b_f/2t_f)\sqrt{F_y})$

For other symmetrical members which do not meet the above, F_b is calculated as the larger value computed as per AISC formulas F1-6 or F1-7 and F1-8 as applicable, but not more than $0.60F_y$. An unstiffened member subject to axial compression or compression due to bending is considered fully effective when the width-thickness ratio is not greater than the following:

- $76.0/\sqrt{F_y}$, for single angles or double angles with separators
- $95.0/\sqrt{F_y}$, for double angles in contact
- $127.0/\sqrt{F_y}$, for stems of tees

When the actual width-thickness ratio exceeds these values, the allowable stress is governed by B_5 of the AISC code.

Tension and compression for the double symmetric (I & H) sections with $b_f/2t_f$ less than $65/\sqrt{F_y}$ and bent about their minor axis, $F_b = 0.75 F_y$. If $b_f/2t_f$ exceeds $65/\sqrt{F_y}$, but is less than $95/\sqrt{F_y}$, $F_b = F_y(1.075 - 0.005(b_f/2t_f)\sqrt{F_y})$.

For tubes, meeting the subparagraphs b and c of this Section, bent about the minor axis, $F_b = 0.66F_y$; failing the subparagraphs B and C but with a width:thickness ratio less than $238/\sqrt{F_y}$, $F_b = 0.6F_y$.

2.4.1.1.5 Combined Compression and Bending

Members subjected to both axial compression and bending stresses are proportioned to satisfy AISC formula H1-1 and H1-2 when f_a/F_a is greater than 0.15, otherwise formula H1-3 is used. It should be noted that during code checking or member selection, if f_a/F_a exceeds unity, the program does not compute the second and third part of the formula H1-1, because this would result in a misleadingly liberal ratio. The value of the coefficient C_m is taken as 0.85 for sidesway and $0.6 - 0.4 (M_1/M_2)$, but not less than 0.4 for no sidesway.

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

2.4.1.1.6 Singly Symmetric Sections

For double angles and Tees which have only one axis of symmetry, the KL/r ratio about the local Y-Y axis is determined using the clauses specified on page 3-53 of the AISC ASD 9th ed. Manual.

2.4.1.1.7 Torsion per Publication T114

The AISC 89 code of specifications for steel design currently does not have any provisions specifically meant for design of sections for Torsion. However, AISC has published a separate document called "Torsional Analysis of Steel Members" which provides guidelines on transforming torsional moments into normal stresses and shear stresses which can then be incorporated into the interaction equations explained in Chapter H of the AISC 89 code. The guidelines of the publication have now been incorporated into the AISC-89 steel design modules of STAAD.

To consider stresses due to torsion in the code checking or member selection procedure, specify the parameter **TORSION** with a value of 1.0. See Table 2.1 for more details.

Methodology

If the user were to request design for torsion, the torsional properties required for calculating the warping normal stresses, warping shear stresses and pure shear stresses are first determined. These depend of the "boundary" conditions that prevail at the ends of the member. These boundary conditions are defined as "Free", "Pinned" or "Fixed". They are explained below:

Free

represents the boundary condition such as that which exists at the free end of a cantilever beam. It means that there is no other member connected to the beam at that point.

Pinned

represents the condition that corresponds to either a pinned support defined at the joint through the Support command or a release of any of the moments at the joint through a Member Release Specification.

Fixed

represents the condition where a fixed support exists at the joint. In the absence of a support at that joint, it represents a condition where a rigid frame connection exists between the given member and at least one other member connected to that joint. Also, no member releases should be present at that joint on the given member.

After the boundary conditions are determined, the normal and shear stresses are determined. The guidelines specified in the publication T114 for concentrated torsional moments acting at the ends of the member are used to determine these stresses.

The warping normal stresses are added to the axial stresses caused by axial load. These are then substituted into the interaction equations in Chapter H of the AISC 89 code for determining the ratio. The plane shear and warping shear stresses are added to the shear stresses caused by actual shear forces and compared against the allowable shear stresses on the cross section.

Torsional boundary conditions at a joint where a FIXED BUT type of support is specified

If the end of a the member is declared a **FIXED BUT** type of support, the torsional boundary conditions at that end are determined in the following manner.

If the member framing into that support does not have any "member releases" specified at that node, then,

- a. If all of the 3 translational degrees of freedom at that support are either free to displace, or have a spring, then, that end of the member is considered torsionally **FREE**.

Example:

45 FIXED BUT MX MY MZ KFX 75 KFY 115

In this example, at joint 45, a spring has been specified along KFX and KFY, and, no restraint is provided for translation along global Z. So, the member which has joint 45 as one of its nodes is considered torsionally free at joint 45.

- b. If any of the 3 translational degrees of freedom at that support are restrained, and, any of the moment degrees of freedom are unrestrained or have a spring, then, that end of the member is considered torsionally **PINNED**.

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

Examples:

78 FIXED BUT FX MZ

In this example, joint 78 is prevented from translation along global Y and Z, and free to rotate about global Z. So, the member which has joint 78 as one of its nodes is considered torsionally PINNED at joint 78.

17 FIXED BUT MX MY

In this example, joint 17 is prevented from translation along global X, Y and Z, and free to rotate about global X and Y. So, the member which has joint 17 as one of its nodes is considered torsionally PINNED at joint 17.

85 FIXED BUT FZ MZ KFY 1.0E8 KMX 1.6E6

In this example, the joint is prevented from translation along global X, has a rotational spring for resisting moments about global X and is free to rotate about global Z. So, the member which has joint 85 as one of its nodes is considered torsionally PINNED at joint 85.

Restrictions

This facility is currently available for Wide Flange shapes (W, M & S), Channels, Tee shapes, Pipes and Tubes. It is not available for Single Angles, Double Angles, members with the **PRISMATIC** property specification, Composite sections (Wide Flanges with concrete slabs or plates on top), or Double Channels. Also, the stresses are calculated based on the rules for concentrated torsional moments acting at the ends of the member.

2.4.1.1.8 Design of Web Tapered Sections

Appendix F of AISC-89 provides specifications for design of Web-Tapered members. These specifications have been incorporated into STAAD to perform code checking on web tapered wide flange shapes. Please note that member selection cannot be performed on web-tapered members.

2.4.1.1.9 Slender Compression Elements

For cross sections with elements which fall in the category of slender as per Table B5.1 of the AISC ASD code (the others being compact and non-compact), the rules of Appendix B of the code have been implemented. For stiffened compression elements, the effective cross section properties are calculated and used. For unstiffened compression elements, the allowable stresses are reduced per the Appendix.

2.4.1.2 Design Parameters

The program contains a large number of parameter names which are needed to perform designing and code checking. These parameter names, with their default values, are listed in Table 2-3. These parameters communicate design decisions from the engineer to the program.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements of an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure. For example, by default the **KZ** (k value in local z-axis) value of a member is set to 1.0, while in the real structure it may be 1.5. In that case, the KZ value in the program can be changed to 1.5, as shown in the input instructions. Similarly, the **TRACK** value of a member is set to 0.0, which means no allowable stresses of the member will be printed. If the allowable stresses are to be printed, the **TRACK** value must be set to 1.0.

The parameters **PROFILE**, **DMAX**, and **DMIN** are only used for member selection.

Table 2-2: AISC (9th Ed.) Design Parameters

Parameter Name	Default Value	Description
<u>AXIS</u>	1	Select axis about which single angles are design 1 = Design single angles for bending about their principle axis. 2 = Design single angles for bending about their geometric axis.

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

Parameter Name	Default Value	Description
<u>BEAM</u>	1.0	Used to specify the number of sections at which the member design is evaluated. 0.0 = design at ends and those locations specified by the SECTION command. 1.0 = design at ends and at every 1/12 th point along member length.
<u>BMAX</u>	83.3333 ft	Maximum allowable width of the flange. Used in the design of tapered sections.
<u>CAN</u>	0	Specifies the method used for deflection checks 0 = deflection check based on the principle that maximum deflection occurs within the span between DJ ₁ and DJ ₂ . 1 = deflection check based on the principle that maximum deflection is of the cantilever type (see note 1)

Parameter Name	Default Value	Description
<u>CB</u>	1.0	Cb value as used in Section 1.5 of AISC. Use 0.0 to direct the program to calculated Cb. Any other value be used in lieu of the program calculated value.
<u>CDIA</u>	0.0	The diameter of circular openings. If a member has more than one circular opening, they can have different diameters.
<u>CHOLE</u>	NONE	Section locations of circular openings along the length of the member. Maximum three locations can be specified for each member when there is no rectangular opening.

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

Parameter Name	Default Value	Description
<u>CMP</u>	0	Composite action with connectors 0 = design as non-composite beam 1 = design as a composite beam if the slab is in bending compression throughout the span, design as a non-composite beam if the slab is in tension anywhere along the span 2 = design as a composite beam only. Ignore moments which cause tension in the slab.
<u>CMY</u> <u>CMZ</u>	0.85 for side-sway and calculated for no sidesway	Cm value in local y and z axes, respectively.
<u>CYC</u>	500,000	Cycles of maximum stress to which the shear connectors are subject.
<u>DFL</u>	none (mandatory for deflection check)	"Deflection Length" / Maximum allowable local deflection
<u>DIA</u>	0.625 in.	Diameter of the shear connectors

Parameter Name	Default Value	Description
<u>DINC</u>	1 in	Incremental depth value used in the design of tapered sections.
<u>DJ₁</u>	Start Joint of member	Joint No. denoting starting point for calculation of "Deflection Length" (see note 1)
<u>DJ₂</u>	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length" (see note 1)
<u>DMAX</u>	1000 in.	Maximum allowable section depth.
<u>DMIN</u>	0.0 in.	Minimum allowable section depth.
<u>DR₁</u>	0.4	Ratio of moment due to dead load applied before concrete hardens to total moment.
<u>DR₂</u>	0.4	Ratio of moment due to dead load applied after concrete hardens to total moment.
<u>ELECTRODE</u>	1	Weld material to be used for reinforced opening. 0 = E60XX 1 = E70XX 2 = E80XX 3 = E90XX 4 = E100XX 5 = E110XX

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

Parameter Name	Default Value	Description
<u>FBINC</u>	0	Incremental bottom flange width used in the design of tapered sections. In this case, the top flange width will remain unchanged.
<u>FLX</u>	1	Single angle member bracing 1 = Single angle member is <i>not</i> fully braced against lateral torsional buckling. 2 = Single angle member is fully braced against lateral torsional buckling. 3 = Single angle member is braced against lateral torsional buckling at the point of maximum moment.
<u>FPC</u>	3.0 ksi	Compressive strength of concrete at 28 days
<u>FSS</u>	1	Is the full section to be used for shear design? 0 = No (False) 1 = Yes (True)
<u>FTBINC</u>	0	Incremental flange width (top and bottom) used in the design of tapered sections.

Parameter Name	Default Value	Description
<u>FTINC</u>	0	Incremental top flange width used in the design of tapered sections. In this case, the bottom flange width will remain unchanged.
<u>FU</u>	Depends on FYLD	Ultimate tensile strength of steel in current units. <ul style="list-style-type: none"> • If $FYLD < 40$ KSI, then $FU = 58$ KSI • If $40 \text{ KSI} \leq FYLD \leq 50$ KSI, then $FU = 60$ KSI • If $FYLD > 50$ KSI, then $FU = 65$ KSI
<u>FYLD</u>	36 KSI	Yield strength of steel in current units.
<u>HECC</u>	0.0	Eccentricity of opening with respect to the centerline of the member.
<u>KX</u>	1.0	K value used in computing KL/r for flexural torsional buckling for tees and double angles.
<u>KY</u>	1.0	K value in local y-axis. Usually, this is the minor axis.
<u>KZ</u>	1.0	K value in local z-axis. Usually, this is the major axis.
<u>LX</u>	Member Length	Length value used in computing KL/r for flexural torsional buckling for tees and double angles.

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

Parameter Name	Default Value	Description
<u>LY</u>	Member Length	Length used to calculate slenderness ratio for buckling about the local y-axis.
<u>LZ</u>	Member Length	Same as LY, but in the local z-axis.
<u>MAIN</u>	0.0	Toggles the slenderness check 0.0 = check for slenderness 1.0 = suppress slenderness check Any value greater than 1 = Allowable KL/r in compression.
<u>NSF</u>	1.0	Net section factor for tension members.
<u>OVR</u>	1.0	Overstress factor. All the allowable stress are multiplied by this number. It may be assigned any value greater than 0.0. It is used to communicate increases in allowable stress for loads like wind and earthquake.
<u>PLTHICK</u>	0.0	Thickness of cover plate welded to the bottom flange of the composite beam.
<u>PLWIDTH</u>	0.0	Width of cover plate welded to the bottom flange of the composite beam.
<u>PROFILE</u>		Used in member selection. Refer to Section 5.48.1 for details.

Parameter Name	Default Value	Description
<u>RATIO</u>	1.0	Permissible ratio of actual to allowable stress.
<u>RDIM</u>	0.0	Dimensions of rectangular openings (at each section, RDIM has a length term and a depth term – see syntax below). If a member has more than one rectangular opening they can have different dimensions.
<u>RHOLE</u>	None	Section locations of rectangular openings along the length of the member. Maximum three locations can be specified for each member when there is no circular opening.
<u>RBHEIGHT</u>	0.0	Height of ribs in the form steel deck.
<u>RBWIDTH</u>	2.5 in.	Width of ribs in the form steel deck.
<u>SHE</u>	0	Option for calculating actual shear stress. 0 = Compute the shear stress using VO/Ib 1 = Computer the shear stress based on the area of the section element.

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

Parameter Name	Default Value	Description
<u>SHR</u>	0	Indicates use of temporary shoring during construction. 0 = Without shoring 1 = With shoring
<u>SSY</u>	0.0	Sidesway 0.0 = Sidesway in local y-axis. 1.0 = No sidesway
<u>SSZ</u>	0.0	Same as SSY, but in local z-axis.
<u>STIFF</u>	Member Length or depth of beam, whichever is greater	Spacing of stiffeners for plate girder design.
<u>STP</u>	1	Section type as defined in ASD Manual table. 1 = Rolled 2 = Welded

Parameter Name	Default Value	Description
<u>TAPER</u>	1.0	Design basis for tapered members 0.0 = Design tapered I-section based on rules of Chapter F and Appendix B of AISC only. Do not use the rules in Appendix F of AISC-89. 1.0 = Design tapered I-sections based on the rules of Appendix F of AISC-89.
<u>THK</u>	4.0 in.	Thickness of concrete slab or the thickness of concrete slab above the form steel deck.
<u>TMAIN</u>	300	Any value greater than 1 = Allowable KL/r in tension.
<u>TORSION</u>	0.0	Toggles the check for torsion 0.0 = No torsion check is performed 1.0 = Perform torsion check based on rules of AISC T114.
<u>TRACK</u>	0.0	Controls the level of detail to which results are reported: 0 = minimum detail 1 = intermediate detail 2 = maximum detail (see Figure 2.1)

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

Parameter Name	Default Value	Description
<u>UNB</u>	Member Length	Unsupported length of the bottom* flange for calculating allowable bending compressive stress. Will be used only if flexural compression is on the bottom flange
<u>UNT</u>	Member Length	Unsupported length of the top* flange for calculating allowable bending compressive stress. Will be used only if flexural compression is on the top flange.
<u>WELD</u>	1 for closed sections, 2 for open sections	Weld type, as explained in Section 2.12. A value of 1 will mean welding is on one side only except for wide-flange or tee sections, where the web is always assumed to be welded on both sides. A value of 2 will mean welding on both sides. For closed sections like a pipe or tube, the welding will be on one side only.
<u>WIDTH</u>	0.25 times the member length	Effective width of the concrete slab.
<u>WMAX</u>	See Section 2.12	Maximum welding thickness.
<u>WMIN</u>	See Section 2.12	Minimum welding thickness.
<u>WSTR</u>	0.4 x FYLD	Allowable weld stress.

*Top and Bottom represent the positive and negative side of the local Y axis (local Z axis if **SET Z UP** is used).

Notes

- When performing the deflection check, you can choose between two methods. The first method, defined by a value θ for the **CAN** parameter, is based on the local displacement. See "Printing Section Displacements for Members" on page 724 for details on local displacement.

If the **CAN** parameter is set to 1, the check will be based on cantilever style deflection. Let (DX_1, DY_1, DZ_1) represent the nodal displacements (in global axes) at the node defined by **DJ1** (or in the absence of **DJ1**, the start node of the member). Similarly, (DX_2, DY_2, DZ_2) represent the deflection values at **DJ2** or the end node of the member.

$$\text{Compute Delta} = \sqrt{(DX_2 - DX_1)^2 + (DY_2 - DY_1)^2 + (DZ_2 - DZ_1)^2}$$

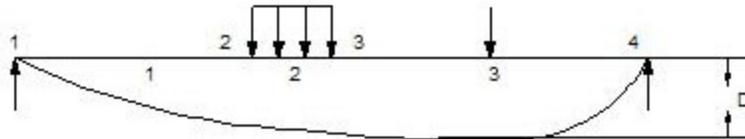
Compute Length = distance between **DJ1** and **DJ2** or, between start node and end node, as the case may be.

Then, if **CAN** is specified a value 1, $dff = L/\text{Delta}$

Ratio due to deflection = **DFD**/ dff

- If **CAN** = θ , deflection length is defined as the length that is used for calculation of local deflections within a member. It may be noted that for most cases the "Deflection Length" will be equal to the length of the member. However, in some situations, the "Deflection Length" may be different.

For example, refer to the figure below where a beam has been modeled using four joints and three members. The "Deflection Length" for all three members will be equal to the total length of the beam in this case. The parameters **DJ1** and **DJ2** should be used to model this situation. Also the straight line joining **DJ1** and **DJ2** is used as the reference line from which local deflections are measured. Thus, for all three members here, **DJ1** should be "1" and **DJ2** should be "4".



D is equal to the maximum local deflection for members 1, 2, and 3.

PARAMETERS

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

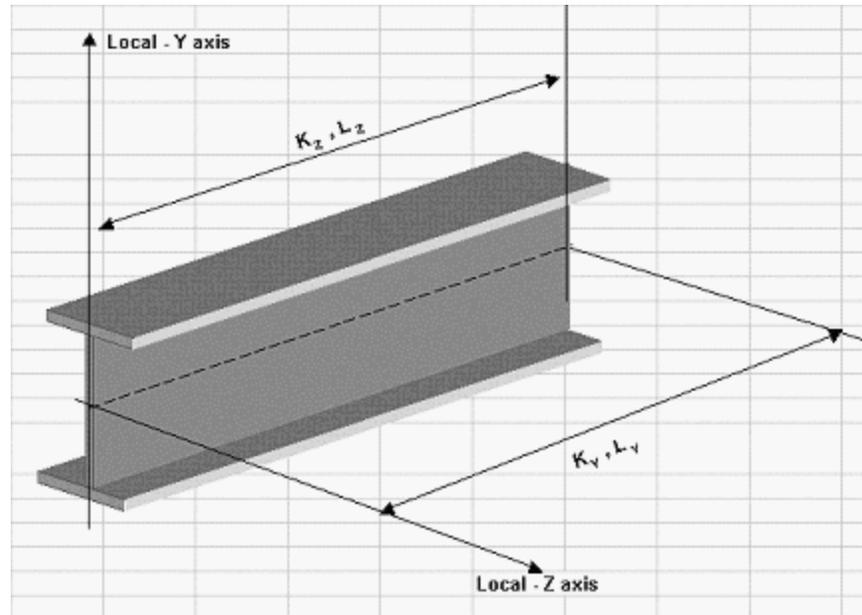
DFF 300. ALL

DJ1 1 ALL

DJ2 4 ALL

3. If **DJ1** and **DJ2** are not used, "Deflection Length" will default to the member length and local deflections will be measured from original member line.
4. It is important to note that unless a **DFF** value is specified, STAAD will not perform a deflection check. This is in accordance with the fact that there is no default value for **DFF**.
5. A critical difference exists between the parameters **UNT/UNB** and the parameters **LY** and **LZ**. Parameters **UNT** and **UNB** represent the laterally unsupported length of the compression flange. It is defined in Chapter F, page 5-47 of the specifications in the AISC 1989 ASD manual as the distance between cross sections braced against twist or lateral displacement of the compression flange. These parameters are used to calculate the allowable compressive stress (**FCZ** and **FCY**) for behavior as a beam. Parameters **LY** and **LZ** are the unbraced lengths for behavior as a column and are used to calculate the KL/r ratios and the allowable axial compressive stress F_A .
6. Parameters **SSY** and **CMY** are based upon two values defined in page 5-55, Chapter H of the AISC 9th ed. manual. **SSY** is a variable which allows the user to define whether or not the member is subject to sidesway in the local Y direction. **CMY** is a variable used for defining the expression called C_m in the AISC manual. When **SSY** is set to 0 (which is the default value), it means that the member is subject to sidesway in the local Y direction. When **SSY** is set to 1.0, it means that the member is not subject to sidesway in the local Y direction. The only effect that **SSY** has is that it causes the program to calculate the appropriate value of **CMY**. If **SSY** is set to 0 and **CMY** is not provided, STAAD will assume **CMY** as 0.85. If **SSY** is set to 1 and **CMY** is not provided, STAAD will calculate **CMY** from the equation on page 5-55. However, if the user provides **CMY**, the program will use that value and not calculate **CMY** at all, regardless of what the user defines **SSY** to be.

Figure 2-1: Terms used in calculating slenderness ratios KL/r for local Y and Z axes



7. For a T shape which is cut from a parent I, W, S, M or H shapes, the **PROFILE** parameter should be assigned a value corresponding to the parent shape. For example, if the T desired is an American WT6, specify W12 for the **PROFILE** parameter.

2.4.1.3 Code Checking

The purpose of code checking is to check whether the provided section properties of the members are adequate. The adequacy is checked as per AISC-89. Code checking is done using the forces and moments at specified sections of the members. If no sections are specified, the program uses the start and end forces for code checking.

When code checking is selected, the program calculates and prints whether the members have passed the code or have failed; the critical condition of the AISC code (like any of the AISC specifications or compression, tension, shear, etc.); the value of the ratio of the critical condition (overstressed for a value more than 1.0 or any other specified **RATIO** value); the governing load case, and the location (distance from the start of the member) of forces in the member where the critical condition occurs.

Code checking can be done with any type of steel section listed in Section 2.2.1 of this manual.

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

2.4.1.4 Member Selection

STAAD is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section (i.e., the lightest section) which fulfills the code requirements for the specified member. The section selected will be of the same type section as originally designated for the member being designed. A wide flange will be selected to replace a wide flange, etc. Several parameters are available to guide this selection. If the **PROFILE** parameter is provided, the search for the lightest section is restricted to that profile. Up to three (3) profiles may be provided for any member with a section being selected from each one. Member selection can also be constrained by the parameters **DMAX** and **DMIN** which limit the maximum and minimum depth of the members. If the **PROFILE** parameter is provided for specified members, **DMAX** or **DMIN** parameters will be ignored by the program in selecting these members.

Member selection can be performed with all the types of steel sections listed in Section 2.2.1 of this manual. Note that for beams with cover plates, the sizes of the cover plate are kept constant while the beam section is iterated.

Selection of members, whose properties are originally input from a user created table, will be limited to sections in that table.

Member selection *cannot* be performed on members whose section properties are input as prismatic.

2.4.1.4.1 Member Selection by Optimization

Steel table properties of an entire structure can be optimized by STAAD. This optimization method involves the following steps.

1. **CHECK CODE ALL**
2. Modify the ratios
3. **SELECT ALL**
4. **PERFORM ANALYSIS**
5. **SELECT ALL**

An additional step of grouping may be performed if the **FIXED GROUP** and **GROUP** commands are provided (See "Group Specification" on page 734). After the last step, a re-analysis is not automatically performed, so users must ensure that they specify the analysis command following the **SELECT OPTIMIZE** command.

2.4.1.4.2 Deflection Check With Steel Design

This facility allows the user to consider deflection as a criteria in the **CODE CHECK** and **MEMBER SELECTION** processes. The deflection check may be controlled using the three parameters **DJ1**, **DJ2**, and **DFF** which are described in Table 2.1.

Deflection is used in addition to other strength and stability related criteria. The local deflection calculation is based on the latest analysis results.

2.4.1.5 Truss Members

As described in Section 1.9, a truss member is capable of carrying only axial forces. So during the design phase, no calculation time (or, matrix bandwidth) is wasted determining the allowable bending or shear stresses. Therefore, if there is any truss member in an analysis (such as a bracing or a strut, etc.), it is advisable to declare it as a truss member rather than as a regular frame member with both ends pinned.

2.4.1.6 Unsymmetric Sections

For unsymmetric sections like single angles, STAAD considers the smaller section modulus for calculating bending stresses.

For single angles, the “specification for allowable stress design of single-angle members”, explained in pages 5-309 to 5-314 of the AISC-ASD 9th edition manual has been incorporated.

2.4.1.7 Composite Beam Design as per AISC-ASD

In Section 1.7.7 of this manual, two methods of specifying the properties of a beam as a composite section (I-shaped beam with concrete slab on top) are described. Those members can be designed as composite beams in accordance with the AISC ASD code provisions. If the properties are assigned using the explicit method as defined in Section 1.7.7, the design parameters must be separately assigned. The **CMP** parameter in particular must be set a value of 1 or 2. If the properties are derived from the composite decks, the design parameters are automatically generated during the deck creation phase, and hence no separate parameters need to be assigned.

Other parameters used in the design of composite members are: **DIA**, **DR1**, **DR2**, **FPC**, **HGT**, **PLTHICK**, **PLWIDTH**, **RBHEIGHT**, **RBWIDTH**, **SHR**, **THK**, and **WIDTH**. Refer to Table 2-3 for details on design parameters.

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

Example

```
UNIT INCH
PARAMETER
CODE AISC
BEAM 1 ALL
TRACK 2 ALL
DR1 0.3135 ALL
WID 69.525 ALL
FPC 3.0 ALL
THK 4.0 ALL
CMP 1 ALL
CHECK CODE ALL
SELECT ALL
```

2.4.1.8 Plate Girders

The requirements of Chapter G – pages 5-51 through 5-53 of the AISC ASD 9th edition manual – are not implemented. Therefore, if the web slenderness ratio h/t_w of a section exceeds $970/\sqrt{F_y}$, STAAD will not perform a design for that member.

2.4.1.9 Tabulated Results of Steel Design

For code checking or member selection, the program produces the results in a tabulated fashion. The items in the output table are explained as follows:

- a. **MEMBER** refers to the member number for which the design is performed.
- b. **TABLE** refers to the AISC steel section name which has been checked against the steel code or has been selected.
- c. **RESULT** prints whether the member has **PASSED** or **FAILED**. If the **RESULT** is **FAIL**, there will be an asterisk (*) mark in front of the member number.
- d. **CRITICAL COND** refers to the section of the AISC code which governed the design.

proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized--ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD implementation of LRFD, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks that code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the LRFD specifications as implemented in STAAD steel design. A detailed description of the design process along with its underlying concepts and assumptions is available in the LRFD manual. However, since the design philosophy is drastically different from the conventional Allowable Stress Design (ASD), a brief description of the fundamental concepts is presented here to initiate the user into the design process.

2.4.2.2 LRFD Fundamentals

The primary objective of the LRFD Specification is to provide a uniform reliability for all steel structures under various loading conditions. This uniformity can not be obtained with the allowable stress design (ASD) format.

The ASD method can be represented by the inequality

$$\Sigma Q_i < R_n / F.S.$$

The left side is the required strength, which is the summation of the load effects, Q_i (forces and moments). The right side, the design strength, is the nominal strength or resistance, R_n , divided by a factor of safety. When divided by the appropriate section property (area or section modulus), the two sides of the inequality become the actual stress and allowable stress respectively. ASD, then, is characterized by the use of unfactored "working" loads in conjunction with a single factor of safety applied to the resistance. Because of the greater variability and, hence, unpredictability of the live load and other loads in comparison with the dead load, a uniform reliability is not possible.

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

LRFD, as its name implies, uses separate factors for each load and resistance. Because the different factors reflect the degree of uncertainty of different loads and combinations of loads and of the accuracy of predicted strength, a more uniform reliability is possible. The LRFD method may be summarized by the inequality

$$\sum y_i Q_i < R_n \phi$$

On the left side of the inequality, the required strength is the summation of the various load effects, Q_i , multiplied by their respective load factors, y_i . The design strength, on the right side, is the nominal strength or resistance, R_n , multiplied by a resistance factor, ϕ .

In the STAAD implementation of LRFD, it is assumed that the user will use appropriate load factors and create the load combinations necessary for analysis. The design portion of the program will take into consideration the load effects (forces and moments) obtained from analysis. In calculation of resistances of various elements (beams, columns etc.), resistance (nominal strength) and applicable resistance factor will be automatically considered.

2.4.2.3 Analysis Requirements

The types of construction recognized by AISC specification have not changed, except that both "simple framing" (formerly Type 2) and "semi-rigid framing" (formerly Type 3) have been combined into the same category, Type PR (partially restrained). "Rigid Framing" (formerly Type 1) is now Type FR (fully restrained). Type FR construction is permitted unconditionally. Type PR construction may necessitate some inelastic, but self-limiting, deformation of a structural steel element. Thus, when specifying Type PR construction, the designer should take into consideration the effects of partial restraint on the stability of the structure, lateral deflections and second order bending moments. As stated in Sect. C1 of the LRFD specification, an analysis of second order effects is required. Thus, when using LRFD code for steel design, the user must use the P-Delta analysis feature of STAAD.

2.4.2.4 Section Classification

The LRFD specification allows inelastic deformation of section elements. Thus local buckling becomes an important criterion. Steel sections are classified as compact, noncompact or slender element sections depending upon their local buckling characteristics. This classification is a function of the geometric properties of the section. The design procedures are different depending on the

section class. STAAD is capable of determining the section classification for the standard shapes and user specified shapes and design accordingly.

2.4.2.5 Limit States

2.4.2.5.1 Axial Tension

The criteria governing the capacity of tension members is based on two limit states. The limit state of yielding in the gross section is intended to prevent excessive elongation of the member. The second limit state involves fracture at the section with the minimum effective net area. The net section area may be specified by the user through the use of the parameter **NSF** (see Table 2.6). STAAD calculates the tension capacity of a given member based on these two limit states and proceeds with member selection or code check accordingly.

2.4.2.5.2 Axial Compression

The column strength equations have been revised in LRFD to take into account inelastic deformation and other recent research in column behavior. Two equations governing column strength are available, one for inelastic buckling and the other for elastic or Euler buckling. Both equations include the effects of residual stresses and initial out-of-straightness. Compression strength for a particular member is calculated by STAAD according to the procedure outlined in Chapter E of the LRFD specifications. For slender elements, the procedure described in Appendix B5.3 is used.

Singly symmetric and unsymmetric compression members are designed on the basis of the limit states of flexural-torsional and torsional buckling. The procedure of Appendix E3 is implemented for the determination of design strength for these limit states.

Effective length for calculation of compression resistance may be provided through the use of the parameters KY, KZ and/or LY, LZ. If not provided, the entire member length will be taken into consideration.

In addition to the compression resistance criterion, compression members are required to satisfy slenderness limitations which are a function of the nature of use of the member (main load resisting component, bracing member, etc.). In both the member selection and code checking process, STAAD immediately does a slenderness check on appropriate members before continuing with other procedures for determining the adequacy of a given member.

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

2.4.2.5.3 Flexural Design Strength

In LRFD, the flexural design strength of a member is determined by the limit state of lateral torsional buckling. Inelastic bending is allowed and the basic measure of flexural capacity is the plastic moment capacity of the section. The flexural resistance is a function of plastic moment capacity, actual laterally unbraced length, limiting laterally unbraced length, buckling moment and the bending coefficient. The limiting laterally unbraced length L_r and buckling moment M_r are functions of the section geometry and are calculated as per the procedure of Chapter F. The purpose of bending coefficient C_b is to account for the influence of the moment gradient on lateral-torsional buckling. This coefficient can be specified by the user through the use of parameter **CB** (see Table 2.6) or may be calculated by the program (if **CB** is specified as 0.0). In the absence of the parameter **CB**, a default value of 1.0 will be used. The procedure for calculation of design strength for flexure also accounts for the presence of residual stresses of rolling. To specify laterally unsupported length, either of the parameters **UNB** and **UNT** (see Table 2.6) can be used.

2.4.2.5.4 Combined Axial Force and Bending

The interaction of flexure and axial forces in singly and doubly symmetric shapes is governed by formulas H1-1a and H1-1b. These interaction formulas cover the general case of biaxial bending combined with axial force. They are also valid for uniaxial bending and axial force.

2.4.2.5.5 Design for Shear

The procedure of Sect. F2 of the LRFD Specification is used in STAAD to design for shear forces in members. Shear strength as calculated in LRFD is governed by the following limit states: Eq. F2-1a by yielding of the web; Eq. F2-2a by inelastic buckling of the web; Eq. F2-3a by elastic buckling of the web. Shear in wide flanges and channel sections is resisted by the area of the web, which is taken as the overall depth times the web thickness.

2.4.2.6 Design Parameters

Design per LRFD specifications is requested by using the **CODE** parameter. Other applicable parameters are summarized in Table 2-4. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

The parameters **DMAX** and **DMIN** are applicable for member selection only.

Table 2-3: AISC LRFD (2nd and 3rd Ed.) Design Parameters

Parameter Name	Default Value	Description
<u>AXIS</u>	1	1 = Design single angles for bending about their principle axis. 2 = Design single angles for bending about their geometric axis.
<u>BEAM</u>	1.0	0.0 = design at ends and those locations specified by the SECTION command. 1.0 = design at ends and at every 1/12 th point along member length.
<u>CAN</u>	0	0 = deflection check based on the principle that maximum deflection occurs within the span between DJ ₁ and DJ ₂ . 1 = deflection check based on the principle that maximum deflection is of the cantilever type (see note 2 of Table 2.1)
<u>CB**</u>	1.0	Coefficient C _b per Chapter F of AISC LRFD. If C _b is set to 0.0, it will be calculated by the program. Any of value will be used directly in design.
<u>DFE</u>	None (Mandatory for deflection check)	"Deflection Length" / Maximum allowable local deflection

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

Parameter Name	Default Value	Description
<u>DJ1</u>	Start Joint of Member	Joint No. denoting starting point for calculation of "Deflection Length" (see note 1 of Table 2.1)
<u>DJ2</u>	End Joint of Member	Joint No. denoting end point for calculation of "Deflection Length" (see note 1 of Table 2.1)
<u>DMAX</u>	45.0 in.	Maximum allowable section depth.
<u>DMIN</u>	0.0 in.	Minimum allowable section depth
<u>FLX</u>	1	1 = Single angle member is <i>not</i> fully braced against lateral torsional buckling. 2 = Single angle member is fully braced against lateral torsional buckling. 3 = Single angle member is braced against lateral torsional buckling at the point of maximum moment.
<u>FYLD</u>	36.0 ksi	Yield strength of steel.
<u>FU</u>	60.0 ksi	Ultimate tensile strength of steel.
<u>KX</u>	1.0	K value for flexural-torsional buckling.
<u>KY</u>	1.0	K value in the local y-axis. Usually this is the minor axis.
<u>KZ</u>	1.0	K value in the local z-axis. Usually this is the major axis.
<u>LX</u>	Member Length	Length used for flexural-torsional buckling.

Parameter Name	Default Value	Description
<u>LY</u>	Member Length	Length to calculate the slenderness ratio for buckling about the local y-axis.
<u>LZ</u>	Member Length	Length to calculate the slenderness ratio for buckling about the local z-axis.
<u>MAIN</u>	0.0	0.0 = check for slenderness 1.0 = suppress slenderness check Any value greater than 1 = Allowable KL/r in compression.
<u>NSF</u>	1.0	Net section factor for tension members.
<u>PROFILE</u>		Used in member selection. Refer to Section 5.48.1 for details.
<u>RATIO</u>	1.0	Permissible ratio of actual load effect to design strength.
<u>STIFF</u>	Member Length or depth, whichever is greater	Spacing of stiffeners for beams for shear design.
<u>STP</u>	1	Section type to determine F_r (compressive residual stress in flange) per 3rd Ed. LRFD spec., p 16.1-97. 1 = Rolled section ($F_r = 10$ ksi) 2 = Welded section ($F_r = 16.5$ ksi)

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

Parameter Name	Default Value	Description
<u>TMAIN</u>	300	Any value greater than 1 = Allowable KL/r in tension.
<u>TRACK</u>	0.0	Specified the level of detail included in the output. 0.0 = Suppress all design strengths 1.0 = Print all design strengths 2.0 = Print expanded design output
<u>UNB</u>	Member Length	Unsupported length (L_h) of the bottom* flange for calculating flexural strength. Will be used only if flexural compression is on the bottom flange.
<u>UNT</u>	Member Length	Unsupported length (L_h) of the top* flange for calculating flexural strength. Will be used only if flexural compression is on the top flange.

*Top and Bottom represent the positive and negative side of the local Y axis (local Z axis if **SET Z UP** is used).

Note: For a description of the deflection check parameters **DFF**, **DJ1**, **DJ2**, and **CAN**, see the Notes section of Table 2.1 of this manual.

The **STIFF** parameter represents the term “a” as defined in Section F2, page 6-113 of the LRFD 2nd edition manual.

** Non-default values of CB must be re-entered before every subsequent **CHECK CODE** or **SELECT** command.

2.4.2.7 Code Checking and Member Selection

Both code checking and member selection options are available in **STAAD LRFD** implementation. See "Code Checking" on page 131 and See "Member Selection" on page 132 for general information on these options. See "Code Checking Specification" on page 731 and See "Member Selection Specification" on page 732 for information on specification of these commands.

Example for the LRFD-2001 code

```
UNIT KIP INCH
PARAMETER
CODE LRFD
FYLD 50 ALL
UNT 72 MEMBER 1 TO 10
UNB 72 MEMB 1 TO 10
MAIN 1.0 MEMB 17 20
SELECT MEMB 30 TO 40
CHECK CODE MEMB 1 TO 30
```

Example for the LRFD-1994 code

```
UNIT KIP INCH
PARAMETER
CODE LRFD2
FYLD 50 ALL
UNT 72 MEMBER 1 TO 10
UNB 72 MEMB 1 TO 10
MAIN 1.0 MEMB 17 20
SELECT MEMB 30 TO 40
CHECK CODE MEMB 1 TO 30
```

2.4.2.8 Tabulated Results of Steel Design

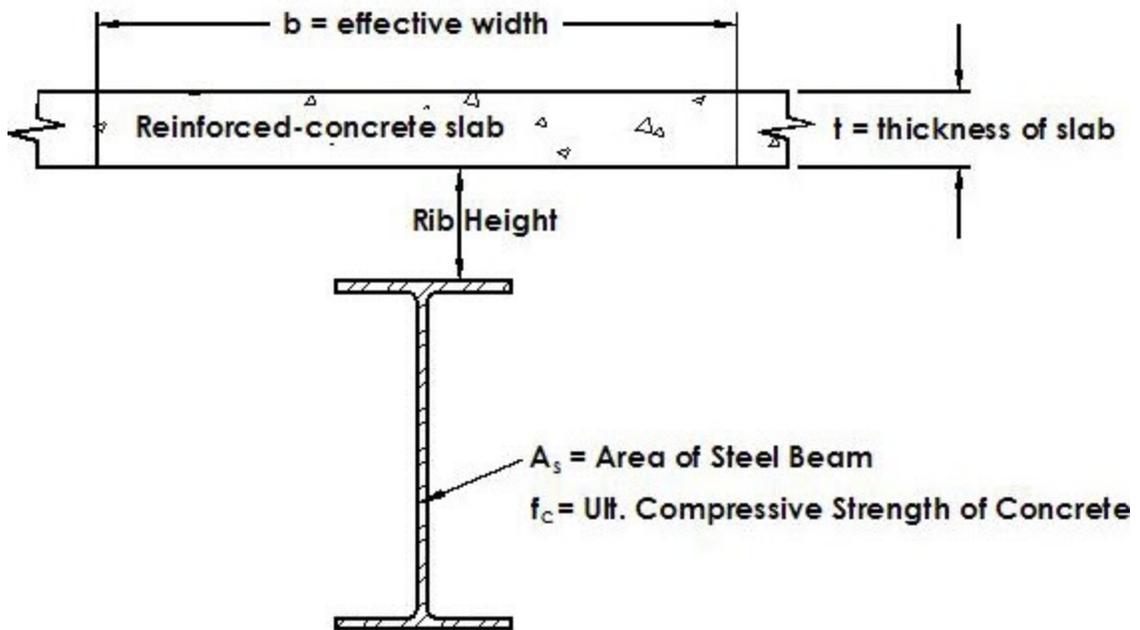
Results of code checking and member selection are presented in a tabular format. A detailed discussion of the format is provided in Section 2.11. Following exceptions may be noted: **CRITICAL COND** refers to the section of the LRFD specifications which governed the design.

If the **TRACK** is set to 1.0, member design strengths will be printed out.

2.4.2.9 Composite Beam Design per the American LRFD 3rd edition code

The design of composite beams per the 3rd edition of the American LRFD code has been implemented. The salient points of this feature are as follows:

Figure 2-2: Nomenclature of composite beams



Theoretical Basis

1. Find the maximum compressive force carried by concrete as:

$$0.85 f_c \cdot b \cdot t$$

2. Find the maximum tensile force carried by the steel beam as:

$$A_s \cdot f_y$$

Tensile strength of concrete is ignored.

3. If step 1 produces a higher value than step 2, plastic neutral axis (PNA) is in the slab. Else, it is in the steel beam.

Location of the Plastic Neutral Axis (PNA) defines the moment capacity:

- Case 1: PNA in the slab

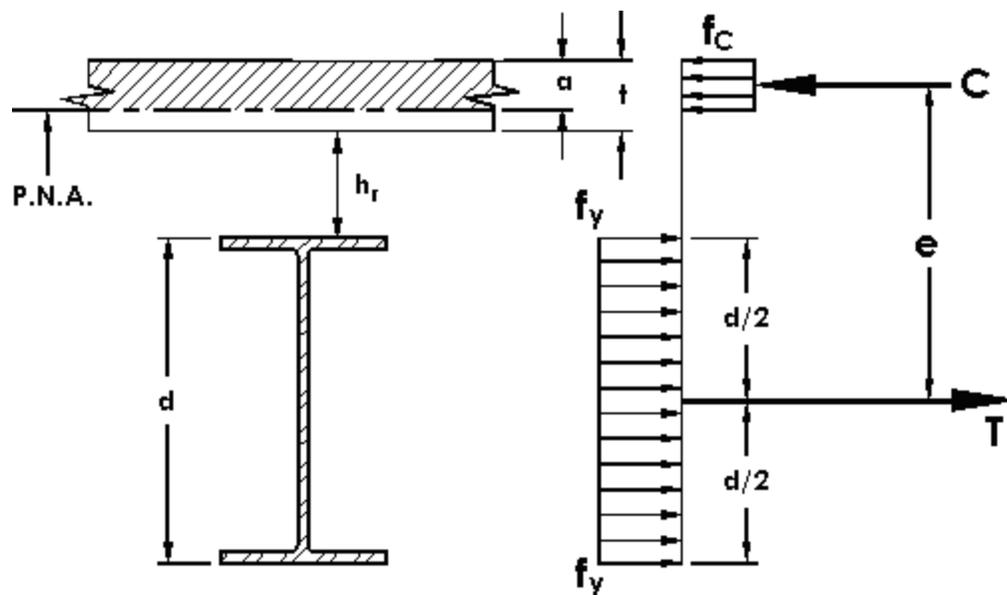
Find the depth of the PNA below the top of the slab as:

$$0.85f_c \cdot b \cdot a = A_s \cdot f_y$$

Rearranging terms:

$$a = A_s \cdot f_y / (0.85f_c \cdot b)$$

Figure 2-3: Plastic neutral axis in the concrete slab



Lever arm

$$e = d/2 + h_r + t - a/2$$

Moment Capacity

$$\phi_b (A_s \cdot f_y) e$$

- Case 2: PNA in Steel Beam

Define:

$$C_s = \text{Compressive force in slab} = 0.85f_c \cdot b \cdot t$$

$$C_b = \text{Compressive force in steel beam}$$

$$T_b = \text{Tensile force in steel beam}$$

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

$$C_s + C_b = T_b$$

Since the magnitude of C_b + magnitude of $T_b = A_s \cdot f_y$

Substituting for T_b as $(A_s \cdot f_y - C_b)$ gives:

$$C_s + C_b = A_s \cdot f_y - C_b$$

Rearranging terms:

$$C_b = (A_s \cdot f_y - C_s)/2$$

Determine whether the PNA is within the top flange of the steel beam or inside the web:

$$C_f = \text{Maximum compressive force carried by the flange} = A_f \cdot f_y$$

Where:

$$A_f = \text{Area of the flange}$$

If $C_f \geq C_b$, the PNA lies within the flange (Case 2A)

If $C_f < C_b$, the PNA lies within the web (Case 2B)

- Case 2A: PNA in Flange of Steel Beam

Calculate:

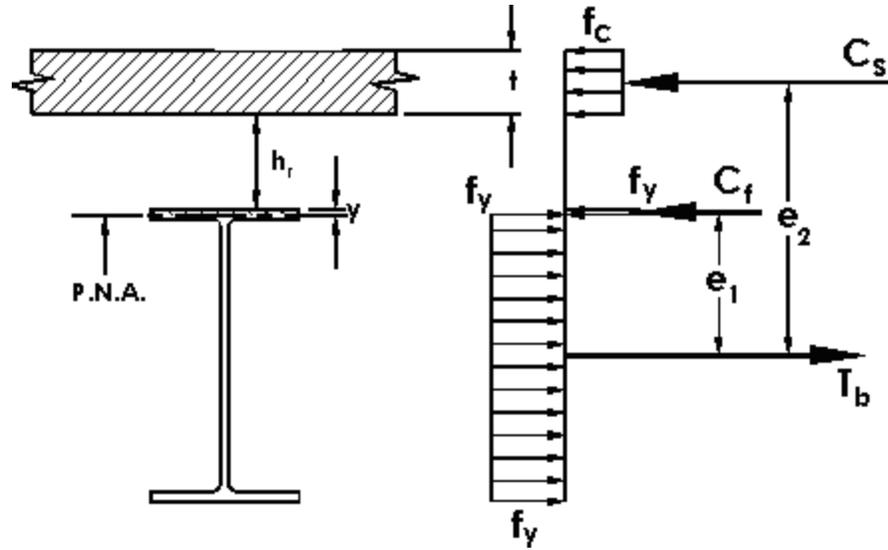
$$y = C_f / (b_f \cdot f_y)$$

Where:

$$b_f = \text{width of the flange}$$

The point of action of the tensile force is the centroid of the steel area below the PNA. After finding that point, e_1 and e_2 can be calculated.

Figure 2-4: Plastic neutral axis falls within the top flange

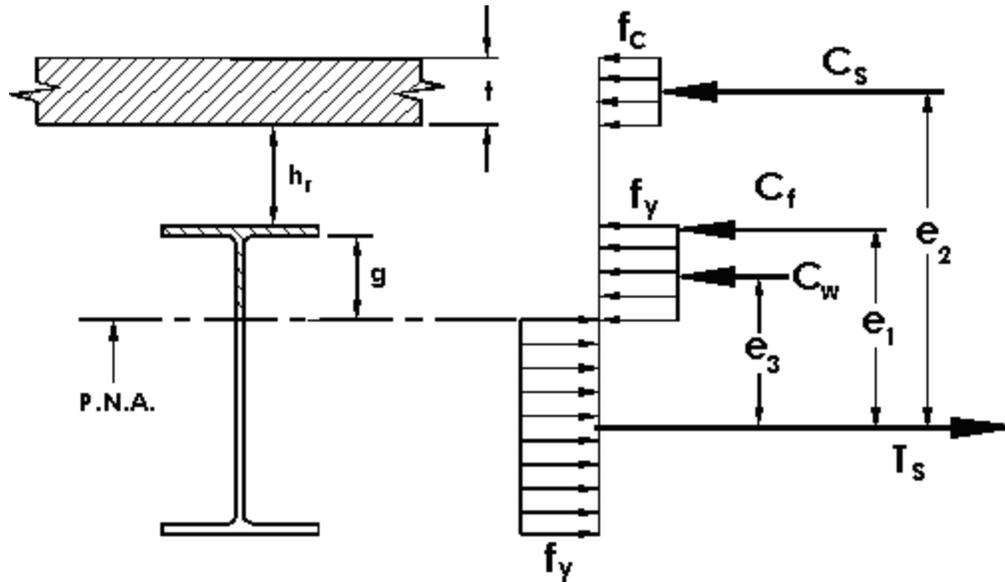


Moment Capacity

$$\phi_b (C_f \cdot e_1 + C_s \cdot e_2)$$

- Case 2B: PNA in Web of Steel Beam

Figure 2-5: Plastic neutral axis falls within the web



C_w = Compressive force in the web = $C_b - C_f$

$$g = C_w / (t_w \cdot f_y)$$

Where:

t_w = thickness of the web

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

The point of action of the tensile force is the centroid of the steel area below the PNA. After finding that point, e_1 , e_2 , and e_3 can be calculated.

Moment Capacity

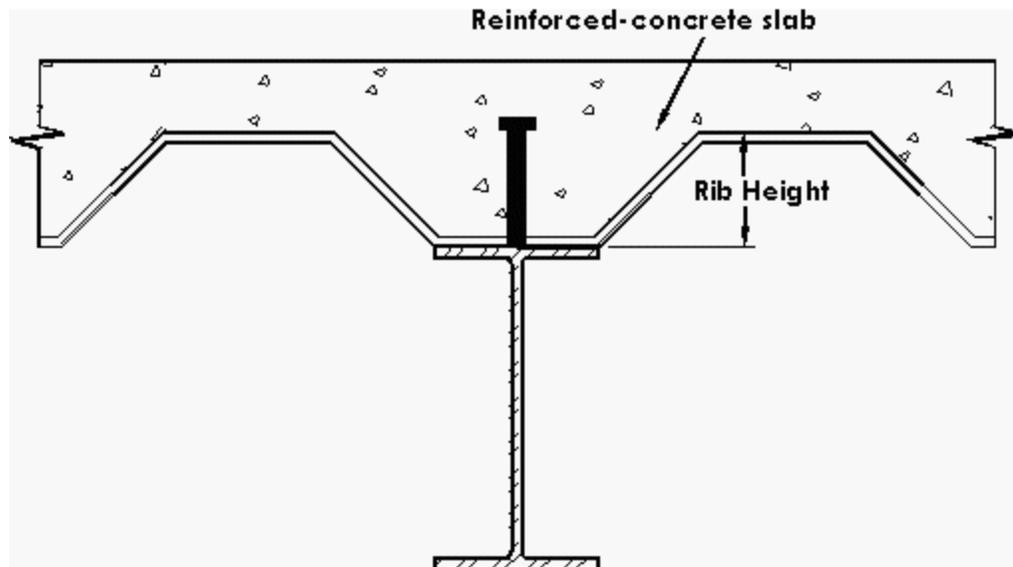
$$\phi_b (C_s \cdot e_2 + C_f \cdot e_1 + C_w \cdot e_3)$$

Utilization Ratio = Applied Moment / Moment Capacity

Notes

1. Rib Height is the distance from top of flange of steel beam to lower surface of concrete.
2. If the slab is flush on top of the steel beam, set the Rib Height to zero.

Figure 2-6: Steel deck form ribs



3. For moments which cause tension in the slab (called positive moments in STAAD convention), design of the beam is presently not carried out.
4. Shear connectors are presently not designed.
5. Member selection is presently not carried out.
6. In order to design a member as a composite beam, the member property specification during the analysis phase of the data must contain the "CM" attribute. See "Assigning Properties from Steel Tables" on page 341 for details.

Table 2-4: Composite Beam Design Parameters for AISC-LRFD

Parameter Name	Default Value	Description
<u>RBH</u>	0.0 in.	Rib height for steel form deck.
<u>EFFW</u>	Value used in analysis	Effective width of the slab.
<u>FPC</u>	Value used in analysis	Ultimate compressive strength of the concrete slab.

Example

```

STAAD SPACE
...
MEMBER PROPERTY
1 TA CM W12X26 CT 6.0 FC 4.0 CW 40.0
...
PERFORM ANALYSIS
...
PARAMETER
CODE LRFD
RBH 5.0 MEMB 1
CHECK CODE MEMB 1
FINISH

```

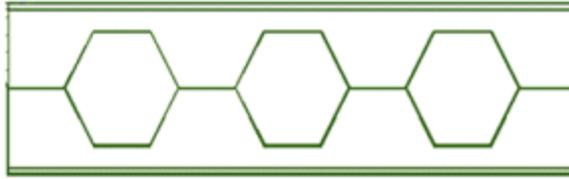
2.4.3 Castellated Beams

STAAD.Pro comes with the non-composite castellated beam tables supplied by the steel products manufacturer SMI Steel Products. Details of the manufacture and design of these sections may be found at the SMI Steel Product's web site.

Figure 2-7: Castellated beam elevation

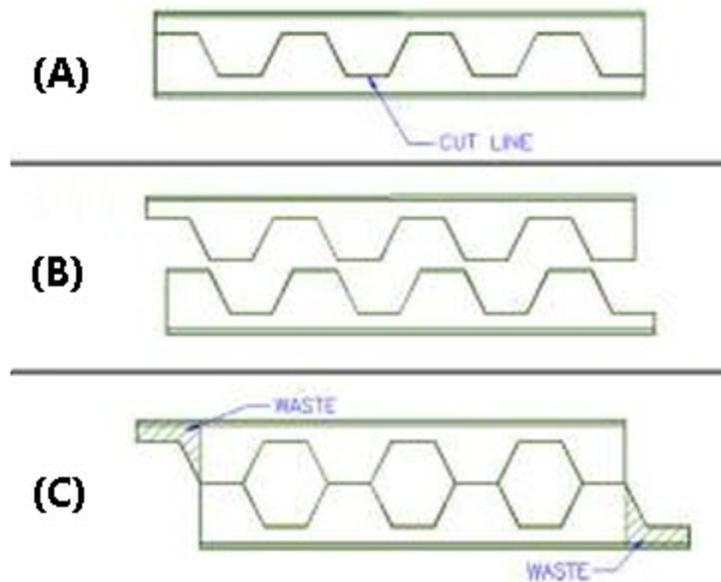
Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition



According to the manufacturer, castellated beams are manufactured by cutting a wide flange along the web in a zig-zag pattern, offsetting the two halves, and welding the two halves together, as shown in the next figure. As a result, the underlying steel section is a wide flange (W shapes) in the AISC table or a B shape. STAAD currently supports only the ones derived from W shapes.

Figure 2-8: Manufacturing process for castellated beams: A) Cut hexagonal line, B) Stagger top and bottom sections, and C) discard waste at ends



STAAD.Pro's design of castellated beams is based on the information gathered from the following sources:

- Design of Welded Structures -- Omer W. Blodget, published by The James Lincoln Arc Welding Foundation, pages 4.7-8 and 4.7-9
- AISC 9th edition manual – Allowable stress design
- ASCE Journal of Structural Engineering 124:10, October 1998 – castellated beam web buckling in shear, R.G. Redwood and S. Demirdjian

2.4.3.1 Analysis and Design Criteria

The local axis system (local X, local Y and local Z) of a castellated beam is identical to that for a wide flange, and is shown in Section 1.5.2.

It is important recognize that there are two basic issues to be understood with regard to these members a) analysis b) steel design

First, the design issues because only then will their relationship with the analysis issues become apparent. Design of a castellated beam is done only for FY (shear along the web) and MZ (moment about the major axis which is the Z axis). If at the start of the design process, the program detects that the beam has axial force (FX), shear along local-Z (FZ), torsion (MX) or moment about the minor axis (MY), design of that member will be terminated.

Next is how these design limitations have a bearing on the analysis issues. If you intend to design these members, as a result of the above restrictions, he/she must model it in such a way that none of the 4 unacceptable degrees of freedom end up with a non-zero value anywhere along the length of the member. That means, if the member ends are defined as supports, the support conditions must be defined with the above in mind. Similarly, if the castellated member is attached to other members, its end conditions (MEMBER RELEASES) must be modeled taking the above facts into consideration.

The design limitations also have a bearing on the type of loads that are applied to the member. Loads which cause any of the above-mentioned four degrees of freedom to end up with a non-zero value will cause the member design to be terminated.

However, if you wish to only analyze the structure, and are not interested in performing a steel design, the above described restrictions for supports, member end conditions or loading are not applicable.

The design method is the allowable stress method, using mainly the rules stated in the AISC ASD 9th edition code. Only code checking is currently available for castellated beams. Member selection is not.

Note: STAAD does not multiply the analysis moment by 1.7 for ASD method. It is upto you to multiply the dead and live loads by 1.7 in load combination and using this load case in design. The reason is that if program internally multiplies the analysis moment by 1.7 for ASD method (it is 1.2 for dead and 1.6 for live loads for LRFD method) then you must ensure that the analysis moment is the unfactored moment. If by mistake the 1.7 factor is used during load combination and the I-beam with web opening is designed with this load, the program will further increase the load by 1.7. Hence, it has been intentionally left to your engineering judgement to use whatever load factor you see fit before designing I-beam with opening with that factored load case.

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

2.4.3.2 Design Parameters

The following table contains a list of parameters and their default values.

Table 2-5: American Castelled Beam Design Parameters

Parameter Name	Default Value	Description
<u>CB</u>	1.0	Cb value used for computing allowable bending stress per Chapter F of the AISC specifications.
<u>CMZ</u>	0.85	Cm value in local z-axis. Used in the interaction equations in Chapter H of the AISC specifications.
<u>EOPEN</u>	$1.5e + b$	Distance from the ecenter of the last hole to the end of the member. $1.5e + b$ is the minimum allowable value. Any value greater than or equal to this minimum will be used by the program. See Figure 2.13 for the definition of e and b.
<u>FYLD</u>	36 ksi	Yield stress of steel.
<u>RATIO</u>	1.0	Permissible maximum ratio of actual load to section capacity. Any input value will be used to change the right hand side of the governing interaction equations in Chapter H and elsewhere in the AISC specifications.

Parameter Name	Default Value	Description
<u>SOPEN</u>	$1.5e + b$	Distance from the start of the member to the center of the first hole. $1.5e + b$ is the minimum allowable value. Any value greater than or equal to this minimum will be used by the program. See Figure 2.13 for the definition of e and b .
<u>TRACK</u>	0	Used to control the level of description of design output. 0 = Detailed output suppressed 1 = Detail output included
<u>UNL</u>	Member Length	Unsupported length of compression flange for calculating allowable bending stress.

2.4.3.3 Design Procedure

Cross-Section Checks

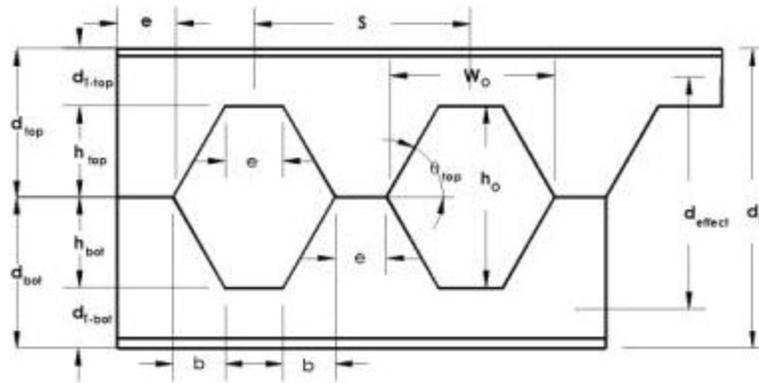
The first check that is carried out is a verification whether the member properties satisfy certain basic requirements. If the member fails these checks, the remainder of the checks are not performed.

The cross section checks are the following:

Figure 2-9: Castellated beam nomenclature

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition



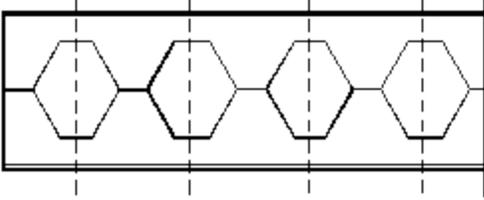
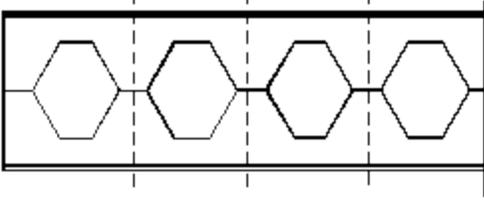
1. Web Post Width (e) should be at least 3.0 inches
2. Tee Depth (d_{T-top} and d_{T-bot}) should be greater than the thickness of flange plus one inch.
3. Angle θ should be between 45 and 70 degrees.
4. In order for the program to determine the number of holes which are admissible for the beam, the parameters SOPEN and EOPEN need to be assigned. In the figure above, there is a term shown as S . This value is part of the section tables supplied with STAAD.Pro, so it retrieves that value from there. It then computes the number of holes, and the remainder of the terms shown in the above diagram.
5. SOPEN and EOPEN (see the parameter table shown earlier) have to be at least $1.5e + b$, with “ e ” and “ b ” as shown in the earlier figure. If you inputs a value less than these minima, the minimum values are used.

Checking the member for adequacy in carrying the applied loading

This consists of five different checks:

1. Global Bending
2. Vierendeel Bending
3. Horizontal Shear
4. Vertical Shear
5. Web Post Buckling

Table 2-6: Cross section considered for limit states

Design For	Section Considered in the Design (shown with the vertical dotted lines)
Vier-endeel Bending	
Global Bending Vertical Shear Horizontal Shear Web Post Buckling	

1. Global Bending:

Global bending check is done at the web post section. This is the region of the member where the full cross section is active, without interference of the holes.

The actual bending stress is computed at the middle of the web post location and is obtained by dividing the moment by the section modulus of the full section.

For computing the allowable bending stress, the compactness of the section is first determined in accordance with Table B5.1 in the Chapter B of the AISC 9th edition specifications. The rules applicable to I-shaped sections are

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

used for this. Following this, the allowable bending stress is computed per chapter F of the same.

The ratio is computed by dividing the actual stress by the allowable stress.

2. Vierendeel Bending:

This is checked at the middle of the hole locations. The effective cross section at these locations is a Tee. The overall moment (M_z) at the span point corresponding to the middle of the hole is converted to an axial force and a moment on the Tee.

The actual stress is computed at the top and bottom of each Tee section.

$$f_a = M / (d_{\text{effect}} * A_t)$$

where A_t is the area of the Tee section

$$f_b = V * e * a / (2 S)$$

where a is the area factor. For the top Tee section, $a = \text{Area of Top Tee} / (\text{Area of Top Tee} + \text{Area of Bottom Tee})$

Allowable Stresses for vierendeel bending:

- Axial Stress: The allowable axial stress is computed as per the Chapter E of the AISC specifications. The unsupported length for column buckling is equal to e .
- Bending Stress: The allowable bending stress is computed for the top and bottom Tee section as per the Chapter F of the AISC manual.

The axial stress plus bending stress is computed at the top and bottom of each tee section. If it is compressive then it is checked against equations H1-1 and H1-2 of Chapter H of the AISC manual. If it is tensile then it is checked against equation H2-1

3. Horizontal Shear

Allowable Shear stress is computed as $0.4 F_y$.

Actual Stress: Please refer to pages 4.7-8 and 4.7-9 of the reference book on welded structures mentioned under Item (a) earlier.

4. Vertical Shear

Allowable Shear stress is computed as $0.4 F_y$.

The actual shear stress is computed at the middle of the web post location.

5. Web Post buckling

Please refer to pages 1202-1207 of the ASCE journal mentioned under Item (c) earlier.

2.4.3.4 General Format

The command syntax in the STAAD input file for assigning castellated beams is:

```
MEMBER PROPERTY AMERICAN
Member-list TABLE ST section-name
```

Example

```
MEMBER PROPERTY AMERICAN
2 TABLE ST CB12X28
```

Assigning Design parameters

Under the **PARAMETERS** block on input, the code name must be specified as:

```
CODE AISC CASTELLATED
```

Example

```
PARAMETER
CODE AISC CASTELLATED
UNL 0.01 MEMB 25 31
FYLD 50 MEMB 25 31
SOPEN 11.124 MEMB 25 31
...
CHECK CODE MEMB 25 31
```

2.4.3.5 Steel Design Output

The following is a typical **TRACK 2** level output page from a STAAD output file.

```
STAAD.PRO CODE CHECKING - (AISC CASTELLATED) v1.0
*****
ALL UNITS ARE - Kip and Inches (UNLESS OTHERWISE NOTED)
```

```
Castellated Steel Design for Member      2
```

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

```
=====
Section Name  ST  CB27X40

Design Results
-----
Design Status: Pass   Critical Ratio: 0.96

Check for Global Bending
-----
Load =      3          Section =   260.874
Fy =      0.76          Mz =  -3020.39
Fb top =    33.00      Fb Bot =    33.00
fb =      26.83
Ratio =  0.81

Check for Vierendeel Bending
-----
Load =      3          Section =   214.624
Fy =      4.61          Mz =  -2894.76
Fa =     29.91          Fb =    30.00
Klr =      1.46          Fe =  69606.88
fa =     26.08          fb =     2.79
Ratio =  0.96

Check for Vertical Shear
-----
Load =      3          Section =     0.000
Fy =     22.50          Mz =     0.00
Fv =     20.00          fv =     2.62
Ratio =  0.13

Check For Horizontal Shear ( Web Post )
-----
Load =      3          Section =   519.874
Fy =    -20.82          Mz =  -415.10
Fv =     20.00          fv =    14.73
Ratio =  0.74

Check for Web Post Buckling
-----
Load =      3          Section =   519.874
Fy =    -20.82          Mz =  -415.10
Mallow =   141.32      Mact =   189.47
Ratio =  0.75
```

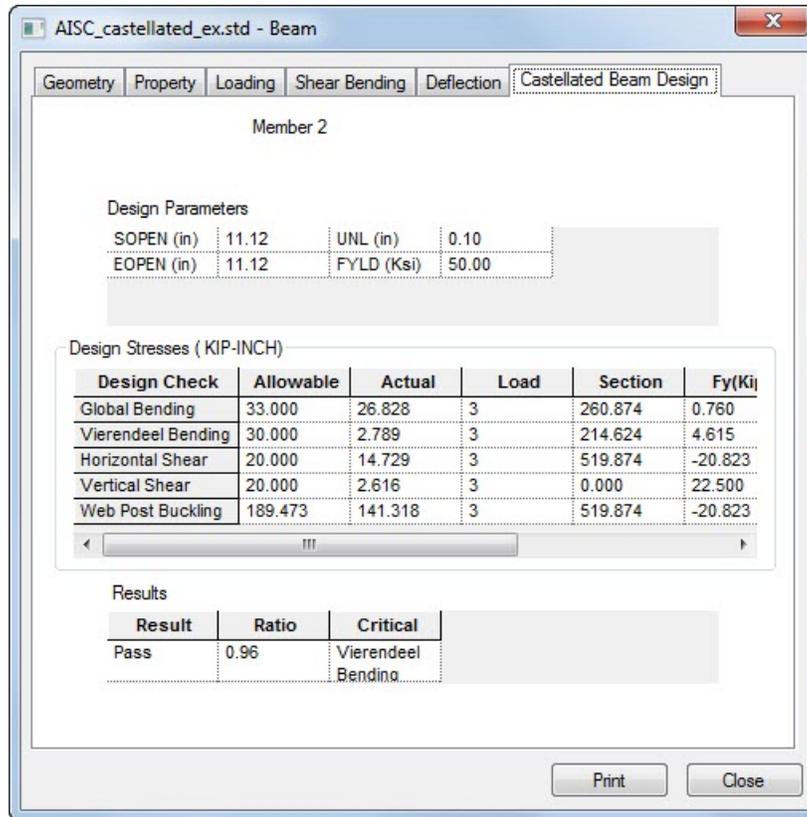
Viewing the design results in the GUI

1. After the analysis and design is completed, double click on the castellated member in the STAAD.Pro View window.

The Beam dialog box.

2. Select the **Castellated Beam Design** tab.

Figure 2-10: Beam dialog box Castellated Beam Design tab



3. (Optional) Click **Print** to create a hard copy of the Castellated Beam data for this beam.
4. Click **Close**.
The dialog closes.

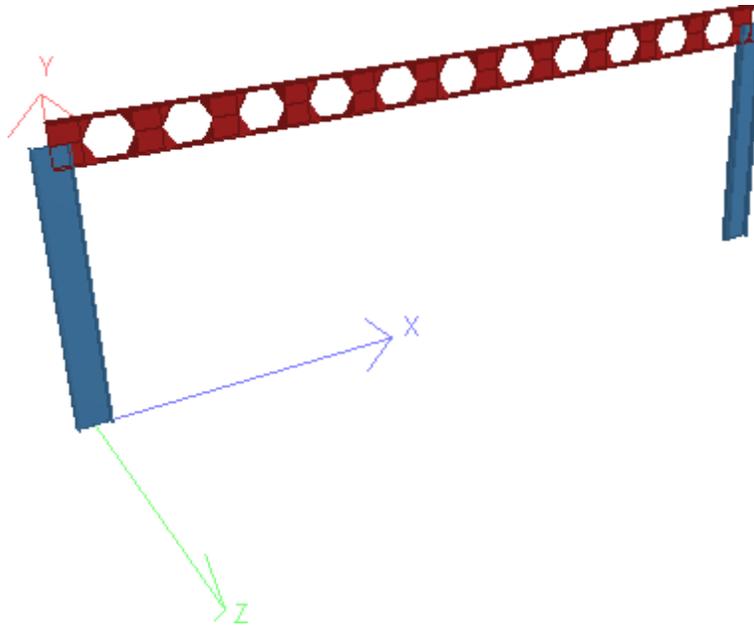
2.4.3.6 Example

The following is an example STAAD Input for a portal frame with a castellated beam.

Figure 2-11: 3D Rendering of the example structure

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition



Hint: You can copy the input code below and paste into the STAAD Editor or into a plain text editor program and save as an .STD file for use in STAAD.Pro.

STAAD PLANE EXAMPLE PROBLEM FOR

*CASTELLATED BEAM DESIGN

UNIT FT KIP

JOINT COORDINATES

1 0. 0. ; 2 45 0

3 0 15 ; 4 45 15

MEMBER INCIDENCE

1 1 3; 2 3 4; 3 4 2

MEMBER PROPERTY AMERICAN

2 TA ST CB27X40

1 3 TA ST W21X50

UNIT INCH

CONSTANTS

E STEEL ALL

DEN STEEL ALL

POISSON STEEL ALL

```
MEMBER RELEASE
2 START MX MY MZ
2 END MY MZ
UNIT FT
SUPPORT
1 2 FIXED
LOADING 1 DEAD AND LIVE LOAD
MEMB LOAD
2 UNI Y -0.4
LOADING 2 WIND FROM LEFT
MEMBER LOAD
2 UNI Y -0.6
LOAD COMB 3
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PRINT MEMBER FORCES
PRINT SUPPORT REACTION
UNIT KIP INCH
PARAMETER
CODE AISC CASTELLATED
UNL 0.01 MEMB 2
FYLD 50 MEMB 2
CMZ 0.85 MEMB 2
CB 1.1 MEMB 2
TRACK 2.0 ALL
SOPEN 11.124 MEMB 2
EOPEN 11.124 MEMB 2
CHECK CODE MEMB 2
FINISH
```

2.4.4 Design of Beams with Web Openings

Design of steel members with web openings per AISC Steel Design Guide 2 - ASD specifications may be performed in STAAD for members whose yield strength is 65 ksi or less.

Note: The web openings are given consideration only during the design phase. The reduction in section properties caused by the presence of the openings is not considered automatically during the analysis phase. Hence, the analysis is performed as if the full section properties are effective for such members.

During the design process, the program first determines the utilization ratio (U.R.) at the location of the opening as though it is an unreinforced opening. If the U.R. is less than 1.0, the member is presumed to have passed the requirements at that location. If the U.R. exceeds 1.0, then it determines the U.R. as though it is a reinforced opening. If it fails this too, the cause of the failure along with the associated numerical values is reported.

2.4.4.1 Description

The following are the salient points of the design process.

- A. Only a code check operation is permitted on members with web openings. The **MEMBER SELECTION** process will not be performed if a web opening is specified for the member.
- B. The **CODE CHECK** operation is performed at the following locations along the member span:
 - a. The 13 equally spaced points along the member span customary with the **BEAM 1** parameter
OR
The section locations specified using the **SECTION** command, if the **BEAM** parameter is set to 0.0
OR
The 2 member ends if **BEAM** parameter is set to 0, and the **SECTION** command is not specified.
 - b. At the web openings locations defined using the **RHOLE** and **CHOLE** parameters (Refer to Table 2.18-1 below).

If any of the locations defined under (a) above happen to coincide with those in (b), such locations are designed as places where openings are located, and not as an unperforated section location.

The utilization ratio (U.R.) is determined for all the locations in (a) above, as well as all the locations in (b) above. The highest value among these locations is deemed critical from the design standpoint.

The design output consists of the critical value obtained from checking the locations under (a), and each of the locations under (b).

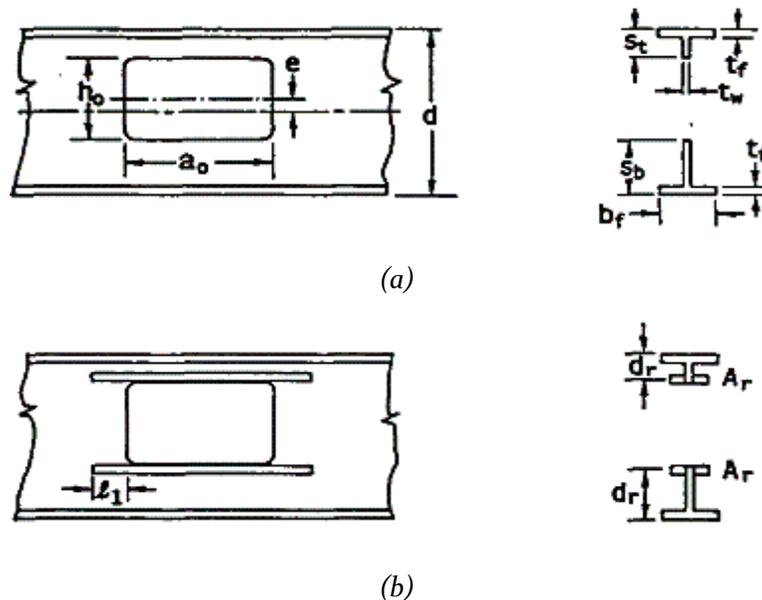
The critical location among those in (b) is not displayed in the post-processing pages of the program such as the Beam-Unit Check page, or the Member Query-Steel Design tab.

Members declared as **TRUSS** (trusses) or **TENSION** (tension-only) are not designed for web openings.

2.4.4.2 Design steps for Steel Beam with Web Opening

At the location of web holes, the capacity of the section is determined using the rules explained below.

Figure 2-12: Opening configurations for steel beams, (a) unreinforced opening, (b) reinforced opening



Section Properties Required:

A_s = Cross-sectional area of steel in the unperforated member (at a

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

section along the beam where there is no opening)

d = Depth of beam

t_w = Thickness of web

b_f = Width of flange

t_f = Thickness of flange

Z = Plastic section modulus of member without opening

J = Torsional constant of the beam

L = Length of the member

L_b = Unbraced length of compression flange

Opening Information Required:

e = Eccentricity of opening, specified using the **HECC** parameter
(distanced from mid-depth of beam to mid-depth of the opening) = $|e|$

sign convention for eccentricity: upward eccentricity + e , downward eccentricity - e

Loading:

V_u = Factored shear at different opening locations

M_u = Factored bending moment at different opening locations

Calculated Parameters

Circular Opening Properties:

Unreinforced Opening Properties:

$h_o = D_o$ for bending

$h_o = 0.9 D_o$ for shear

$a_o = 0.45 D_o$

Reinforced Opening Properties:

$h_o = D_o$ for bending and shear

$a_o = 0.45 D_o$

Tee Properties:

Refer to Figure 2.17(a), above

$$s_t = \text{Depth of top tee} = d/2 - (h_o / 2 + e)$$

$$s_b = \text{Depth of bottom tee} = d/2 - (h_o / 2 - e)$$

Reinforcement Properties:

A_r = Area of reinforcement on each side of the opening

t_r = Thickness of reinforcement bar

b_r = Width of reinforcement bar

D_r = Depth of reinforcement bar

2.4.4.3 Calculation Steps

1. Check for local buckling of compression flange and reinforcement (if any)

(AISC Design Guide 2 for Web Openings: section 3.7.a.1)

Width to thickness ratio of compression flange, $F_1 = b_f / 2t_f$

Width to thickness ratio of web reinforcement, $F_2 = b_r / t_r$

Limiting width to thickness ratio, $B_1 = 65 / \sqrt{F_y}$

F_1 and F_2 must not exceed B_1 .

2. Check for web buckling

(AISC Design Guide 2 for Web Openings: section 3.7.a.2)

Width to thickness ratio of web, $W_1 = (d - 2t_f) / t_w$

W_1 must not exceed $520 / \sqrt{F_y}$

3. Check for opening dimensions to prevent web buckling

(AISC Design Guide 2 for Web Openings: section 3.7.a.2 & section 3.7.b.1)

- a. Limit on a_o / h_o as given below,

If W_1 is $\leq 420 / \sqrt{F_y}$ then web qualifies as stocky,

a_o / h_o must not exceed 3.0,

If W_1 is $> 420 / \sqrt{F_y}$ but $\leq 520 / \sqrt{F_y}$ then

a_o / h_o must not exceed 2.2,

- b. h_o / d must not exceed 0.7

- c. The opening parameter, $p_o = (a_o / h_o) + (6 h_o / d)$ must not exceed 5.6

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

4. Check for tee dimensions

(AISC Design Guide 2 for Web Openings: section 3.7.b.1)

- a. The maximum depth of the web opening is governed by the following rules:

$$\text{Depth } s_t \geq 0.15d, s_b \geq 0.15d$$

- b. Aspect ratios of the tees ($v = a_o / s$) should not be greater than 12

$$a_o / s_b \leq 12, a_o / s_t \leq 12$$

5. Check for buckling of tee-shaped compression zone

(AISC Design Guide 2 for Web Openings: section 3.7.a.3)

The tee, which is in compression, is investigated as an axially loaded column. For unreinforced members this is not done when the aspect ratio of the tee is less than or equal to four:

$$F_y t_w d / \sqrt{3} \leq 4$$

For reinforced openings, this check is only done for large openings in regions of high moment.

6. Calculation for Maximum Moment Capacity, M_m

(AISC Design Guide 2 for Web Openings: section 3.5.a)

For unperforated section

$$M_p = F_y Z$$

$$\Delta A_s = h_o t_w - 2A_r$$

Unreinforced Opening:

$$M_m = M_p \left[1 - \frac{\Delta A_s \left(\frac{h_o}{4} + e \right)}{Z} \right]$$

Reinforced Opening:

- a. If $t_w e < A_r$

$$M_m = M_p \left[1 - \frac{t_w \left(\frac{h_o^2}{4} + h_o e - e^2 \right) - A_r h_o}{Z} \right] \leq M_p$$

b. If $t_w e \geq A_r$

$$M_m = M_p \left[1 - \frac{\Delta A_s \left(\frac{h_o}{4} + e - \frac{A_r}{2t_w} \right)}{Z} \right] \leq M_p$$

7. Calculation for Maximum Shear Capacity, V_m

(AISC Design Guide 2 for Web Openings: section 3.6.a)

V_{pb} and V_{pt} = Plastic shear capacity of the web of the tee

$$V_{pb} = F_y t_w s_b / \sqrt{3}$$

$$V_{pt} = F_y t_w s_t / \sqrt{3}$$

The values of aspect ratios v_b and v_t and factors μ_b and μ_t (which appear in the equations shown below) are different for reinforced and unreinforced openings.

Unreinforced Opening:

For the bottom tee, $v_b = a_o / s_b$ and $\mu_b = 0$

For the top tee, $v_t = a_o / s_t$ and $\mu_t = 0$

Reinforced Opening:

s_{t1} and s_{b1} are used to calculate for n reinforced opening.

$$s_{t1} = s_t - A_r / (2b_f)$$

$$s_{b1} = s_b - A_r / (2b_f)$$

P_r = Force in reinforcement along edge of opening = $F_y A_r \leq F_y t_w a_o / (2\sqrt{3})$

d_{rt} and d_{rb} = Distance from outside edge of flange to centroid of reinforcement.

$$d_{rt} = s_t - \frac{1}{2} t_r$$

$$d_{rb} = s_b - \frac{1}{2} t_r$$

For the bottom tee, $v_b = a_o / s_{b1}$ and $\mu_b = 2P_r d_{rb} / (V_{pb} s_b)$

For the top tee, $v_t = a_o / s_{t1}$ and $\mu_t = 2P_r d_{rt} / (V_{pt} s_t)$

General Equations:

Using equations given below for α_{vb} and α_{vt}

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

α_{vb} = Ratio of nominal shear capacity of bottom tee, V_{mb} to plastic shear capacity of the web of the tee = $(\sqrt{6 + \mu_b}) / (v_b + \sqrt{3}) \leq 1$

α_{vt} = Ratio of nominal shear capacity of top tee, V_{mt} to plastic shear capacity of the web of the tee = $(\sqrt{6 + \mu_t}) / (v_t + \sqrt{3}) \leq 1$

$$V_{mb} = V_{pb} \alpha_{vb}$$

$$V_{mt} = V_{pt} \alpha_{vt}$$

V_m = Maximum nominal shear capacity at a web opening = $V_{mb} + V_{mt}$

8. Check against Maximum Shear Capacity V_m

(AISC Design Guide 2 for Web Openings: section 3.7.a.2)

V_p = Plastic shear capacity of unperforated web = $F_y t_w d / \sqrt{3}$

If $W_1 \leq 420/\sqrt{F_y}$, V_m must not exceed $2/3 V_p$

If $W_1 > 420/\sqrt{F_y}$ but $\leq 520/\sqrt{F_y}$, V_m must not exceed $0.45 V_p$

9. Check against Moment Shear Interaction

AISC Design Guide 2 for Web Openings: section 3.2 & section 3.4)

$$R1 = V_u / V_m \leq 1.0$$

$$R2 = M_u / M_m \leq 1.0$$

$$R1^3 + R2^3 \leq R^3, R \leq 1.0$$

10. Corner Radii (for reinforced opening only)

(AISC Design Guide 2 for Web Openings: section 3.7.b.2)

Minimum radii = the greater of $2t_w$ or 5/8 inch

11. Calculation of length of fillet weld (for reinforced opening only)

(AISC Design Guide 2 for Web Openings: section 3.7.b.5)

For reinforcing bars on both sides / on one side of the web:

Fillet welds should be used on both sides of the reinforcement on extensions past the opening. The required strength of the weld within the length of the opening is,

$$R_{wr} = 2P_r$$

Where:

R_{wr} = Required strength of the weld

The reinforcement should be extended beyond the opening by a distance

$$L_1 = a_o / 4 \text{ or } L_1 = A_r \sqrt{3} / (2t_w)$$

whichever is greater, on each side of the opening. Within each extension, the required strength of the weld is

$$R_{wr} = F_y A_r$$

Additional requirements for reinforcing bars on one side of the web:

$$A_f = \text{area of flange} = b_f \cdot t_f$$

- a. $A_r \leq A_f / 3$
- b. $a_o / h_o \leq 2.5$
- c. $V_1 = s_t / t_w$ or $V_2 = s_b / t_w$
 V_1 and $V_2 \leq 140 / \sqrt{F_y}$
- d. $M_u / (V_u d) \leq 20$

12. Calculation for spacing of openings

(AISC Design Guide 2 for Web Openings: section 3.7.b.6)

Rectangular Opening:

$$S \geq h_o$$

$$S \geq a_o (V_u / V_p) / [1 - (V_u / V_p)]$$

Circular Opening:

$$S \geq 1.5 D_o$$

$$S \geq D_o (V_u / V_p) / [1 - (V_u / V_p)]$$

13. Check for deflection

The deflection check is performed using the approximate procedure described in section 6.2 of the AISC Design Guide 2 for Web Openings.

2.4.4.4 General Format

RHOLE r1 r2 r3 Memb <list>

CHOLE c1 c2 c3 Memb <list>

Where:

r1, r2 and r3 and c1, c2 and c3 are the section locations of three

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

rectangular and three circular openings respectively, along the length of the member in ascending order from the start of the member (i.e., $r_1 < r_2 < r_3$ and $c_1 < c_2 < c_3$)

Notes

The maximum number of openings allowed for each member is three. Thus there can be three rectangular openings, three circular openings, or a combination of three rectangular and circular openings for each member.

RDIM [l1 d1] [l2 d2] [l3 d3] Memb <list>

Where l_1 , l_2 and l_3 are the three different lengths and d_1 , d_2 and d_3 are the three different depths of the rectangular openings.

CDIA d1 d2 d3 Memb <list>

Where d_1 , d_2 and d_3 are the three different diameters of the circular openings.

HECC e1 e2 e3 Memb <list>

If the eccentricity of the opening is in the negative local Y-axis of the member the sign should be negative.

ELECTRODE f Memb <list>

Where f is the weld material used to calculate size and length of fillet weld required to connect reinforcing bars on beam web at opening.

Example

```
UNIT INCH PARAMETER RHOLE 0.4 0.6 MEMB 5
RDIM 10.0 5.0 20.0 10.0 MEMB 5 CHOLE 0.8 MEMB 5 CDIA 10.0
MEMB 5 ELECTRODE 3 MEMB 5
```

The above example shows that member 5 contains two rectangular openings at sections 0.4 and 0.6 whereas one circular opening is located at section 0.8 of the member. The dimensions of rectangular openings are 10.0 X 5.0 and 20.0 X 10.0 inch respectively whereas diameter of circular opening is 10.0 inch.

Note: Refer to "AD.2005.3.1 Designing I-beams with web openings per AISC ASD" in the STAAD.Pro 2005 Release Report for verification examples using this feature.

2.4.4.5 Example

Note: Refer to "AD.2005.3.1 Designing I-beams with web openings per AISC ASD" in the STAAD.Pro 2005 Release Report for verification examples using this feature.

```

STAAD PLANE EXAMPLE PROBLEM NO. 1
START JOB INFORMATION
ENGINEER DATE 18-MAY-05
END JOB INFORMATION
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0; 3 0 20 0; 4 10 20 0; 5 20 20 0; 6 30
20 0; 7 0 35 0;
8 30 35 0; 9 7.5 35 0; 10 22.5 35 0; 11 15 35 0; 12 5 38
0; 13 25 38 0;
14 10 41 0; 15 20 41 0; 16 15 44 0;
MEMBER INCIDENCES
1 1 3; 2 3 7; 3 2 6; 4 6 8; 5 3 4; 6 4 5; 7 5 6; 8 7 12;
9 12 14;
10 14 16; 11 15 16; 12 13 15; 13 8 13; 14 9 12; 15 9 14;
16 11 14;
17 11 15; 18 10 15; 19 10 13; 20 7 9; 21 9 11; 22 10 11;
23 8 10;
MEMBER PROPERTY AMERICAN
1 3 4 TABLE ST W14X90
2 TABLE ST W10X49
5 6 7 TABLE ST W21X50
8 TO 13 TABLE ST W18X35
14 TO 23 TABLE ST L40404
MEMBER TRUSS
14 TO 23
MEMBER RELEASE

```

Section 2 American Steel Design

2.4 Steel Design per AISC 9th Edition

```
5 START MZ
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
DENSITY 0.000283
ISOTROPIC STEEL
E 29732.7
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2E-005
DAMP 0.03
END DEFINE MATERIAL
CONSTANTS
BETA 90 MEMB 3 4
MATERIAL MATERIAL1 MEMB 1 TO 4 6 TO 23
MATERIAL STEEL MEMB 5
UNIT FEET KIP
SUPPORTS
1 FIXED
2 PINNED
PRINT MEMBER INFORMATION LIST 1 5 14
PRINT MEMBER PROPERTIES LIST 1 2 5 8 14
LOAD 1 DEAD AND LIVE LOAD
SELFWEIGHT X 1
SELFWEIGHT Y -1
JOINT LOAD
4 5 FY -15
11 FY -35
MEMBER LOAD
```

```

8 TO 13 UNI Y -0.9
6 UNI GY -1.2
CALCULATE RAYLEIGH FREQUENCY
LOAD 2 WIND FROM LEFT
MEMBER LOAD
1 2 UNI GX 0.6
8 TO 10 UNI Y -1
* 1/3 RD INCREASE IS ACCOMPLISHED BY 75% LOAD
LOAD COMB 3 75 PERCENT DL LL WL
1 0.75 2 0.75
LOAD COMB 4 75 PERCENT DL LL WL
1 2.75 2 2.75
PERFORM ANALYSIS
LOAD LIST 4
UNIT INCHES KIP
PARAMETER
CODE AISC
*WEB OPENINGS
*****
RHOLE 0.6 MEMB 5
RDIM 20.0 10.0 MRMB 5
ELECTRODE 3
*****
CHECK CODE MEMB 5 6
FINISH

```

2.5 Steel Design per AASHTO Specifications

Design to AASHTO Standard Specifications for Highway Bridges utilizing the ASD and LRFD approaches is implemented in STAAD. These are described in the following two sections.

To utilize the ASD method, specify the commands

PARAMETER

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

CODE AASHTO

Or

PARAMETER

CODE AASHTO ASD

To utilize the LRFD method, specify the commands

PARAMETER

CODE AASHTO LRFD

2.5.1 AASHTO (ASD)

The design of structural steel members in accordance with the AASHTO Standard Specifications for Highway Bridges, 17th edition has been implemented.

2.5.1.1 General

The section of the above code implemented in STAAD is Chapter 10, Part C – Service Load design Method, Allowable Stress design. Sections 10.32.1.A and 10.36 are implemented. As per the AASHTO committee, this is the last edition for this code (the ASD approach) and only technical errors will be fixed in the future for this code.

In general, the concepts followed in **MEMBER SELECTION** and **CODE CHECKING** procedures are similar to that of the AISC based design. It is assumed that the user is familiar with the basic concepts of steel design facilities available in STAAD. Please refer to Section 2 of the STAAD Technical Reference Manual for detailed information on this topic. This section specifically addresses the implementation of steel design based on the AASHTO specifications.

Design is available for all standard sections listed in the AISC ASD 9th edition manual, namely, Wide Flanges, S, M, HP, Tees, Channels, Single Angles, Double Angles, Tubes and Pipes. The design of HSS sections (those listed in the 3rd edition AISC LRFD manual) and Composite beams (I shapes with concrete slab on top) are not supported.

2.5.1.2 Allowable Stresses

The member design and code checking in STAAD is based upon the allowable stress design method. It is a method for proportioning structural members using design loads and forces, allowable stresses, and design limitations for the appropriate material under service conditions. It is beyond the scope of this manual to describe every aspect of structural steel design per AASHTO

specifications because of practical reasons. This section will discuss the salient features of the allowable stresses specified by the AASHTO code. Table 10.32.1A of the AASHTO code specifies the allowable stresses.

Axial Stress

Allowable tension stress, as calculated in AASHTO is based on the net section. This tends to produce a slightly conservative result. Allowable tension stress on the net section is given by,

$$F_t = 0.55 \cdot F_y$$

Allowable compressive stress on the gross section of axially loaded compression members is calculated based on the following formula:

$$F_a = F_y / FS \cdot [1 - (Kl/r)^2 F_y / (4\pi^2 E)]$$

when $(Kl/r) \leq C_c$

$$F_a = \pi^2 E / [FS \cdot (Kl/r)^2]$$

when $(Kl/r) > C_c$

Where:

$$C_c = (2\pi^2 E / F_y)^{1/2}$$

It should be noted that AASHTO does not have a provision for increase in allowable stresses for a secondary member and when $1/r$ exceeds a certain value.

Bending Stress

Allowable stress in bending compression for rolled shape girders and built-up sections whose compression flanges are supported laterally through their full length by embedment in concrete is given by:

$$F_b = 0.55 \cdot F_y$$

For similar members with unsupported or partially supported flange lengths, the allowable bending compressive stress is given by

$$F_b = 0.55 \cdot F_y [1 - (1/r)^2 F_y / (4\pi^2 E)]$$

Where:

$$r^2 = b^2 / 12$$

Due to inadequate information in the AASHTO Code, the allowable tensile stresses due to bending for both axes are set to be the same as the corresponding allowable bending compressive stresses.

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

Shear Stress

Allowable shear stress on the gross section is given by:

$$F_v = 0.33 F_y$$

For shear on the web, the gross section is defined as the product of the total depth and the web thickness. The AASHTO code does not specify any allowable stress for shear on flanges. The program assumes the same allowable for shear stress ($0.33F_y$) for both shear on the web and shear on the flanges. For shear on the flanges, the gross section is taken as $2/3$ times the total flange area.

Bending-Axial Stress Interaction

Members subjected to both axial and bending stresses are proportioned according to section 10.36 of the AASHTO steel code. All members subject to bending and axial compression are required to satisfy the following formula:

$$f_a/F_a + C_{mx} \cdot f_{bx}/[(1 - f_a/F_{ex}) \cdot F_{bx}] + C_{my} \cdot f_{by}/[(1 - f_a/F_{ey}) \cdot F_{by}] < 1.0$$

at intermediate points, and

$$f_a/(0.472 \cdot F_y) + f_{bx}/F_{bx} + f_{by}/F_{by} < 1.0$$

at the ends of the member.

The start and end nodes of a member are treated as support points.

For members subject to axial tension and bending, the following equations are checked:

$$f_a/F_a + f_{bx}/F_{bx} + f_{by}/F_{by} < 1.0$$

at intermediate points, and

$$f_a/(0.472 \cdot F_y) + f_{bx}/F_{bx} + f_{by}/F_{by} < 1.0$$

at the ends of the member.

2.5.1.3 AASHTO (ASD) Design Parameters

The following table outlines the parameters that can be used with the AASHTO (ASD) code along with the default values used if not explicitly specified.

Table 2-7: AASHTO (ASD) Design Parameters

Parameter Name	Default Value	Description
<u>BEAM</u>	1.0	0.0 = Design at ends and those locations specified by the SECTION command. 1.0 = Design at ends and every 1/12 th point along the member length.
<u>CB</u>	1.0	Cb value as used in the calculation of Fb 0.0 = Cb value to be calculated Any other value will be used in the calculations.
<u>CMY</u> <u>CMZ</u>	0.85 for sidesway and calculated for no sidesway	Cm value in local y & z axes
<u>DFE</u>	None. (Mandatory for a deflection check)	“Deflection length” / Maximum allowable local axis deflection.
<u>DJ1</u>	Start joint of member	Joint No. denoting starting point for calculating “Deflection Length”.
<u>DJ2</u>	End joint of member	Joint No. denoting ending point for calculating “Deflection Length”.

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

Parameter Name	Default Value	Description
<u>D</u> MAX	1000.0	Maximum allowed section depth (in current length units) for a section to be selected with the SELECT command.
<u>D</u> MIN	0.0	Minimum allowed section depth (in current length units) for a section to be selected with the SELECT command.
<u>F</u> YLD	36 KSI	Yield strength of steel in current units.
<u>K</u> Y	1.0	K value in local y-axis. Usually, this is minor axis.
<u>K</u> Z	1.0	K value in local z-axis. Usually, this is major axis.
<u>L</u> Y	Member Length	Length to calculate slenderness ratio for buckling about local Y axis.
<u>L</u> Z	Member Length	Same as above except in local z-axis.
<u>M</u> AIN	0.0	0.0 =check for slenderness 1.0 =suppress slenderness check

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

Parameter Name	Default Value	Description
<u>NSF</u>	1.0	Ratio of 'Net cross section area' / 'Gross section area' for tension member design.
<u>PROFILE</u>	None	Used in member selection. Refer to Section 5.48.1 for details.
<u>PUNCH</u>		1.0 = K-Overlap 2.0 = K-Gap 3.0 = T and Y 4.0 = Cross with diaphragms 5.0 = Cross without diaphragms
<u>RATIO</u>	1.0	Permissible ratio of the actual to allowable stresses.
<u>SSY</u>	0.0	0.0 = Sidesway in local y-axis. 1.0 = No sidesway in local y-axis
<u>SSZ</u>	0.0	0.0 = Sidesway in local z-axis. 1.0 = No sidesway in local z-axis.
<u>STIFF</u>	Greater of member length or depth of beam.	Spacing of stiffeners for plate girder design in current length units.

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

Parameter Name	Default Value	Description
<u>TRACK</u>	0	Level of detail in Output File: 0 = Print the design output at the minimum detail level. 1 = Print the design output at the intermediate detail level. 2 = Print the design output at maximum detail level.
<u>UNF</u>	1.0	Unsupported length factor of the compression flange for calculating the allowable bending compressive strength.
<u>UNL</u>	Member Length	Unsupported length of compression flange for calculating allowable bending compressive stress.
<u>WSTR</u>	0.4 x FYLD	Allowable welding stress

2.5.1.4 AASHTO (ASD) Verification Problem

The following compares the solution of a design performed using STAAD.Pro against a hand calculation. This problem is included as

.../SPROV8I/STAAD/EXAMP/US/AASHTO_VER.STD with the program.

Type

Allowable Stress Steel Design per the AASHTO Standard Specifications for Highway Bridges, 17th Edition (2002)

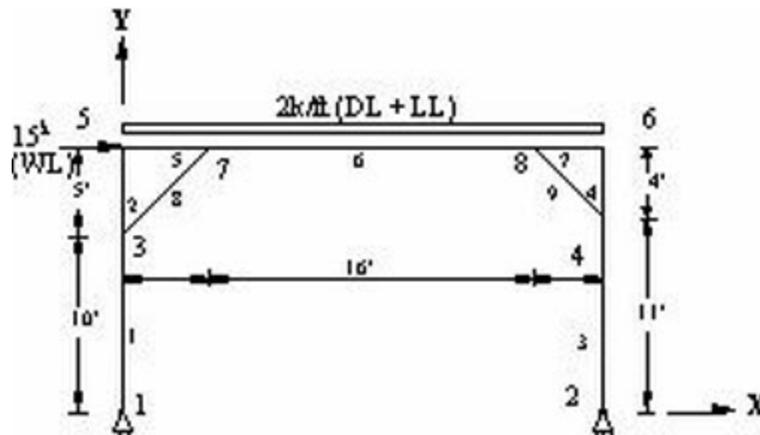
Reference

Following step by step hand calculation as per AASHTO code.

Problem

Determine the allowable stresses (AASHTO code) for the members of the structure as shown in figure. Also, perform a code check for these members based on the results of the analysis.

Figure 2-13: AASHTO ASD verification problem



Members 1, 2 = W12X26, Members 3, 4 = W14X43

Members 5, 6, 7 = W16X36, Memb8= L40404,

Member 9 = L50506

The frame is subject to the following load cases:

1. a uniform, gravity load along the beam of 2 kips/ft
2. a lateral, wind load of 15 kips

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

- a combination of 75% of load 1 and 75% of load 2

Comparison

Table 2-8: Comparison of governing ratios for members in the AASHTO ASD verification example

Member Number	STAAD.Pro Results	Hand Calculation	Difference
1	1.218	1.217	none
2	1.093	1.092	none
3	1.204	1.203	none
4	1.126	1.126	none
5	0.690	0.689	none
6	1.052 (0.832 when slenderness check is suppressed)	0.732	none*
7	0.809	0.808	none
8	1.091	1.091	none
9	0.928	0.927	none

Solution

Only the AASHTO steel design elements are checked here. No structural analysis calculations are included in these hand verifications.

Though the program does check shear per the AASHTO specifications, those calculations are not reflected here. Only the controlling stress ratios are presented.

As all members are grade 36 steel, the following critical slenderness parameter applies to each:

$$C_c = \sqrt{\frac{2\pi^2 E}{F_y}} = \sqrt{\frac{2\pi^2 29,000}{36}} = 126.1$$

1. Member 1

Size W 12X26, L = 10 ft., a = 7.65 in², S_z = 33.39 in³

From observation, Load case 1 will govern

- F_x = 25.0 kip (compression)
- M_z = 56.5 k-ft

Calculate the allowable stress as per Table 10.32.1A.

Bending Minor Axis

Allowable minor axis bending stress:

$$F_{TY} = F_{TZ} = 0.55 \cdot F_Y = 19.8 \text{ ksi}$$

Bending Major Axis

$$F_{CZ} = \frac{50(10)^6 C_b \left(\frac{I_{yc}}{l} \right)}{S_{xc}} \sqrt{0.772 \frac{J}{I_{yc}} + 9.87 \left(\frac{d}{l} \right)^2} \leq 0.55 F_y$$

Where:

$$C_b = 1.75 + 1.05(M1/M2) + 0.3x(M1/M2)^2$$

$$M1 = 0, \text{ so } C_b = 1.75$$

$$S_{zc} = \text{Section modulus with respect to the compression flange} \\ = 204 / (0.5 \cdot 12.22) = 33.38789 \text{ in}^3$$

$$I_{YC} = t_b^3 / 12 = 0.38 \cdot 6.49^3 / 12 = 8.6564 \text{ in}^4$$

$$J = (2 \times 6.49 \cdot 0.38^3 + (12.22 - 2 \cdot 0.38) \cdot 0.23^3) / 3 = 0.28389 \text{ in}^4$$

$$F_{CZ} = \frac{50(10)^6 1.75 \left(\frac{8.6564}{120} \right)}{33.38789} \sqrt{0.772 \frac{0.28389}{8.6564} + 9.87 \left(\frac{11.46}{120} \right)^2} 10^{-3} = 64.375 \text{ ksi}$$

which is larger than $0.55 \cdot F_Y = 19.8 \text{ ksi}$, so $F_{CZ} = 19.8 \text{ ksi}$

Axial Compression

$$\text{Critical } (kL/r) = 1.0 \cdot 120 / 1.5038 = 79.7978$$

As $(kL/r) < C_c$, the allow axial stress in compression is given by:

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

$$F_a = \frac{F_y}{F.S.} \left[1 - \frac{(kL/r)^2 F_y}{4\pi^2 E} \right] = \frac{36}{2.12} \left[1 - \frac{(79.79)^2 36}{4\pi^2 29,000} \right] = 13.58 \text{ ksi}$$

Actual Stress

Actual axial stress, $f_a = 25/7.65 = 3.26$ ksi

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.42 in section 10-36 to calculate the design ratio.

Actual bending stress = $f_{bz} = 56.5 \cdot 12/33.4 = 1.692 \cdot 12 = 20.3$ ksi

$$F_{ez} = \frac{\pi^2 E}{F.S. (kL/r)^2} = \frac{\pi^2 29,000}{2.12 (120/5.17)^2} = 250.6 \text{ ksi}$$

From Table 10-36A, $C_{mz} = 0.85$

Equation 10-42

$$\frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{\left(1 - \frac{f_a}{F'_{ez}}\right) F_{bz}} + \frac{C_{my} f_{by}}{\left(1 - \frac{f_a}{F'_{ey}}\right) F_{by}} = \frac{3.26}{13.58} + \frac{0.58 \cdot 20.3}{\left(1 - \frac{3.26}{250.6}\right) 19.8} + 0 = 1.122$$

For the end section, use Equation 10.43:

$$\frac{f_a}{0.472 F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{3.26}{0.472(36)} + \frac{20.3}{19.8} + 0 = 1.217$$

The critical stress ratio is thus 1.217. The value calculated by STAAD is 1.218

2. Member 2

Size W 12X26, L = 5 ft., a = 7.65 in², Sz = 33.4 in³

From observation Load case 1 will govern, Forces at the midspan are

- Fx = 8.71 kip (compression)
- Mz = 56.5 k-ft

Calculate the allowable stress as per Table 10.32.1A.

Bending Minor Axis

Allowable minor axis bending stress:

$$F_{TY} = F_{TZ} = 0.55 \cdot F_Y = 19.8 \text{ ksi}$$

Bending Major Axis

$$F_{cz} = \frac{50(10)^6 C_b \left(\frac{I_{yc}}{l} \right) \sqrt{0.772 \frac{J}{I_{yc}} + 9.87 \left(\frac{d}{l} \right)^2}}{S_{xc}} \leq 0.55 F_y$$

Where:

$$C_b = 1.75 + 1.05(M1/M2) + 0.3x(M1/M2)^2$$

$$M1 = 39.44 \text{ and } M2 = 677.96, \text{ so } C_b = 1.69$$

$$S_{zc} = \text{Section modulus with respect to the compression flange} \\ = 204 / (0.5 \cdot 12.22) = 33.38789 \text{ in}^3$$

$$I_{YC} = t_b^3 / 12 = 0.38 \cdot 6.49^3 / 12 = 8.6564 \text{ in}^4$$

$$J = (2 \times 6.49 \cdot 0.38^3 + (12.22 - 2 \cdot 0.38) \cdot 0.23^3) / 3 = 0.28389 \text{ in}^4$$

$$F_{cz} = \frac{50(10)^6 1.69 \left(\frac{8.6564}{60} \right) \sqrt{0.772 \frac{0.28389}{8.6564} + 9.87 \left(\frac{11.46}{60} \right)^2}}{33.38789} 10^{-3} = 227.34 \text{ ksi}$$

which is larger than $0.55 \cdot F_Y = 19.8 \text{ ksi}$, so $F_{CZ} = 19.8 \text{ ksi}$

Axial Compression

$$\text{Critical } (kL/r) = 1.0 \cdot 60 / 1.504 = 39.92$$

As $(kL/r) < C_c$, the allow axial stress in compression is given by:

$$F_a = \frac{F_y}{F.S.} \left[1 - \frac{(kL/r)^2 F_y}{4\pi^2 E} \right] = \frac{36}{2.12} \left[1 - \frac{(39.92)^2 36}{4\pi^2 29,000} \right] = 16.13 \text{ ksi}$$

Actual Stress

$$\text{Actual axial stress, } f_a = 8.71 / 7.65 = 1.138 \text{ ksi}$$

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.42 in section 10-36 to calculate the design ratio.

$$\text{Actual bending stress} = f_{bz} = 56.5 \cdot 12 / 33.4 = 1.691 \cdot 12 = 20.3 \text{ ksi}$$

$$(KL/r)_z = 1 \cdot 60 / 5.16 = 11.618$$

$$F_{ez} = \frac{\pi^2 E}{F.S. (kL/r)^2} = \frac{\pi^2 29,000}{2.12 (11.618)^2} = 998.5 \text{ ksi}$$

From Table 10-36A, $C_{mz} = 0.85$

Equation 10-42

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

$$\frac{f_a}{F_a} + \frac{C_{mz}f_{bz}}{\left(1 - \frac{f_a}{F'_{ez}}\right)F_{bz}} + \frac{C_{my}f_{by}}{\left(1 - \frac{f_a}{F'_{ey}}\right)F_{by}} = \frac{1.138}{16.13} + \frac{0.58 \cdot 20.3}{\left(1 - \frac{1.138}{998.5}\right)19.8} + 0 = 0.942$$

For the end section, use Equation 10.43:

$$\frac{f_a}{0.472F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{1.138}{0.472(36)} + \frac{20.3}{19.8} + 0 = 1.092$$

The critical stress ratio is thus 1.092. The value calculated by STAAD is 1.093.

3. Member 3

Size W 14X43, L = 11 ft., a = 12.6 in², Sz = 62.7 in³

From observation Load case 3 will govern, Forces at the end are

- $F_x = 25.5$ kip (compression)
- $M_z = 112.17$ k-ft

Calculate the allowable stress as per Table 10.32.1A.

Bending Minor Axis

Allowable minor axis bending stress:

$$F_{TY} = F_{TZ} = 0.55 \cdot F_Y = 19.8 \text{ ksi}$$

Bending Major Axis

$$F_{Cz} = \frac{50(10)^6 C_b}{S_{xc}} \left(\frac{I_{yc}}{l} \right) \sqrt{0.772 \frac{J}{I_{yc}} + 9.87 \left(\frac{d}{l} \right)^2} \leq 0.55 F_y$$

Where:

$$C_b = 1.75 + 1.05(M1/M2) + 0.3x(M1/M2)^2$$

$$M1 = 0, \text{ so } C_b = 1.75$$

$$S_{zc} = \text{Section modulus with respect to the compression flange} = 428 / (0.5 \cdot 13.66) = 62.7 \text{ in}^3$$

$$I_{YC} = tb^3/12 = 0.53 \cdot 8.0^3/12 = 22.61 \text{ in}^4$$

$$J = (2 \cdot 8.0 \cdot 0.53^3 + (13.66 - 2 \cdot 0.53) \cdot 0.305^3) / 3 = 0.913 \text{ in}^4$$

$$F_{Cz} = \frac{50(10)^6 1.75 \left(\frac{22.61}{132} \right) \sqrt{0.772 \frac{0.917}{22.61} + 9.87 \left(\frac{12.6}{132} \right)^2} 10^{-3} = 83.19 \text{ ksi}$$

which is larger than $0.55 \cdot F_Y = 19.8 \text{ ksi}$, so $F_{CZ} = 19.8 \text{ ksi}$

Axial Compression

$$\text{Critical } (kL/r) = 1.0 \cdot 132/1.894 = 69.69$$

As $(kL/r) < C_c$, the allow axial stress in compression is given by:

$$F_a = \frac{F_y}{F.S.} \left[1 - \frac{(kL/r)^2 F_y}{4\pi^2 E} \right] = \frac{36}{2.12} \left[1 - \frac{(69.69)^2 36}{4\pi^2 29,000} \right] = 14.39 \text{ ksi}$$

Actual Stress

$$\text{Actual axial stress, } f_a = 25.5 / 12.6 = 2.024 \text{ ksi}$$

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.42 in section 10-36 to calculate the design ratio.

$$\text{Actual bending stress} = f_{bz} = 112.17 \cdot 12/62.7 = 1.789 \cdot 12 = 21.467 \text{ ksi}$$

$$(KL/r)_z = 1 \cdot 132/5.828 = 22.649$$

$$F_{ez} = \frac{\pi^2 E}{F.S. (kL/r)^2} = \frac{\pi^2 29,000}{2.12 (22.648)^2} = 263.18 \text{ ksi}$$

$$\text{From Table 10-36A, } C_{mz} = 0.85$$

Equation 10-42

$$\frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{\left(1 - \frac{f_a}{F'_{ez}}\right) F_{bz}} + \frac{C_{my} f_{by}}{\left(1 - \frac{f_a}{F'_{ey}}\right) F_{by}} = \frac{2.024}{14.39} + \frac{0.58 \cdot 21.467}{\left(1 - \frac{2.024}{263.21}\right) 19.8} + 0 = 1.069$$

For the end section, use Equation 10.43:

$$\frac{f_a}{0.472 F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{2.024}{0.472(36)} + \frac{21.467}{19.8} + 0 = 1.203$$

The critical stress ratio is thus 1.203. The value calculated by STAAD is 1.204.

4. Member 4

$$\text{Size W } 14 \times 43, L = 4 \text{ ft.}, a = 12.6 \text{ in}^2, S_z = 62.6 \text{ in}^3$$

From observation Load case 3 will govern, Forces at the end are

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

- $F_x = 8.75$ kip (tension)
- $M_z = 112.17$ k-ft

Calculate the allowable stress as per Table 10.32.1A.

Bending Minor Axis

Allowable minor axis bending stress:

$$F_{TY} = F_{TZ} = 0.55 \cdot F_Y = 19.8 \text{ ksi}$$

Bending Major Axis

$$F_{Cz} = \frac{50(10)^6 C_b}{S_{xc}} \left(\frac{I_{yc}}{l} \right) \sqrt{0.772 \frac{J}{I_{yc}} + 9.87 \left(\frac{d}{l} \right)^2} \leq 0.55 F_y$$

Where:

$$C_b = 1.75 + 1.05(M1/M2) + 0.3 \cdot (M1/M2)^2$$

$$M1 = -191.36 \text{ Kip-in}, M2 = -1346.08 \text{ Kip-in} \text{ so } C_b = 1.606$$

$$I_{YC} = tb^3/12 = 0.53 \cdot 8.0^3/12 = 22.61 \text{ in}^4$$

$$J = (2 \cdot 8.0 \cdot 0.53^3 + (13.66 - 2 \cdot 0.53) \cdot 0.305^3)/3 = 0.913 \text{ in}^4$$

$$F_{Cz} = \frac{50(10)^6 1.606}{62.6} \left(\frac{22.61}{48} \right) \sqrt{0.772 \frac{0.911}{22.61} + 9.87 \left(\frac{12.6}{48} \right)^2} 10^{-3} = 508.8 \text{ ksi}$$

which is larger than $0.55 \cdot F_Y = 19.8$ ksi, so $F_{CZ} = 19.8$ ksi

Axial Tension

Note: No connection information is specified and no reduction of section is assumed.

$$F_t = 0.55 \cdot F_Y = 19.8 \text{ ksi}$$

Actual Stress

Actual axial stress, $f_a = 8.75 / 12.6 = 0.694$ ksi

Actual bending stress = $f_{bz} = 112.17 \cdot 12/62.7 = 1.789 \cdot 12 = 21.47$ ksi, which exceeds F_{CZ} .

$$f_{bz}/F_{CZ} = 21.47/19.8 = 1.084$$

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.43 in section 10-36 to calculate the design ratio for the end section.

$$\frac{f_a}{0.472F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{0.694}{0.472(36)} + \frac{21.467}{19.8} + 0 = 1.125$$

The critical stress ratio is thus 1.125. The value calculated by STAAD is 1.126.

5. Member 5

Size W 16X36, L = 5 ft., a = 10.6 in², Sz = 56.5 in³

From observation Load case 3 will govern, Forces at the end are

- Fx = 14.02 kip (compression)
- Mz = 57.04 k-ft

Calculate the allowable stress as per Table 10.32.1A.

Bending Minor Axis

Allowable minor axis bending stress:

$$F_{TY} = F_{TZ} = 0.55 \cdot F_Y = 19.8 \text{ ksi}$$

Bending Major Axis

$$F_{cz} = \frac{50(10)^6 C_b \left(\frac{I_{yc}}{l} \right) \sqrt{0.772 \frac{J}{I_{yc}} + 9.87 \left(\frac{d}{l} \right)^2}}{S_{xc}} \leq 0.55 F_y$$

Where:

$$C_b = 1.75 + 1.05(M1/M2) + 0.3 \cdot (M1/M2)^2$$

$$M1 = 40.14, M2 = -684.4 \text{ so } C_b = 1.81$$

$$S_{zc} = \text{Section modulus with respect to the compression flange} = 448 / (0.5 \cdot 15.86) = 56.5 \text{ in}^3$$

$$I_{Yc} = tb^3/12 = 0.43 \cdot 6.99^3/12 = 12.238 \text{ in}^4$$

$$J = (2 \cdot 6.99 \cdot 0.43^3 + (15.86 - 2 \cdot 0.43) \cdot 0.29^3)/3 = 0.5 \text{ in}^4$$

$$F_{cz} = \frac{50(10)^6 1.81 \left(\frac{12.238}{60} \right) \sqrt{0.772 \frac{0.5}{12.238} + 9.87 \left(\frac{15}{60} \right)^2}}{56.5} 10^{-3} = 263.1 \text{ ksi}$$

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

which is larger than $0.55 \cdot F_Y = 19.8$ ksi, so $F_{CZ} = 19.8$ ksi

Axial Compression

Critical $(kL/r) = 1.0 \cdot 60/1.52 = 69.69$

As $(kL/r) < C_c$, the allow axial stress in compression is given by:

$$F_a = \frac{F_y}{F.S.} \left[1 - \frac{(kL/r)^2 F_y}{4\pi^2 E} \right] = \frac{36}{2.12} \left[1 - \frac{(39.474)^2 36}{4\pi^2 29,000} \right] = 16.15 \text{ ksi}$$

Actual Stress

Actual axial stress, $f_a = 14.02 / 10.6 = 1.323$ ksi

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.42 in section 10-36 to calculate the design ratio.

Actual bending stress = $f_{bz} = 57.04 \cdot 12/56.5 = 1.001 \cdot 12 = 12.115$ ksi

$$(KL/r)_z = 1 \cdot 60/6.5 = 9.231$$

$$F_{ez} = \frac{\pi^2 E}{F.S. (kL/r)^2} = \frac{\pi^2 29,000}{2.12 (9.231)^2} = 1,584.4 \text{ ksi}$$

From Table 10-36A, $C_{mz} = 0.85$

Equation 10-42

$$\frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{\left(1 - \frac{f_a}{F'_{ez}}\right) F_{bz}} + \frac{C_{my} f_{by}}{\left(1 - \frac{f_a}{F'_{ey}}\right) F_{by}} = \frac{1.323}{16.149} + \frac{0.85 \cdot 12.115}{\left(1 - \frac{1.323}{1,584.4}\right) 19.8} + 0 = 0.602$$

For the end section, use Equation 10.43:

$$\frac{f_a}{0.472 F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{1.323}{0.472(36)} + \frac{12.115}{19.8} + 0 = 0.689$$

The critical stress ratio is thus 0.689. The value calculated by STAAD is 0.690.

6. Member 6

Size W 16X36, L = 16 ft., a = 10.6 in², Sz = 56.5 in³

From observation Load case 3 will govern, Forces at the end are

- Fx = 10.2 kip (compression)
- Mz = 62.96 k-ft

Calculate the allowable stress as per Table 10.32.1A.

Bending Minor Axis

Allowable minor axis bending stress:

$$F_{TY} = F_{TZ} = 0.55 \cdot F_Y = 19.8 \text{ ksi}$$

Bending Major Axis

$$F_{Cz} = \frac{50(10)^6 C_b \left(\frac{I_{yc}}{l} \right) \sqrt{0.772 \frac{J}{I_{yc}} + 9.87 \left(\frac{d}{l} \right)^2}}{S_{xc}} \leq 0.55 F_Y$$

Where:

$$C_b = 1.75 + 1.05(M1/M2) + 0.3 \cdot (M1/M2)^2$$

$$M1 = 8.947 \text{ M2} = 183.05 \text{ so } C_b = 1.69$$

$$S_{zc} = \text{Section modulus with respect to the compression flange} = 448 / (0.5 \cdot 15.86) = 56.5 \text{ in}^3$$

$$I_{YC} = tb^3/12 = 0.43 \cdot 6.99^3/12 = 12.238 \text{ in}^4$$

$$J = (2 \cdot 6.99 \cdot 0.43^3 + (15.86 - 2 \cdot 0.43) \cdot 0.29^3) / 3 = 0.5 \text{ in}^4$$

$$F_{Cz} = \frac{50(10)^6 1.81 \left(\frac{12.238}{192} \right) \sqrt{0.772 \frac{0.5}{12.238} + 9.87 \left(\frac{15}{192} \right)^2}}{56.5} 10^{-3} = 287.9 \text{ ksi}$$

which is larger than $0.55 \cdot F_Y = 19.8 \text{ ksi}$, so $F_{CZ} = 19.8 \text{ ksi}$

Axial Compression

$$\text{Critical } (kL/r) = 1.0 \cdot 192/1.52 = 126.3$$

As $(kL/r) > C_c$, the allow axial stress in compression is given by:

$$F_a = \frac{\pi^2 E}{F.S. (kL/r)^2} = \frac{\pi^2 29,000}{2.12(126.3)^2} = 8.46 \text{ ksi}$$

Actual Stress

$$\text{Actual axial stress, } f_a = 10.2 / 10.6 = 0.962 \text{ ksi}$$

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.42 in section 10-36 to calculate the design ratio.

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

$$\text{Actual bending stress} = f_{bz} = 62.96 \cdot 12/56.5 = 1.114 \cdot 12 = 13.37 \text{ ksi}$$

$$(KL/r)_z = 1 \cdot 192/6.51 = 29.49$$

$$F_{ez} = \frac{\pi^2 E}{F.S. (KL/r)^2} = \frac{\pi^2 29,000}{2.12(29.49)^2} = 155.2 \text{ ksi}$$

From Table 10-36A, $C_{mz} = 0.85$

Equation 10-42

$$\frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{\left(1 - \frac{f_a}{F'_{ez}}\right) F_{bz}} + \frac{C_{my} f_{by}}{\left(1 - \frac{f_a}{F'_{ey}}\right) F_{by}} = \frac{0.962}{8.46} + \frac{0.85 \cdot 13.37}{\left(1 - \frac{0.926}{155.2}\right) 19.8} + 0 = 0.691$$

For the end section, use Equation 10.43:

$$\frac{f_a}{0.472 F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{0.962}{0.472(36)} + \frac{13.37}{19.8} + 0 = 0.732$$

The critical stress ratio is thus 0.732. The value calculated by STAAD is 0.732

Note: The program gives this value when the slenderness check is suppressed (**MAIN 1.0** for member 6); otherwise the member fails as a compression member with a slenderness parameter greater than 120).

7. Member 7

Size W 16X36, L = 4 ft., a = 10.6 in², Sz = 56.5 in³

From observation Load case 3 will govern, Forces at the midspan are

- Fx = 24.05 kip (tension)
- Mz = 62.96 k-ft

Calculate the allowable stress as per Table 10.32.1A

Bending Minor Axis

Allowable minor axis bending stress:

$$F_{TY} = F_{TZ} = 0.55 \cdot F_Y = 19.8 \text{ ksi}$$

Bending Major Axis

$$FCZ = 0.55 \cdot F_Y = 19.8 \text{ ksi}$$

Axial Tension

Note: No connection information is specified and no reduction of section is assumed.

$$F_a = 0.55 \cdot F_Y = 19.8 \text{ ksi}$$

Actual Stress

Actual axial stress, $f_a = 24.05 / 10.6 = 2.268$ ksi, hence, ok.

Actual bending stress = $f_{bz} = 62.96 \cdot 12 / 56.5 = 1.114 \cdot 12 = 13.37$ ksi

So the combined ratio is

$$f_a / F_a + f_{bz} / F_{TZ} + f_{by} / F_{TY} = 2.268 / 19.8 + 13.37 / 19.8 + 0 = 0.790$$

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.43 in section 10-36 to calculate the design ratio for the end section.

$$\frac{f_a}{0.472F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{2.268}{0.472(36)} + \frac{13.37}{19.8} + 0 = 0.809$$

The critical stress ratio is thus 0.809. The value calculated by STAAD is 0.809 .

8. Member 8

Size L4x4x1/4, L = 7.07 ft., a = 1.938 in²

From observation Load case 1 will govern, Forces

$F_x = 23.04$ kip (compression)

Calculate the allowable stress as per Table 10.32.1A

Axial Compression

Critical $(KL/r)_y = 1.0 \cdot 7.07 \cdot 12 / 0.795 = 106.7$

As $(KL/r) < C_c$, the allow axial stress in compression is given by:

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

$$F_a = \frac{F_y}{F.S.} \left[1 - \frac{(kL/r)^2 F_y}{4\pi^2 E} \right] = \frac{36}{2.12} \left[1 - \frac{(106.7)^2 36}{4\pi^2 29,000} \right] = 10.89 \text{ ksi}$$

Actual Stress

Actual axial stress, $f_a = 23.04 / 1.938 = 11.88$ ksi

$$f_a / F_a = 11.88 / 10.89 = 1.091$$

The value calculated by STAAD is 1.091.

9. Member 9

Size L5x5x3/8, L = 5.657 ft, A = 3.61 in²

From observation Load case 1 will govern, Forces

$F_x = 48.44$ kip (compression)

Calculate the allowable stress as per Table 10.32.1A

Axial Compression

Critical $(KL/r)_y = 1.0 \cdot 5.657 \cdot 12 / 0.99 = 68.57$

As $(kL/r) < C_c$, the allow axial stress in compression is given by:

$$F_a = \frac{F_y}{F.S.} \left[1 - \frac{(kL/r)^2 F_y}{4\pi^2 E} \right] = \frac{36}{2.12} \left[1 - \frac{(68.57)^2 36}{4\pi^2 29,000} \right] = 14.47 \text{ ksi}$$

Actual Stress

Actual axial stress, $f_a = 48.44 / 3.61 = 13.42$ ksi

$$f_a / F_a = 13.42 / 14.47 = 0.927$$

The value calculated by STAAD is 0.928.

2.5.2 AASHTO (LRFD)

The following outlines the implementation of the AASHTO Standard Specifications for Highway Bridges (LRFD, 1998) which has been implemented in STAAD.Pro.

2.5.2.1 General

The design philosophy embodied in the Load and Resistance Factor Design (LRFD) Specification is built around the concept of limit state design, the current state-of-the-art in structural engineering. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD implementation of AASTHO-LRFD, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks that code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the AASTHO-LRFD specifications as implemented in STAAD steel design.

2.5.2.2 Capacities per AASHTO (LRFD) Code

Axial Strength

The criteria governing the capacity of tension members is based on two limit states. The limit state of yielding in the gross section is intended to prevent excessive elongation of the member. The second limit state involves fracture at the section with the minimum effective net area. The net section area may be specified through the use of the parameter NSF. STAAD calculates the tension capacity of a given member based on these two limit states and proceeds with member selection or code check accordingly

$$P_r = \phi_y P_{ny} = \phi_y F_y A_g$$

$$P_r = \phi_u P_{nu} = \phi_u F_u A_n U$$

Where:

$$P_{ny} = \text{Nominal tensile resistance for yielding in gross section (kip)}$$

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

F_y = Yield strength (ksi)

A_g = Gross cross-sectional area of the member (in²)

P_{nu} = Nominal tensile resistance for the fracture in the net section (kip)

F_u = Tensile strength (ksi)

A_n = Net area of the member

U = reduction factor to account for shear lag.

ϕ_y = resistance factor for yielding of tension member

ϕ_u = resistance factor for fracture of tension members

Allowable compressive stress on the gross section of axially loaded compression members is calculated based on the following formula:

$$\lambda = \left(\frac{Kl}{r_z \pi} \right)^2 \frac{F_y}{E}$$

if $\lambda \leq 2.25$

Nominal compressive resistance,

$$P_n = 0.66 \lambda^2 F_y A_s$$

if $\lambda > 2.25$

Nominal compressive resistance

$$P_n = 0.88 F_y A_s / \lambda$$

Where:

A_s = Gross sectional area

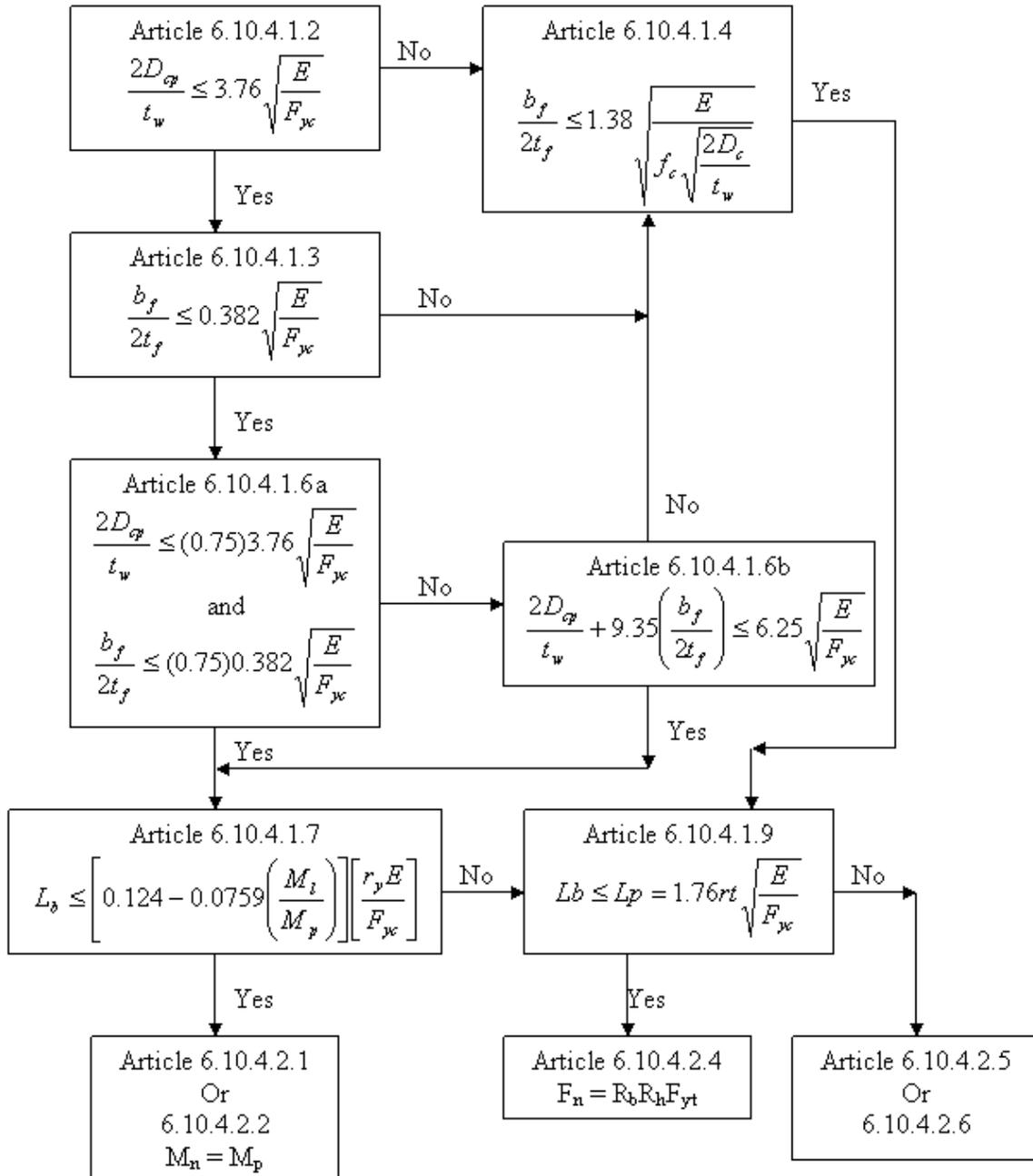
The Factored resistance

$$P_r = \phi_c P_n$$

Bending Strength

The flow to calculate the allowable bending strength for rolled shape girders and built-up sections is given by the following flow chart.

Figure 2-14: Flow used to calculate the allowable bending strength for rolled shape girders and built-up sections



Shear Strength

The nominal shear resistance of un-stiffened webs of homogeneous girders shall be calculated as.

If $D/t_w \leq 2.46\sqrt{E/F_y}$, then

$$V_n = V_p = 0.58F_{yw} D t_w$$

If $2.46\sqrt{E/F_y} < D/t_w \leq 3.07\sqrt{E/F_y}$, then

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

$$V_n = 1.48 \cdot t_w^2 \sqrt{E \cdot F_y}$$

If $D/t_w > 3.07 \sqrt{E/F_y}$, then

$$V_n = 4.55 \cdot t_w^3 \cdot E/D$$

Bending-Axial Interaction

Members subjected to both axial forces and bending moments are proportioned according to section 6.9.2.2 of the AASHTO steel code. All members subject to bending and axial compression or axial tension are required to satisfy the following formula:

If $P_u/P_r < 0.2$, then

$$\frac{P_u}{2.0P_r} + \left(\frac{M_{ux}}{M_{rx}} + \frac{M_{uy}}{M_{ry}} \right) \leq 1.0$$

If $P_u/P_r \geq 0.2$, then

$$\frac{P_u}{P_r} + \frac{8}{9} \left(\frac{M_{ux}}{M_{rx}} + \frac{M_{uy}}{M_{ry}} \right) \leq 1.0$$

2.5.2.3 AASHTO (LRFD) Design Parameters

The following table outlines the parameters that can be used with the AASHTO (LRFD) code along with the default values used if not explicitly specified.

Table 2-9: AASHTO (LRFD) Design Parameters

Parameter Name	Default Value	Description
<u>BEAM</u>	1.0	Identify where beam checks are performed: o = Perform design at ends and those locations specified in the SECTION command. 1 = Perform design at ends and 1/12th section locations along member length.
<u>DMAX</u>	1000	Maximum allowed section depth (in current length units) for a section to be selected with the SELECT command.
<u>DMIN</u>	o	Minimum allowed section depth (in current length units) for a section to be selected with the SELECT command.
<u>DFL</u>	o	“Deflection Length”/Max allowable local deflection If set to o, (default) then no deflection check is performed.
<u>DJ1</u>	Start joint of member	Joint No. denoting starting point for calculating “Deflection Length”.

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

Parameter Name	Default Value	Description
<u>DJ2</u>	End joint of member	Joint No. denoting ending point for calculating "Deflection Length".
<u>GRADE</u>	1	Grade of Steel: 1: Grade 36 2: Grade 50 3 Grade 50W 4: Grade 70W 5: Grade 100/100W Refer to AASHTO LRFD, Table 6.4.1-1
<u>KY</u>	1.0	K value in local y-axis. Usually, this is the minor axis.
<u>KZ</u>	1.0	K value in local z-axis. Usually, this is the major axis.
<u>LY</u>	Member Length	Length to calculate slenderness ratio for buckling about the local Y axis.
<u>LZ</u>	Member Length	Same as above except in local z-axis.
<u>MAIN</u>	0.0	Flag for checking slenderness limit: 0.0 =check for slenderness 1.0 =suppress slenderness check

Parameter Name	Default Value	Description
<u>NSF</u>	1.0	Net Section Factor. Ratio of (Net Area)/(Gross Area)
<u>NSF</u>	1.0	Net section factor for tension members.
<u>TRACK</u>	0	Level of detail in Output File: 0 = Print the design output at the minimum detail level. 1 = Print the design output at the intermediate detail level. 2 = Print the design output at maximum detail level..
<u>UNB</u>	Member Length	Unsupported length of bottom flange. Used for calculating the moment of resistance when the bottom of beam is in compression.
<u>UNT</u>	Member Length	Unsupported length of top flange. Used for calculating the moment of resistance when top of beam is in compression.

2.5.2.3.1 AASHTO (LRFD) Verification Problem

The following compares the solution of a design performed using STAAD.Pro against a hand calculation.

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

Type

AASHTO (LRFD) Steel Design.

Reference

The following step by step hand calculation as per AASHTO_LRFD(1998) code.

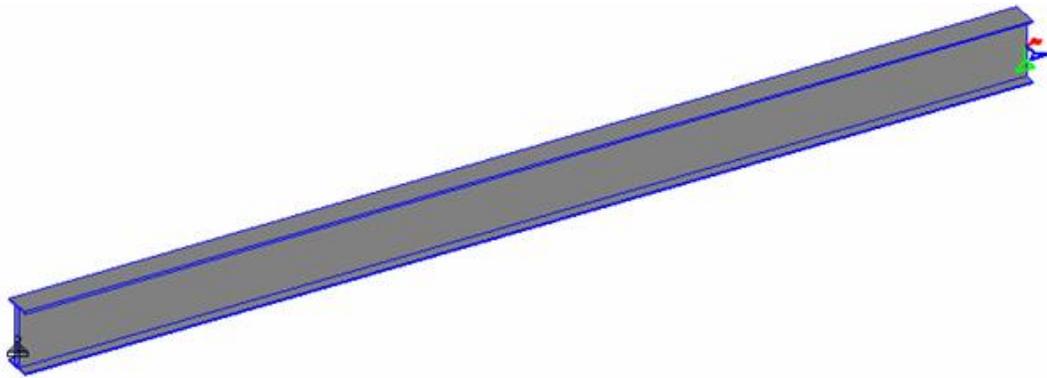
Problem

Determine the allowable resistances (AASHTO_LRFD, 1998 code) for the member of the structure as shown in figure. Also, perform a code check for the member based on the results of the analysis.

A 48 foot long, non-composite girder is assumed to be simply supported. The section is a plate girder with 16" x 1" plate flanges and a 34" x 3.6" plate web (36 in. total depth). All plates are grade 36 steel.

The beam is subject to a 9.111 kip/ft uniform load.

Figure 2-15: AASHTO LRFD verification example



Comparison

Table 2-10: Comparison of results for AASHTO LRFD verification example

Value	STAAD.Pro Results	Hand Calculations	Difference
Critical Interaction Ratio	1.654	1.657	none

Solution

$$A = 154.4 \text{ in}^2, I_z = 21,594 \text{ in}^4, I_y = 814.9 \text{ in}^4$$

$$S_z = 21,594(2)/36 = 1,200 \text{ in}^3$$

$$r_y = \sqrt{\frac{I_y}{A}} = \sqrt{\frac{814.9}{154.4}} = 2.30 \text{ in}$$

$$r_z = \sqrt{\frac{I_z}{A}} = \sqrt{\frac{21,594}{154.4}} = 11.83 \text{ in}$$

From observation Load case 1 will govern,

$$M_z = 2,624 \text{ kip-ft}$$

Axial Compression Capacity

Refer Clause 6.9.4 of the code.

$$(kL/r)_y = 0.333 \times 576 / 2.297 = 83.50$$

$$(kL/r)_z = 1 \times 576 / 11.826 = 48.71$$

$$(kL/r)_{\text{crit}} = 83.50 < 120, \text{ ok.}$$

Calculation of Width/Thickness ratio for axial compression

Plate buckling coefficients taken from Table 6.9.4.2-1:

$$k_w = 1.49, k_f = 0.56$$

Slenderness ratio for the web:

$$(d - 2 \cdot t_f) / t_w = (36 - 2 \cdot 1) / 3.6 = 9.444$$

Slenderness ratio for the 1/2 flange:

$$b_f / (2 \cdot t_f) = 16 / (2 \cdot 1) = 8.0$$

Critical ratio for web:

$$k_w \sqrt{\frac{E}{F_y}} = 1.49 \sqrt{\frac{29,000}{36}} = 42.29 > 9.444$$

Critical ratio for flange:

$$k_f \sqrt{\frac{E}{F_y}} = 0.56 \sqrt{\frac{29,000}{36}} = 15.89 > 8.0$$

Thus, **OK** [AASHTO LRFD Cl. 6.4.9.2]

Slenderness ratio about major and minor axis:

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

$$\lambda_z = \left(\frac{Kl}{r_z \pi} \right)^2 \frac{F_y}{E} = \left(\frac{1.0 \cdot 576}{11.83\pi} \right)^2 \frac{36}{29,000} = 0.298$$

$$\lambda_y = \left(\frac{Kl}{r_y \pi} \right)^2 \frac{F_y}{E} = \left(\frac{0.333 \cdot 576}{2.30\pi} \right)^2 \frac{36}{29,000} = 0.877$$

λ_y governs, thus $\lambda = 0.877$

$\lambda < 2.25$, so Equation 6.9.4.1-1 is used to determine the nominal compressive resistance

$$P_n = 0.66 \lambda F_y A_s = 0.66^{0.877} \cdot 36 \cdot 154.4 = 3,861 \text{ kips}$$

The factored compressive resistance, $P_r = \phi_c P_n = 0.9 \cdot 3,861 = 3,475 \text{ kips}$

Major Axis Bending Capacity

The compression flange moment of inertia:

$$I_{yc} = 1(16)^3/12 = 341.3 \text{ in}^4$$

$$I_{yc}/I_y = 341.3/814.9 = 0.419, > 0.1 \text{ and } < 0.9$$

Thus, **OK** [AASHTO LRFD Cl. 6.10.2.1]

$$\frac{2D_o}{t_w} = \frac{2(17)}{3.6} = 9.445 < 6.77 \sqrt{\frac{E}{F_y}} = 6.77 \sqrt{\frac{29,000}{36}} = 192.1$$

Thus, **OK** [AASHTO LRFD Cl. 6.10.2.2]

Calculation of depth of the web in compression at the plastic moment, D_{cp}
(Clause 6.10.3.3.2)

$$\text{Area of Web, } A_w = (36 - 2 \times 1) \times 3.6 = 122.4 \text{ in}^2$$

Area of flange area in tension, A_{ft} = Area of flange in compression, $A_{fc} = 16 \times 1 = 16 \text{ in}^2$

$$D_{cp} = \left(\frac{D_w}{2A_w F_y} \right) F_y (A_{ft} + A_w - A_{fc}) = \left(\frac{34}{2 \cdot 122.4 \cdot 36} \right) 36 (16 + 122.4 - 16) = 17 \text{ in}$$

$$\frac{2D_{cp}}{t_w} = \frac{2(17)}{3.6} = 9.445 < 3.76 \sqrt{\frac{E}{F_y}} = 3.76 \sqrt{\frac{29,000}{36}} = 108.1$$

Thus, **OK** [AASHTO LRFD Cl. 6.10.4.1.2]

$$\frac{b_f}{t_f} = \frac{16.0}{2(1.0)} = 8.0 < 0.382 \sqrt{\frac{E}{F_y}} = 0.382 \sqrt{\frac{29,000}{36}} = 10.98$$

Thus, **OK** [AASHTO LRFD Cl. 6.10.4.3]

Check clause 6.10.4.1.6a

$$(B/t)_{\text{flange}} < 0.75 \times (B/t)_{\text{flange_limit}} = 0.75 \times 10.978$$

$$(D/T)_{\text{web}} < 0.75 \times (D/T)_{\text{web_limit}} = 0.75 \times 108.057$$

Check clause 6.10.4.1.7

$$M_{pz} = P_z \times F_y = 1,600 \text{ in}^3(36 \text{ ksi}) = 57,614 \text{ in-k}$$

$$L_b = \left[0.124 - 0.0759 \left(M_l / M_{pz} \right) \right] \left[\frac{r_y E}{F_y} \right] = \left[0.124 - 0.0759 \left(0 / 57,614 \right) \right] \left[\frac{2.30(29,000)}{36} \right] = 229.5 \text{ in}$$

Unsupported length, $L_u = 576 \text{ in} > L_b$; section is non-compact.

Check clause 6.10.4.1.9

A notional section comprised of the compression flange and one-third of the depth of the web in compression, taken about the vertical axis.

$$A_{rt} = A_{cf} + \left(\frac{c - t_f}{3} \right) t_w = 16 + \left(\frac{18 - 1.0}{3} \right) 3.6 = 36.4 \text{ in}^2$$

$$I_{rt} = t_f \left[\left(\frac{c - t_f}{3} \right) \frac{t_w^3}{12} \right] = 1 \left[\left(\frac{18 - 1}{3} \right) \frac{3.6^3}{12} \right] = 363.4 \text{ in}^4$$

$$r_{rt} = \sqrt{\frac{363.4}{36.4}} = 3.149 \text{ in}$$

$$L_p = 1.76 \left(3.159 \right) \sqrt{\frac{29,000}{36}} = 159.8 \text{ in}$$

$$L_b > L_p$$

Clause 6.10.4.2.6

$$L_r = 4.44 \sqrt{\frac{I_{yc} d E}{S_{xc} F_y}} = 4.44 \sqrt{\frac{341.3(36) 29,000}{1,200 36}} = 403.3 \text{ in}$$

Minimum radius of gyration of the compression flange taken about the vertical axis:

$$r_t = \sqrt{\frac{I_{yc}}{A_{cf}}} = \sqrt{\frac{341.3}{16}} = 4.619 \text{ in}$$

Hybrid factor, $R_h = 1.0$ (For Homogeneous sections, Hybrid factors shall be taken as 1.0 per clause 6.10.4.3.1)

As per clause 6.10.4.3.2, Load-shedding factor, R_b

If area of the compression flange, $A_{cf} \geq$ the area of the tension flange, A_{tf}

$$l_b = 5.76$$

$$D_c = 17.00$$

Section 2 American Steel Design

2.5 Steel Design per AASHTO Specifications

$$\frac{2D_c}{t_w} = \frac{2(17)}{3.6} = 9.445 < L_b \sqrt{\frac{E}{F_y}} = 5.76 \sqrt{\frac{29,000}{36}} = 163.5$$

Thus:

$$R_{b,comp} = R_{b,ten} = 1.0$$

$L_r < L_b$

$$C_b = 1.75 + 1.05(M1/M2) + 0.3x(M1/M2)^2$$

Here $M_1 = 0$, $M_2 = 0$ so $C_b = 1.75$

$$M_{nz,comp} = C_{bz} R_{b,comp} R_h \frac{M_y}{2} \left(\frac{L_r}{L_b} \right)^2 = 1.75 \left(1.0 \right) \left(1.0 \right) \left[\frac{36(1,200)}{2} \right] \left(\frac{408.4}{576} \right)^2 = 19,000 \text{ kip} \cdot \text{in}$$

$$M_{nz,ten} = R_{b,ten} R_h F_y Z = 1.0 (1.0) (36) (1,200) = 43,188 \text{ kip} \cdot \text{in}$$

$$M_{nz} = 19,000 \text{ kip} \cdot \text{in}$$

Resisting Moment

$$M_r = Q_f \cdot M_n = 1.0 (19,000) = 19,000 \text{ kip} \cdot \text{in}$$

Actual Moment = 31,488 kip·in

Interaction ratio = 31,488 / 19,000 = 1.657

STAAD Input File

The following input is used in this verification example.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 23-MAY-11
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 38 0 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 1
UNIT INCH KIP
WIDE FLANGE
AASHTOGIRDER
154.4 36 3.6 16 1 21593.85 814.86 539.4 129.6 32.0
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
```

```

DENSITY 0.000283
ALPHA 6E-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 UPTABLE 1 AASHTOGIRDER
CONSTANTS
MATERIAL STEEL ALL
UNIT FEET KIP
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
UNIT INCHES KIP
LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.76
PERFORM ANALYSIS
PARAMETER 1
CODE AASHTO LRFD
TRACK 2 ALL
CHECK CODE ALL
FINISH

```

2.6 Design per American Cold Formed Steel Code

Provisions of the AISI Specification for the Design of Cold-Formed Steel Structural Members, 1996 Edition have been implemented. The program allows design of single (non-composite) members in tension, compression, bending, shear, as well as their combinations using the LRFD Method. For flexural members, the Nominal Section Strength is calculated on the basis of initiation of yielding in the effective section (Procedure I). Strength increase from Cold Work of Forming is a user selectable option.

2.6.1 Cross-Sectional Properties

you specifies the geometry of the cross-section by choosing one of the section shape designations from the STAAD Steel Tables for cold-formed sections, which mirror the Gross Section Property Tables published in the "Cold- Formed Steel Design Manual", AISI, 1996 Edition.

The Tables are currently available for the following shapes:

- Channel with Lips
- Channel without Lips
- Angle with Lips

Section 2 American Steel Design

2.6 Design per American Cold Formed Steel Code

- Angle without Lips
- Z with Lips
- Z without Lips
- Hat

Shape selection may be done using the member property pages of the graphical user interface (GUI) or by specifying the section designation symbol in the input file. Details of the latter are explained below.

2.6.2 The AISI Steel Section Library

The command-line syntax for assigning steel sections from the AISI library is as explained below:

C-Section with Lips

```
20 TO 30 TA ST 14CS3.75X135
33 36 TA ST 12CS1.625X102
42 43 TA ST 4CS4X060
```

C-Section with Lips

```
50 TO 60 TA ST 10 CU1.25X071
32 33 TA ST 3CU1.25X057
21 28 TA ST 1.5CU1.25X035
```

Z-Section with Lips

```
1 3 4 TA ST 12ZS3.25X135
33 45 TA ST 10ZS3X060
12 13 TA ST 6ZS2X048
```

Z-Section without Lips

```
2 3 TA ST 12ZU1.25X105
4 5 TA ST 4ZU1.25X036
6 7 TA ST 1.5ZU1.25X048
```

Equal Leg Angles without Lips

8 9 TA ST 4LS4X105
 10 11 TA ST 3LS3X060
 12 13 TA ST 2LS2X075

Equal Leg Angles without Lips

1 5 TA ST 4LU4X135
 7 8 TA ST 2.5LU2.5X105
 4 9 TA ST 2LU2X060

Hat Sections without Lips

4 8 TA ST 10HU5X075
 5 6 TA ST 6HU9X105
 1 7 TA ST 3HU4.5X135

2.6.3 Current Limitations

At the present time, only standard single sections are available for specification. Options such as double angles, double channels, and user provided sections including pipes and tubes will be available at a later date. Additionally, combination sections, such as an angle placed on top of a channel, or a plate welded to the top, bottom or side of one of the above shapes, are not available at this time.

STAAD.Pro uses unreduced section properties in the structure analysis stage. Both unreduced and effective section properties are used in the design stage, as applicable.

2.6.4 Design Procedure

The following two design modes are available:

1. Code Checking

The program compares the resistance of members with the applied load effects, in accordance with the LRFD Method of the AISI code. Code checking is carried out for locations specified by the user via the SECTION command or the BEAM parameter. The results are presented in a form of a PASS/FAIL identifier and a RATIO of load effect to resistance for each

Section 2 American Steel Design

2.6 Design per American Cold Formed Steel Code

member checked. The user may choose the degree of detail in the output data by setting the TRACK parameter.

2. Member Selection

You may request that the program search the cold formed steel shapes database (AISI standard sections) for alternative members that pass the code check and meet the least weight criterion. In addition, a minimum and/or maximum acceptable depth of the member may be specified. The program will then evaluate all database sections of the type initially specified (i.e., channel, angle, etc.) and, if a suitable replacement is found, present design results for that section. If no section satisfying the depth restrictions or lighter than the initial one can be found, the program leaves the member unchanged, regardless of whether it passes the code check or not.

The program calculates effective section properties in accordance with the following Sections:

- B2.1, Uniformly Compressed Stiffened Elements
- B2.3, Webs and Stiffened Elements with Stress Gradient
- B3.1, Uniformly Compressed Unstiffened Elements
- B3.2, Unstiffened Elements and Edge Stiffeners with Stress Gradient
- B4.2, Uniformly Compressed Elements with an Edge Stiffener

Cross-sectional properties of members are checked for compliance with the following Sections:

- B1.1(a), Maximum Flat-Width-to-Thickness Ratios, and
- B1.2, Maximum Web Depth-to-Thickness Ratio

The program checks member strength in accordance with Chapter C of the specification as follows:

1. Tension Members

Resistance is calculated in accordance with Section C2.

2. Flexural Members

a. C3.1, Strength for bending only:

- C3.1.1, Nominal Section Strength, Procedure I
- C3.1.2, Lateral Buckling Strength

- b. C_{3.2}, Strength for Shear Only
- c. C_{3.3}, Strength for Combined Bending and Shear
- 3. Concentrically Loaded Compression Members.
 - a. C_{4.1}, Sections not subject to Torsional or Torsional-Flexural Buckling, and
 - b. C_{4.2}, Doubly or Singly Symmetric sections subject to Torsional or Torsional-Flexural Buckling.
- 4. Combined Axial Load and Bending.
 - a. C_{5.1}, Combined Tensile Axial Load and Bending, and
 - b. C_{5.2}, Combined Compressive Axial Load and Bending.

2.6.5 Design Parameters

The following table contains the input parameters for specifying values of design variables and selection of design options.

Table 2-11: AISI Cold Formed Steel Design Parameters

Parameter Name	Default Value	Description
<u>BEAM</u>	1.0	0.0 = design at ends and those locations specified by the SECTION command. 1.0 = design at ends and at every 1/12 th point along member length.
<u>CMZ</u>	1.0	End moment coefficient for bending about z-axis. See AISI C5.2.2. Used for combined axial load and bending design. Values range from 0.4 to 1.0.
<u>CMY</u>	0.0	End moment coefficient for bending about y-axis. See AISI C5.2.2. Used for combined axial load and bending design. Values range from 0.4 to 1.0.

Section 2 American Steel Design

2.6 Design per American Cold Formed Steel Code

Parameter Name	Default Value	Description
<u>CWY</u>	0	Specifies whether the cold work of forming strengthening effect should be included in resistance computation. See AISI A7.2. 0 = effect <i>not</i> included 1 = effect included
<u>DMAX</u>	1000.0	Maximum depth permissible for the section during member selection, in current units.
<u>DMIN</u>	0.0	Minimum depth required for the section during member selection, in current units.
<u>FLX</u>	1	Specifies whether torsional-flexural buckling restraint is provided or is not necessary for the member. See AISI C4.1 0 = Section subject to torsional-flexural buckling and restraint not provided 1 = Restraint provided or unnecessary
<u>FU</u>	58 ksi	Ultimate tensile strength of steel in current units.
<u>FYLD</u>	36 ksi	Yield strength of steel in current units.
<u>KT</u>	1.0	Effective length factor for torsional buckling used to compute the KL/r ratio for determining the capacity in axial compression. Values range from 0.01 (for a column completely restrained against torsional buckling) to any large value.

Parameter Name	Default Value	Description
<u>KY</u>	1.0	Effective length factor for overall column buckling about the local y-axis; used to compute the KL/r ratio for determining the capacity in axial compression. Values can range from 0.01 (for a column completely restrained against buckling) to any large value.
<u>KZ</u>	1.0	Effective length factor for overall column buckling about the local z-axis; used to compute the KL/r ratio for determining the capacity in axial compression. Values can range from 0.01 (for a column completely restrained against buckling) to any large value.
<u>LT</u>	Member Length	Unbraced length used in computing KL/r for twisting, in current units of length.
<u>LY</u>	Member Length	Length used to calculate slenderness ratio for buckling about the local y-axis.
<u>LZ</u>	Member Length	Same as LY, but in the local z-axis.
<u>NSF</u>	1.0	Net section factor for tension members. See AISI C2.
<u>STIFF</u>	Member Length	Spacing in the longitudinal direction of shear stiffeners for reinforced web elements, in current units of length. See AISI C3.2.

Section 2 American Steel Design

2.6 Design per American Cold Formed Steel Code

Parameter Name	Default Value	Description
<u>TRACK</u>	0	<p>Used to control the level of detail in which the design output is reported in the output file.</p> <p>0 = Prints only the member number, section name, ration, and PASS/FAIL status.</p> <p>1 = Prints the design summary in addition to that printed by TRACK 0</p> <p>2 = Prints member and material properties in addition to that printed by TRACK 1</p>
<u>TSA</u>	1	<p>Specifies whether the bearing and intermediate transverse stiffeners are present. If set to 1, the program uses more liberal set of interaction equations found in AISI C3.3.2.</p> <p>0 = Beams with unreinforced webs</p> <p>1 = Beams with transverse web stiffeners</p>

Section 3

American Concrete Design

3.1 Design Operations	217
3.2 Section Types for Concrete Design	218
3.3 Member Dimensions	219
3.4 Design Parameters	220
3.5 Slenderness Effects and Analysis Consideration	224
3.6 Beam Design	225
3.7 Column Design	231
3.8 Designing elements, shear walls, slabs	236

3.1 Design Operations

STAAD has the capabilities for performing concrete design. It will calculate the reinforcement needed for the specified concrete section. All the concrete design calculations are based on the current edition of ACI 318 (unless a previous version is specified).

Section 3 American Concrete Design

3.2 Section Types for Concrete Design

Four versions of the code are currently implemented: the 1999, 2002, 2005, and 2008 editions.

To access the 1999 edition, specify the commands:

```
START CONCRETE DESIGN  
CODE ACI 1999
```

To access the 2002 edition, specify the commands:

```
START CONCRETE DESIGN  
CODE ACI 2002
```

To access the 2005 edition, specify the commands:

```
START CONCRETE DESIGN  
CODE ACI 2005
```

To access the 2008 edition, specify the commands:

```
START CONCRETE DESIGN  
CODE ACI
```

or

```
CODE ACI 2008
```

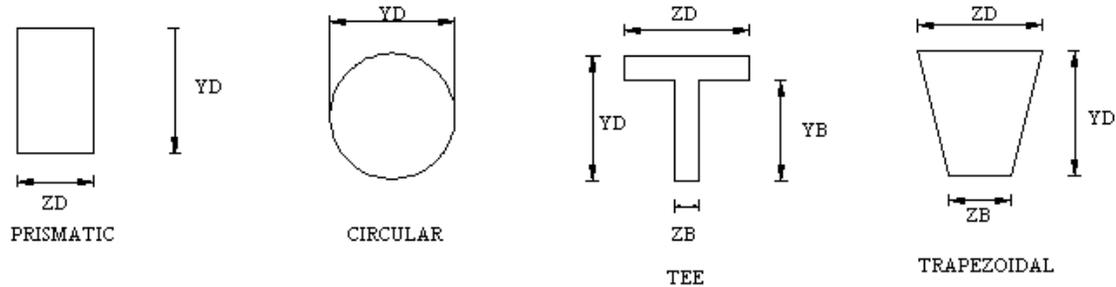
Note: Only the latest version of the code, ACI 318-08, is available for selection through the User Interface. For design per older editions, you must specify the code year manually in the STAAD input file through the STAAD Editor.

3.2 Section Types for Concrete Design

The following types of cross sections can be defined for concrete design.

- For Beams Prismatic (Rectangular & Square), Trapezoidal and T-shapes
- For Columns Prismatic (Rectangular, Square and Circular)
- For Slabs Finite element with a specified thickness.
- Walls/Plates

Figure 3-1: Section types for concrete design



3.3 Member Dimensions

Concrete members which will be designed by the program must have certain section properties input under the **MEMBER PROPERTY** command. The following example shows the required input:

```

UNIT INCH
MEMBER PROPERTY
1 3 TO 7 9 PRISM YD 18. ZD 12. IZ 2916 IY 1296
11 13 PR YD 12.
14 TO 16 PRIS YD 24. ZD 48. YB 18. ZB 12.
17 TO 19 PR YD 24. ZD 18. ZB 12.

```

In the above input, the first set of members are rectangular (18 inch depth and 12 inch width) and the second set of members, with only depth and no width provided, will be assumed to be circular with 12 inch diameter. Note that no area (AX) is provided for these members. For concrete design, this property must not be provided. If shear areas and moments of inertias are not provided, the program calculates these values from YD and ZD. Notice that in the above example the IZ and IY values provided are actually 50% of the values calculated using YD and ZD. This is a conventional practice which takes into consideration revised section parameters due to cracking of section.

The third and the fourth set of members in the above example represent a T-shape and a TRAPEZOIDAL shape respectively. Depending on the properties (YD, ZD, YB, ZB, etc.) provided, the program will determine whether the section is rectangular, trapezoidal or T-shaped and the BEAM design will be done accordingly.

3.4 Design Parameters

The program contains a number of parameters which are needed to perform design by the ACI code. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed. Table 3.1 is a complete list of the available parameters and their default values.

Section 5.53.2 of this manual describes the commands required to provide these parameters in the input file. For example, the values of **SFACE** and **EFACE** (parameters that are used in shear design), the distances of the face of supports from the end nodes of a beam, are assigned values of zero by default but may be changed depending on the actual situation. Similarly, beams and columns are designed for moments directly obtained from the analyses without any magnification. The factor **MMAG** may be used for magnification of column moments. For beams, the user may generate load cases which contain loads magnified by the appropriate load factors.

Table 3-1: ACI 318 Design Parameters

Parameter Name	Default Value	Description
<u>CLB</u>	1.5 in. for beams* 0.75 in. for plate elements*	Clear cover for bottom reinforcement.
<u>CLS</u>	1.5 in.*	Clear cover for side reinforcement. See 'Note a' below.
<u>CLT</u>	1.5 in. for beams* 0.75 in. for plate elements*	Clear cover for top reinforcement.
<u>DEPTH</u>	YD*	Depth of concrete member. This value defaults to YD as provided under MEMBER PROPERTIES .

Parameter Name	Default Value	Description
<u>EFACE</u>	0.0*	Face of support location at end of beam. If specified, the shear force at end is computed at a distance of EFACE + d from the end joint of the member. Note: See "Design for Shear" on page 225 for additional information.
<u>FC</u>	4,000 psi*	Compressive strength of concrete.
<u>FYMAIN</u>	60,000 psi*	Yield stress for main reinforcing steel.
<u>FYSEC</u>	60,000 psi*	Yield stress for secondary steel.
<u>LWF</u>	1.0	Modification factor, λ , for lightweight concrete as specified in ACI 318-08, cl. 8.6.1. Valid entries are between 0.75 and 1.0, inclusive. Used as a reduction factor for the mechanical properties of lightweight concrete.
<u>MAXMAIN</u>	#18 bar	Maximum main reinforcement bar size. Note: See 'Note b' below.
<u>MINMAIN</u>	#4 bar	Minimum main reinforcement bar size (Number 4 - 18) Note: See 'Note b' below.

Section 3 American Concrete Design

3.4 Design Parameters

Parameter Name	Default Value	Description
<u>MINSEC</u>	#4 bar	Minimum secondary reinforcement bar size (Number 4 - 18) Note: See 'Note b' below.
<u>MMAG</u>	1.0	For columns only. Column design moments are magnified by this factor.
<u>NSECTION</u>	12	Number of equally spaced sections to be considered in finding critical moments for beam design. Note: See 'Note c' below.
<u>REINF</u>	0.0	0.0 = Tied column 1.0 = Sprial column
<u>RHOMIN</u>	0.01	Minimum reinforcement required in a concrete column. Enter a value between 0.0 and 0.08, where 0.08 = 8% reinforcement; the maximum allowed by the ACI code.
<u>SFACE</u>	0.0*	Face of support location at start of beam. If specified, the shear force at start is computed at a distance of SFACE + d from the start joint of the member. Note: See "Design for Shear" on page 225 for additional information.

Parameter Name	Default Value	Description
<u>TRACK</u>	0.0	<p>Beam Design:</p> <p>0.0 = Critical moment will not be printed out with beam design report.</p> <p>1.0 = Critical moment will be printed out with beam design report</p> <p>2.0 = Print out required steel areas for all intermediate sections specified by NSECTION.</p> <p>Column Design:</p> <p>0.0 = Prints out detailed design reports</p> <p>1.0 = Prints out column interaction analysis results in addition to TRACK 0.0 output.</p> <p>2.0 = Prints out a schematic interaction diagram and intermediate interaction values in addition to TRACK 2.0 results.</p>
<u>WIDTH</u>	ZD*	Width of concrete member. This value defaults to ZD as provided under MEMBER PROPERTIES .

Note: * These values must be provided in the current unit system being used.

Section 3 American Concrete Design

3.5 Slenderness Effects and Analysis Consideration

Notes

- a. The value used when specifying the **CLS** parameter for column design is taken to be the clear cover for the longitudinal bars in a column. It is not taken as the clear cover for the tie bars. Therefore, the distance from the edge of the column to the centerline of the first row of longitudinal bars is **CLS** plus half the diameter of the main bar.
- b. When using metric units for ACI design, provide values for these parameters in actual 'mm' units instead of the bar number. The following metric bar sizes are available: 6 mm, 8 mm, 10 mm, 12 mm, 16 mm, 20 mm, 25 mm, 32 mm, 40 mm, 50 mm and 60 mm.
- c. **NSECTION** should have no member list since it applies to all members. The minimum value allowed is 12, the maximum is 20. If more than one **NSECTION** entered, then highest value is used.

3.5 Slenderness Effects and Analysis Consideration

Slenderness effects are extremely important in designing compression members. The ACI-318 code specifies two options by which the slenderness effect can be accommodated (Section 10.10 & 10.11 ACI-318). One option is to perform an exact analysis which will take into account the influence of axial loads and variable moment of inertia on member stiffness and fixed-end moments, the effect of deflections on moments and forces, and the effect of the duration of loads. Another option is to approximately magnify design moments.

STAAD uses both these options. To perform the first type of analysis, use the command **PDELTA ANALYSIS** instead of **PERFORM ANALYSIS**. This analysis method will accommodate the requirements as specified in Section 10.10 of the ACI-318 Code, except for the effects of the duration of the loads. It is felt that this effect may be safely ignored because experts believe that the effects of the duration of loads are negligible in a normal structural configuration. If it is desired, STAAD can also accommodate any arbitrary moment magnification factor (second option) as an input, in order to provide some safety due to the effects of the duration of loads.

Although ignoring load duration effects is somewhat of an approximation, it must be realized that the approximate evaluation of slenderness effects is also an approximate method. In this method, moment-magnification is based on empirical formula and assumptions on sidesway.

Considering all this information, it is our belief, that a **PDELTA ANALYSIS**, as performed by STAAD, is most appropriate for the design of concrete members.

However, you should note that to take advantage of this analysis, all combinations of loadings must be provided as primary load cases and not as load combinations. This is due to the fact that load combinations are just algebraic combinations of forces and moments, whereas a primary load case is revised during the PDelta analysis based on the deflections. Also note that the proper factored loads (e.g., 1.4 for DL etc.) should be provided by the user. STAAD does not factor the loads automatically.

3.6 Beam Design

Beams are designed for flexure, shear and torsion. For all these forces, all active beam loadings are prescanned to locate the possible critical sections. The total number of sections considered is 12 (twelve) unless this number is redefined with an **NSECTION** parameter. All of these equally spaced sections are scanned to determine moment and shear envelopes.

3.6.1 Design for Flexure

Reinforcement for positive and negative moments are calculated on the basis of the section properties provided by the user. If the section dimensions are inadequate to carry the applied load, that is if the required reinforcement is greater than the maximum allowable for the cross section, the program reports that beam fails in maximum reinforcement. Effective depth is chosen as Total depth - (Clear cover + diameter of stirrup + half the dia. of main reinforcement), and a trial value is obtained by adopting proper bar sizes for the stirrups and main reinforcements. The relevant clauses in Sections 10.2 to 10.6 of ACI 318 are utilized to obtain the actual amount of steel required as well as the maximum allowable and minimum required steel. These values are reported as ROW, ROWMX, and ROWMN in the output and can be printed using the parameter **TRACK 1.0** (see Table 3.1). In addition, the maximum, minimum and actual bar spacing are also printed.

It is important to note that beams are designed for flexural moment MZ only. The moment MY is not considered in the flexural design.

3.6.2 Design for Shear

Shear reinforcement is calculated to resist both shear forces and torsional moments. Shear forces are calculated at a distance (d+SFACE) and (d+EFACE) away from the end nodes of the beam. Parameters **SFACE** and **EFACE** have default values of zero unless provided under parameters (see Table 3.1). The value of the effective depth, d, used for this purpose is the updated value and accounts for the

Section 3 American Concrete Design

3.6 Beam Design

actual center of geometry of the main reinforcement calculated under flexural design.

Note: If a beam cross section is very deep but the span length is relatively short, an error may result from the start of the shear design occurring beyond the beam's midpoint. In these cases, a negative value of **SFACE** and **EFACE** can be used to start the shear design at reasonable distance. For example, a negative value of *d* may be used to start the design at the start and end nodes.

Clauses 11.1 through 11.6 of ACI 318 are used to calculate the reinforcement for shear forces and torsional moments. Based on the total stirrup reinforcement required, the size of bars, the spacing, the number of bars and the distance over which they are provided are calculated. Stirrups are always assumed to be two-legged.

The reduction factor for lightweight concrete, λ , introduced in ACI 318-08 can be manually specified through the **LWF** parameter. This value is used to reduce the shear strength provided by concrete, V_c , in equations 11-3, 11-4, and 11-5.

3.6.3 Design for Anchorage

In the output for flexural design, the anchorage details are also provided. At any particular level, the **START** and **END** coordinates of the layout of the main reinforcement is described along with the information whether anchorage in the form of a hook or continuation is required or not at these **START** and **END** points. Note that the coordinates of these **START** and **END** points are obtained after taking into account the anchorage requirements. Anchorage length is calculated on the basis of the Clauses described in Chapter 12 of ACI 318.

Example

Beam design per the ACI 318-2002 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 2002
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
```

```
DESIGN BEAM 1 7 10
END CONCRETE DESIGN
```

Note: The command **CODE ACI** can also be used to initiate the ACI 318-2002 design code.

Example

Beam design per the ACI 318-1999 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 1999
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN BEAM 1 7 10
END CONCRETE DESIGN
```

3.6.4 Description of Output for Beam Design

The following is a sample output of an actual reinforcement pattern developed by the program. The following annotations apply to the output:

LEVEL

Serial number of bar level which may contain one or more bar group

HEIGHT

Height of bar level from the bottom of beam

BAR INFO

Reinforcement bar information specifying number of bars and bar size

FROM

Distance from the start of the beam to the start of the reinforcement bar

TO

Distance from the start of the beam to the end of the reinforcement bar

Section 3 American Concrete Design

3.6 Beam Design

ANCHOR (STA/END)

States whether anchorage, either a hook or continuation, is needed at start (STA) or at the end

ROW

Actually required flexural reinforcement (A_s/bd) where b = width of cross section (ZD for rectangular and square section) and d = effective depth of cross section (YD - distance from extreme tension fiber to the c.g. of main reinforcement).

ROWMN

Minimum required flexural reinforcement (A_{min}/bd)

ROWMX

Maximum allowable flexural reinforcement (A_{max}/bd)

SPACING

Distance between centers of adjacent bars of main reinforcement

V_u

Factored shear force at section

V_c

Nominal shear strength provided by concrete

V_s

Nominal shear strength provided by shear reinforcement

T_u

Factored torsional moment at section

T_c

Nominal torsional moment strength provided by concrete

T_s

Nominal torsional moment strength provided by torsion reinforcement

Figure 3-2: Nomenclature used in output

Section 3 American Concrete Design

3.6 Beam Design

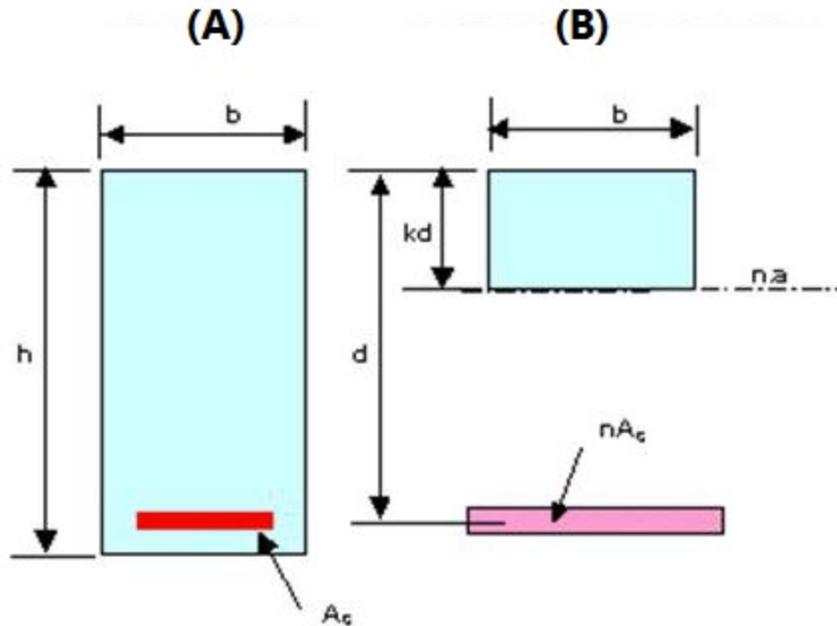
0000 4#11	0000 4#11	0000 4#11	0000 4#11	0000 3#6
	8#5 oooooooo	8#5 oooooooo	8#5 oooooooo	8#5 oooooooo

3.6.5 Cracked Moment of Inertia - ACI Beam Design

When beam design is done per ACI 318, STAAD will report the moment of inertia of the cracked section at the location where the design is performed. The cracked section properties are calculated in accordance with the equations shown below.

Rectangular Sections

Figure 3-3: Gross section (A) and cracked transform section (B) for rectangular shapes



Without compression steel

$$n = E_s/E_c$$

$$B = b/(nA_s)$$

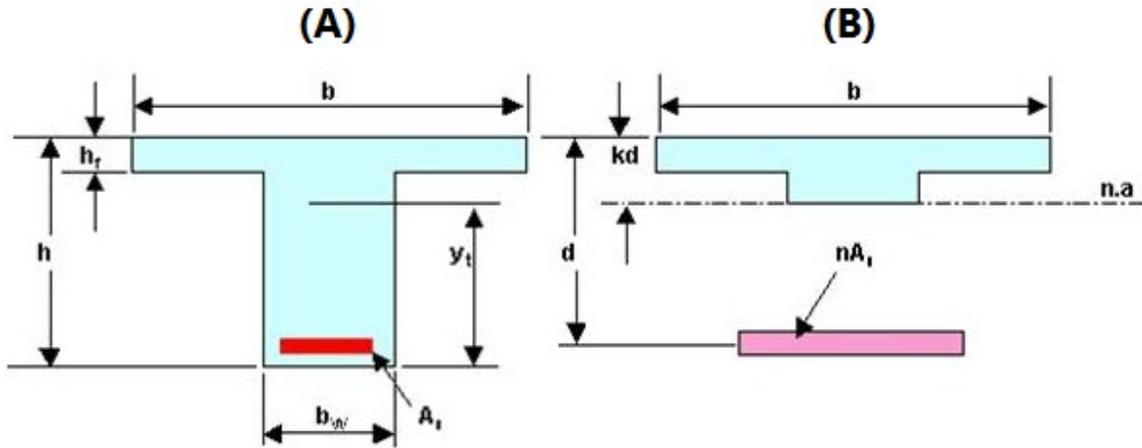
$$I_g = b \cdot h^3/12$$

$$kd = [(2d \cdot B + 1)^{1/2} - 1]/B$$

$$I_{cr} = b(kd)^3/3 + nA_s(d - kd)^2$$

Tee Shaped Sections

Figure 3-4: Gross and cracked transform sections for tee shapes without compression steel



Without compression steel

$$C = b_w/nA_s$$

$$f = h_f(b - b_w)/(nA_s)$$

$$y_t = h - 1/2[(b - b_w) \cdot h_f^2 + b_w h^2]/[(b - b_w) \cdot h_f + b_w h]$$

$$kd = \{[C \cdot (2d + h_f \cdot f) + (1 + f)^2]^{1/2} - (1 + f)\}/C$$

$$I_{cr} = (b - b_w) \cdot h_f^3/12 + b_w(kd)^3/3 + (b - b_w) \cdot h_f \cdot (kd - h_f/2)^2 + nA_s \cdot (d - kd)^2$$

See "Description of Output for Beam Design" on page 227 for an example of output including the calculated cracked moment of inertia.

3.7 Column Design

Columns design in STAAD per the ACI code is performed for axial force and uniaxial as well as biaxial moments. All active loadings are checked to compute reinforcement. The loading which produces the largest amount of reinforcement is called the critical load. Column design is done for square, rectangular and circular sections. For rectangular and circular sections, reinforcement is always assumed to be equally distributed on all faces. This means that the total number of bars for these sections will always be a multiple of four (4). If the MMAG parameter is specified, the column moments are multiplied by the MMAG value to arrive at the ultimate moments on the column. Since the ACI code no longer requires any minimum eccentricity conditions to be satisfied, such checks are not made.

Method used

Bresler Load Contour Method

Known Values

P_u , M_{uy} , M_{uz} , B , D , Clear cover, F_c , F_y

Ultimate Strain for concrete : 0.003

Steps involved

1. Assume some reinforcement. Minimum reinforcement (1%) is a good amount to start with.
2. Find an approximate arrangement of bars for the assumed reinforcement.
3. Calculate $P_{NMAX} = 0.85 P_o$, where P_o is the maximum axial load capacity of the section. Ensure that the actual nominal load on the column does not exceed P_{NMAX} . If P_{NMAX} is less than P_u/ϕ , (ϕ is the strength reduction factor) increase the reinforcement and repeat steps 2 and 3. If the reinforcement exceeds 8%, the column cannot be designed with its current dimensions.
4. For the assumed reinforcement, bar arrangement and axial load, find the uniaxial moment capacities of the column for the Y and the Z axes, independently. These values are referred to as M_{YCAP} and M_{ZCAP} respectively.
5. Solve the interaction equation:

$$\left(\frac{M_{ny}}{M_{ycap}} \right)^a + \left(\frac{M_{nz}}{M_{zcap}} \right)^a \leq 1.0$$

Where $a = 1.24$

If the column is subjected to a uniaxial moment, a is chosen as 1.0

6. If the Interaction equation is satisfied, find an arrangement with available bar sizes, find the uniaxial capacities and solve the interaction equation again. If the equation is satisfied now, the reinforcement details are written to the output file.
7. If the interaction equation is not satisfied, the assumed reinforcement is increased (ensuring that it is under 8%) and steps 2 to 6 are repeated.
8. The maximum spacing of reinforcement closest to the tension force, for purposes of crack control, is given by

$$s = 15 \left(40,000 \frac{40,000}{f_s} \right) - 2.5cc \leq 12 \left(\frac{40,000}{f_s} \right)$$

with f_s in psi and is permitted to be taken equal to $(2/3) f_y$, rather than 60 percent of f_y , as in ACI 318-02.

9. Section 10.9.3 has been modified to permit the use of spiral reinforcement with specified yield strength of up to 100,000 psi. For spirals with f_{yt} greater than 60,000 psi, only mechanical or welded splices may be used.

Column Interaction

The column interaction values may be obtained by using the design parameter TRACK 1.0 or TRACK 2.0 for the column member. If a value of 2.0 is used for the TRACK parameter, 12 different Pn-Mn pairs, each representing a different point on the Pn-Mn curve are printed. Each of these points represents one of the several Pn-Mn combinations that this column is capable of carrying about the given axis, for the actual reinforcement that the column has been designed for. In the case of circular columns, the values are for any of the radial axes. The values printed for the TRACK 1.0 output are:

Po = Maximum purely axial load carrying capacity of the column (zero moment).

Pnmax = Maximum allowable axial load on the column (Section 10.3.5 of ACI 318).

P-bal = Axial load capacity at balanced strain condition.

M-bal = Uniaxial moment capacity at balanced strain condition.

e-bal = M-bal / P-bal = Eccentricity at balanced strain condition.

Mo = Moment capacity at zero axial load.

P-tens = Maximum permissible tensile load on the column.

Des. Pn = Pu/PHI where PHI is the Strength Reduction Factor and Pu is the axial load for the critical load case.

Des. Mn = Mu*MMAG/PHI where PHI is the Strength Reduction Factor and Mu is the bending moment for the appropriate axis for the critical load case. For circular columns,

$$M_u = \sqrt{M_{uy}^2 + M_{uz}^2}$$

$$e/h = (M_n/P_n)/h$$

Section 3 American Concrete Design

3.7 Column Design

Where:

h is the length of the column.

Example

Column design per the ACI 318-2005 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 2005
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN COLUMN 23 25
END CONCRETE DESIGN
```

Example

Column design per the ACI 318-2002 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 2002
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN COLUMN 23 25
END CONCRETE DESIGN
```

Example

Column design per the ACI 318-1999 code

```
UNIT KIP INCH
START CONCRETE DESIGN
```

```

CODE ACI 1999
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN COLUMN 23 25
END CONCRETE DESIGN
    
```

Column Design Output

The samples illustrate different levels of the column design output. The following output is generated without any **TRACK** definition (i.e., using the default of **TRACK 0.0**):

```

=====
      COLUMN NO.      5  DESIGN PER ACI 318-05 - AXIAL + BENDING
FY - 60000 FC - 4000 PSI, SQRE SIZE - 12.00 X 12.00 INCHES, TIED
      AREA OF STEEL REQUIRED = 7.589 SQ. IN.
BAR CONFIGURATION      REINF PCT.  LOAD  LOCATION  PHI
-----
      8 - NUMBER 9      5.556      2      STA      0.650
(PROVIDE EQUAL NUMBER OF BARS ON EACH FACE)
TIE BAR NUMBER 4 SPACING 8.00 IN
    
```

TRACK 1.0 generates the following additional output:

```

      COLUMN INTERACTION: MOMENT ABOUT Z -AXIS (KIP-FT)
-----
      P0      Pn max      P-bal.      M-bal.      e-bal.(inch)
942.40      753.92      179.59      170.75      11.41
      M0      P-tens.      Des.Pn      Des.Mn      e/h
148.52      -480.00      350.15      10.47      0.00249
-----
      COLUMN INTERACTION: MOMENT ABOUT Y -AXIS (KIP-FT)
-----
      P0      Pn max      P-bal.      M-bal.      e-bal.(inch)
942.40      753.92      179.59      170.75      11.41
      M0      P-tens.      Des.Pn      Des.Mn      e/h
148.52      -480.00      350.15      136.51      0.03249
-----
    
```

TRACK 2.0 generates the following output in addition to the above examples:

```

      Pn      Mn      Pn      Mn      (@ Z )
      |      |      |      |
      P0 | *      695.93      77.23      347.96      148.53
      | *      637.93      93.16      289.97      157.71
      | *      579.94      107.06      231.98      164.41
Pn,max| *      521.94      118.23      173.98      170.18
      | *      463.95      129.01      115.99      163.66
      Pn | *      405.96      139.03      57.99      156.37
NOMINAL| *      Pn      Mn      Pn      Mn      (@ Y )
AXIAL | *      695.93      77.23      347.96      148.53
COMPRESSION| *      637.93      93.16      289.97      157.71
      Pb|-----*Mb      579.94      107.06      231.98      164.41
      | *      521.94      118.23      173.98      170.18
      | *      463.95      129.01      115.99      163.66
-----| * M0      Mn,      405.96      139.03      57.99      156.37
      | * BENDING
    
```

Section 3 American Concrete Design

3.8 Designing elements, shear walls, slabs

P-tens|* MOMENT
|

3.8 Designing elements, shear walls, slabs

STAAD currently provides facilities for designing 3 types of entities associated with surface type of structures.

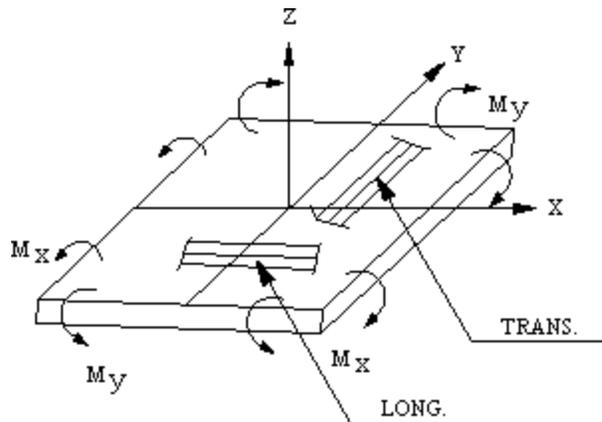
- Individual plate elements – these are designed from the standpoint that one element is independent of the next element. See Section 3.8.1 for details.
- Shear Walls – Structural components modeled using the SURFACE INCIDENCE command can be designed as shear walls. See Section 3.8.2 for details.

3.8.1 Element Design

Element design will be performed only for the moments M_x and M_y at the center of the element. Design will not be performed for S_x , S_y , S_{xy} , S_{qx} , S_{qy} or M_{xy} . Also, design is not performed at any other point on the surface of the element.

A typical example of element design output is shown in Table 3.4. The reinforcement required to resist M_x moment is denoted as longitudinal reinforcement and the reinforcement required to resist M_y moment is denoted as transverse reinforcement (Figure 3.1). The parameters $FYMAIN$, FC , and $CLEAR$ listed in Table 3.1 are relevant to slab design. Other parameters mentioned in Table 3.1 are not applicable to slab design. Please note that the default value of clear cover - parameters CLT and CLB - for plate elements is 0.75 inches, as shown in Table 3.1.

Figure 3-5: Sign convention of loaded plate element



Example Element Design Output

```
ELEMENT DESIGN SUMMARY
-----
ELEMENT      LONG. REINF      MOM-X /LOAD      TRANS. REINF      MOM-Y /LOAD
              (SQ.IN/FT)      (K-FT/FT)        (SQ.IN/FT)        (K-FT/FT)
FY:          60.000 KSI FC:          4.000 KSI COVER (TOP):      0.750 IN
COVER (BOTTOM): 0.750 IN TH:          6.000 IN
47 TOP : Longitudinal direction - Only minimum steel required.
47 TOP : Transverse direction  - Only minimum steel required.
47 TOP :          0.130          0.00 / 0          0.130          0.00 / 0
BOTT:          0.562          11.60 / 1          0.851          14.83 / 1
```

Example

Element design per the ACI 318-2002 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 2002
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN ELEMENT 43
END CONCRETE DESIGN
```

Example

Element design per the ACI 318-1999 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 1999
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN ELEMENT 43
END CONCRETE DESIGN
```

Section 3 American Concrete Design

3.8 Designing elements, shear walls, slabs

3.8.2 Shear Wall Design

Design of shear walls in accordance with ACI 318-02 has been implemented. Shear walls have to be modeled using the Surface element. The program implements provisions of chapters 10, 11 and 14 of ACI 318-02.

The attributes associated with the surface element, and the sections of this manual where the information may be obtained, are listed below:

Attributes	Related Sections
Surfaces Incidences	5.13.3
Openings in surface	5.13.3
Local Coordinates system for surfaces	1.6.3
Specifying sections for stress/force output	5.13.3
Property for surfaces	5.21.2
Material constants	5.26.3
Surface loading	5.32.3.4
Stress/Force output printing	5.42
Shear Wall Design	3.8.2, 5.55

Description

The program implements the provisions of ACI 318-02, except Chapter 21, for the design of shear walls. It performs in-plane shear, compression, as well as in-plane and out-of-plane bending design of reinforcing. The shear wall is modeled by a single or a combination of Surface elements. The use of the Surface element enables the designer to treat the entire wall as one entity. It greatly simplifies the modeling of the wall and adds clarity to the analysis and design output. The

results are presented in the context of the entire wall rather than individual finite elements thereby allowing users to quickly locate required information.

The definition of the shear wall starts with specification of the Surface element (s). The boundary of the wall should include all corner joints as well as any additional joints required for connections with other elements of the building, such as beams. All geometry and material properties must be specified for surface elements separately from specifications for other element types.

The wall may be loaded indirectly through members/elements attached to it, or directly by application of nodal or uniformly distributed loads. All wall elements must be of constant thickness.

The program reports shear wall design results for each load case/combination for a user specified number of sections given by the SURFACE DIVISION (default value is 10) command. The wall is designed at these horizontal sections. The output includes the required horizontal and vertical distributed reinforcing, the concentrated (in-plane bending) reinforcing, and the links required to resist out-of-plane shear.

General Format

START SHEARWALL DESIGN

CODE ACI

FYMAIN f1

FC f2

HMIN f3

HMAX f4

VMIN f5

VMAX f6

EMIN f7

EMAX f8

LMIN f9

LMAX f10

CLEAR f11

TWOLAYERED f12

KSLENDER f13

DESIGN SHEARWALL LIST shearwall-list

Section 3 American Concrete Design

3.8 Designing elements, shear walls, slabs

END

The following table explains parameters used in the shear wall design command block above. All reinforcing bar sizes are English designation (#).

Table 3-2: Shear Wall Design Parameters

Parameter Name	Default Value	Description
<u>FYMAIN</u>	60.0 ksi	Yield strength of steel, in current units.
<u>FC</u>	4.0 ksi	Compressive strength of concrete, in current units.
<u>HMIN</u>	3	Minimum size of horizontal reinforcing bars (range 3 -18).
<u>HMAX</u>	18	Maximum size of horizontal reinforcing bars (size 3 - 18).
<u>VMIN</u>	3	Minimum size of vertical reinforcing bars (range 3 - 18).
<u>VMAX</u>	18	Maximum size of vertical reinforcing bars (range 3 - 18).
<u>EMIN</u>	3	Minimum size of vertical reinforcing bars located in edge zones (range 3 - 18).
<u>EMAX</u>	18	Maximum size of vertical reinforcing bars located in edge zones (range 3 - 18).
<u>LMIN</u>	3	Minimum size of links (range 3 - 18).
<u>LMAX</u>	18	Maximum size of links (range 3 - 18).
<u>CLEAR</u>	3.0 in.	Clear concrete cover, in current units.
<u>TWOLAY-ERED</u>	0	Reinforcement placement mode: 0. single layer, each direction 1. two layers, each direction

Parameter Name	Default Value	Description
<u>K</u> SLENDER	1.5	Slenderness factor for finding effective height.

Example

```

...
SET DIVISION 12
SURFACE INCIDENCES
2 5 37 34 SUR 1
19 16 65 68 SUR 2
11 15 186 165 SUR 3
10 6 138 159 SUR 4
...
SURFACE PROPERTY
1 TO 4 THI 18
SUPPORTS
1 7 14 20 PINNED
2 TO 5 GEN PIN
6 TO 10 GEN PIN
11 TO 15 GEN PIN
19 TO 16 GEN PIN
...
SURFACE CONSTANTS
E 3150
POISSON 0.17
DENSITY 8.68E-005
ALPHA 5.5E-006
...
START SHEARWALL DES

```

Section 3 American Concrete Design

3.8 Designing elements, shear walls, slabs

```
CODE ACI
FC 4
FYMAIN 60
TWO 1
VMIN 5
HMIN 5
EMIN 8
DESIGN SHEA LIST 1 TO 4
END
```

Notes regarding the above example:

1. Command **SET DIVISION 12** indicates that the surface boundary node-to-node segments will be subdivided into 12 fragments prior to finite element mesh generation.
2. Four surfaces are defined by the **SURFACE INCIDENCES** command.
3. The **SUPPORTS** command includes the support generation routine. For instance, the line **2 TO 5 GEN PIN** assigns pinned supports to all nodes between nodes 2 and 5. As the node-to-node distances were previously subdivided by the **SET DIVISION 12** command, there will be an additional 11 nodes between nodes 2 and 5. As a result, all 13 nodes will be assigned pinned supports. Please note that the additional 11 nodes are not individually accessible to the user. They are created by the program to enable the finite element mesh generation and to allow application of boundary constraints.
4. Surface thickness and material constants are specified by the **SURFACE PROPERTY** and **SURFACE CONSTANTS**, respectively.
5. The shear wall design commands are listed between lines **START SHEARWALL DES** and **END**. The **CODE** command selects the design code that will be the basis for the design. The **DESIGN SHEARWALL LIST** command is followed by a list of previously defined Surface elements intended as shear walls and/or shear wall components. Refer to the beginning of this section for references to all related commands.

Technical Overview

The program implements provisions of Chapter 14 of ACI-318-02 and relevant provisions from Chapters 10 and 11, as referenced therein, for all active load cases. The wall is designed as an unbraced reinforced wall. The following steps are performed for each of the horizontal sections of the wall set using the SURFACE DIVISION command whose default value is 10.

Design for in-plane shear (denoted using F_{xy} in the shear wall force output) per Section 11.10 of ACI 318

- Extreme compression fiber to centroid of tension (concentrated) reinforcement distance, d , is taken as 0.8 horizontal length of the wall (ACI - 11.10.4),
- Limit on the nominal shear strength, V_n , is calculated (ACI - 11.10.3),
- Nominal shear strength of concrete is computed (11.10.6),
- If the factored shear force does not exceed $\frac{1}{2}$ of the design strength of concrete, the minimum ratios of shear (distributed) reinforcing are reported, in accordance with 14.3.2 and 14.3.3. Otherwise, the reinforcing ratios are established in accordance with 11.10.9.
- If the factored shear force is greater than $\frac{1}{2}$ but does not exceed the design strength of concrete, the ratios reported are the minima calculated in accordance with 11.10.9.2 and 11.10.9.4.
- If the factored shear force exceeds the design strength of concrete, the distributed reinforcing is calculated based on 11.1.1 and 11.10.9.
- Number of distributed reinforcing layers and reinforcing allocation between layers reflect requirements of 14.3.4.
- Rebar spacing is given c/c and meets the requirements of 14.3.5, 11.10.9.3, and 11.10.9.5 of ACI 318, as applicable.

Design for in-plane bending (denoted by M_z in the shear wall force output) per Section 14.4 of ACI 318

- Walls are assumed to be cantilever beams fixed at their base and carrying loads to the foundation.
- Strength reduction factor is established in accordance with Section 9.3.2.
- Minimum reinforcing is calculated in accordance with 10.5.1 or 10.5.3, whichever produces a smaller ratio.

Section 3 American Concrete Design

3.8 Designing elements, shear walls, slabs

- Extreme compression fiber to centroid of tension reinforcement distance, d , is taken as 0.8 horizontal length of the wall (11.10.4 of ACI 318).
- Flexural design of the wall is carried out in accordance with provisions of Chapter 10.
- The flexural (concentrated) reinforcing is located at both ends (edges) of the length of the wall. Rebar layout conforms to the spacing requirements of Section 7.6.

Design for compression and out-of-plane bending (Section 14.8)

- The design is based on the Alternative Design of Slender Walls procedure.
- The procedure requires that the wall panel be designed as simply supported (at top and bottom), axially loaded with out-of-plane uniform lateral load, with maximum moments and deflections occurring at mid-height.
- Minimum distributed reinforcing ratio is controlled by the in-plane shear design calculations.
- The reinforcing amount required by 14.8 is over and above any reinforcing required due to in-plane shear.

3.8.3 Slabs and RC Designer

Contact the Bentley Technical Support Group for further information.

3.8.4 Design of I-shaped beams per ACI-318

I-shaped sections can be designed as beams per the ACI 318 code. The property for these sections must be defined through a user table, I-section, or using the tapered specification. Information on assigning properties in this manner is available in sections 5.19 (I-section type) and 5.20.3 (Tapered I shape) of the Technical Reference manual.

From the standpoint of the analysis – determining member forces, nodal displacements and support reactions – the same set of facilities and rules which are applicable for any normal reinforced concrete frames or other structures can be used when I-sections or tapered concrete members are specified. In other words, there isn't anything unique or special to account for in the analysis model simply because I-shaped concrete beams are part of it.

From the standpoint of design, the following rules are applicable:

1. The member can be designed as a beam using the general principles explained in Chapter 3 of the Technical Reference manual. It currently

cannot be designed as a column. Design as a beam is done for flexure (MZ), shear (FY) and torsion (MX) just like that for rectangular, tee or trapezoidal beams. Axial forces (FX) are used during the capacity computations in shear and torsion. At each section along the length that the member is designed at, the depth at that section location is used for effective depth computation.

2. The program performs the following tests on the section dimensions before starting the design:
 - If the thickness of the web is the same as the width of the top and bottom flanges, the member is designed as a rectangular section.
 - If the thickness of the web is the same as the width of one of the flanges but not the other, the member is designed as a T-section or a rectangular section, depending on which side the compression due to bending is at.
 - If the web thickness does not match the width of either flange, design is done using the rules applicable for T-beams – one flange is in compression, the other in tension, and tensile capacity of concrete on the tensile side of the neutral axis is ignored.
 - The program is also able to design the beam as a doubly reinforced section if it is unable to design it as a single-reinforced section.
3. The parameters for designing these members are as shown in Table 3.1 of this manual. Detailed output on design at individual section locations along the member length may be obtained by setting the **TRACK** parameter to 3.0.

An example for I-beam design is shown below.

```

STAAD PLANE I BEAM CONCRETE DESIGN PER ACI-318
UNIT FEE KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0
MEMBER INCIDENCES
1 1 2
UNIT INCHES KIP
MEMBER PROPERTY
1 TAPERED 18 10 18 15 2.5
CONSTANTS

```

Section 3 American Concrete Design

3.8 Designing elements, shear walls, slabs

```
E 3300 ALL
DENSITY CONCRETE ALL
POISSON CONCRETE ALL
SUPPORTS
1 2 PINNED
UNIT FEET KIP
LOAD 1 DEAD LOAD
MEMBER LOAD
1 UNI GY -5.76
LOAD 2 LIVE LOAD
1 UNI GY -7.04
LOAD COMB 3 ACI 318-02
1 1.4 2 1.7
PERFORM ANALYSIS
LOAD LIST 3
START CONCRETE DESIGN
CODE ACI 2002
UNIT INCHES KIP
MINMAIN 9 ALL
FC 4 ALL
FYMAIN 60 ALL
TRACK 2.0 ALL
DESIGN BEAM ALL
END CONCRETE DESIGN
FINISH
```

Section 4

American Timber Design

4.1 Design Operations	247
4.2 Allowable Stress per AITC Code	250
4.3 Input Specification	252
4.4 Naming Conventions for Sections	254
4.5 Design Parameters	257
4.6 Member Design Capabilities	262
4.7 Orientation of Lamination	263
4.8 Tabulated Results of Member Design	263
4.9 Examples	265

4.1 Design Operations

STAAD.Pro supports timber design per two versions of the AITC code: 1985 and 1994. The implementation of both the codes is explained below.

To access the 1994 edition, specify the commands:

Section 4 American Timber Design

4.1 Design Operations

CODE AITC

or

CODE AITC 1994

To access the 1984 edition, specify the commands:

CODE AITC 1984

or

CODE TIMBER

4.1.1 1994 AITC code implementation

The salient aspects of design in accordance with the 4th edition (1994) of the Timber Construction Manual published by the American Institute of Timber Construction are:

1. Design can be performed for two types of timber sections: dimensional timber sections (i.e., sawn lumber) and glulam sections.
2. The program includes a database of dimensional timber sections with this code.

Implementation of Dimensional Lumber Properties

The database of sawn lumber sections, listed in Table 8.1 of the 1994 AITC Manual, is implemented in the program. Some of the key aspects of this implementation are:

In the property tables in the AITC manual, one will find that, for any particular species of timber, the Modulus of Elasticity (E) and allowable stresses may vary with the cross-section size. For example, a 2x4 Douglas Fir-Larch, Select Structural member has an E of 1900 ksi and an allowable bending stress, F_b , of 1450 psi. A 5x5 Douglas Fir-Larch, Select Structural, Beam or Stringer member has an E of 1600 ksi and an allowable bending stress, F_b , of 1600 psi. And a 5x5 Douglas Fir-Larch, Select Structural, Post or Timbers member has an E of 1600 ksi and an allowable bending stress, F_b , of 1750 psi.

So, in the STAAD timber database for sawn lumber, for each species and grade of timber, the section size, or properties are associated with the Modulus of Elasticity and allowable stresses for the cross-section. When a section is assigned, its E and allowable stresses are automatically fetched along with its properties. The material properties of Southern Pine members were taken from Table 8.4 of the 1994 AITC manual. For all other species with section sizes 2"-4" wide, the material properties have been taken from Table 8.3. For all non-Southern Pine

species with section sizes greater than 5"x5", the material properties are obtained from Table 8.6 of the 1994 AITC Manual.

Please note that not all section sizes listed in Table 8.1 are available in every species. Some sizes are not produced for particular species. For example, the Aspen species only produces sizes from 2"-4" wide. It does not produce sizes 5"x5" and larger. This can be observed by comparing Table 8.3, where Aspen is listed as an available species, to Table 8.6, where Aspen is not listed as an available species. Also note that although 1" wide members are listed in Table 8.1, there are no values available in the species properties tables; Table 8.4, Table 8.5, and Table 8.6. AITC does not allow for the structural design of these small members.

4.1.2 1985 AITC code implementation

STAAD's Timber design module per the 1985 AITC code (*Timber Construction Manual*, 3rd. Edition, 1985) allows design of Glulam timber sections. It also conforms to the National Design Specification for Wood Construction and Supplement (NDS) and building codes like Uniform Building Code (UBC), Basic/National Building Code and Standard Building Code. Some of the main features of the program are:

1. This feature is for Glulam Timber only (design of dimensional lumber are available for 1994 AITC only).
2. Code check and design of members as per TCM - AITC.
3. Design values for Structural Glued Laminated Timber tables are in-built into the program. The program accepts Table no., Combination and Species specifications as inputs (e.g., 1:16F-V3-SP/SP) and reads design values from in-built tables.
4. Incorporates all the following Allowable stress modifiers:
 - i. Duration of Load Factor
 - ii. Size Factor
 - iii. Form Factor
 - iv. Lateral stability of Beams and Columns
 - v. Moisture Content Factor
 - vi. Temperature and Curvature factors.

The allowable stresses for bending, tension, compression, shear and Moduli of elasticities are modified accordingly.

Section 4 American Timber Design

4.2 Allowable Stress per AITC Code

5. Determines slenderness for beams and columns (Short, intermediate and long) and checks for min. eccentricity, lateral stability, buckling, bending and compression, bending and tension and horizontal shear against both axes.
6. The output results show sections provided or chosen, actual and allowable stresses, governing condition and ratios of interaction formulae and the relevant AITC clause nos. etc for each individual member.

4.2 Allowable Stress per AITC Code

Explanation of terms and symbols used in this section

Table 4-1: Timber design nomenclature

Symbols	Description
f_a	Actual compression or tension stress (in PSI). For tension, the axial load is divided by net sectional area (i.e, NSF x X-area).
FA	Allowable design value for compression or tension (in PSI) modified with applicable modifiers or calculated based on slenderness in case of compression.
f_{bz} f_{by}	Actual bending stresses about local Z and Y axis (in PSI).
FBZ FBY	Allowable design values for bending stresses about local Z and Y axis (in PSI) modified by the applicable modifiers.
JZ JY	Modifier for P-DELTA effect about the Z and Y axis respectively as explained in formula 5-18 of TCM.
f_{vz} f_{vy}	Actual horizontal shear stresses.
FVZ FVY	Allowable horizontal shear stresses.

Symbols	Description
VZ VY	Shear in local Z and local Y direction.
ZD YD	Depth of section in local Z and Y axis.
EZ EY	Minimum eccentricity along Z and Y axis.
CFZ CFY	Values of the size factors in the z-axis and y-axis, respectively.
CLZ CLY	Represent the factors of lateral stability for beams about the z-axis and y-axis, respectively.
RATIO	Permissible ratio of stresses. The default value is 1.0.

Combined Bending and Axial Tension

The following interaction formulae are checked :

$$f_a/FA + f_{bz}/(FBZ \times CFZ) + f_{by}/(FBY \times CFY) \leq \text{RATIO}$$

Lateral stability check with Net compressive stress:

$$f_a/FA + f_{bz}/(FBZ \times CLZ) + f_{by}/(FBY \times CLY) \leq \text{RATIO}$$

Combined Bending and Axial Compression

$$f_a/FA + f_{bz}/(FBZ - JZ \times f_a) + f_{by}/(FBY - JY \times f_a) \leq \text{RATIO}$$

Applicability of the size factor:

- a. When $CF < 1.00$,

if $f_a > FBZ \times (1 - CFZ)$, FBZ is not modified with CFZ. if $f_a > FBY \times (1 - CFY)$, FBY is not modified with CFY.

if $f_a < FBZ \times (1 - CFZ)$ FBZ is taken as $FBZ \times CFZ + f_a$ but shall not exceed $FBZ \times CLZ$

Section 4 American Timber Design

4.3 Input Specification

if $f_a < \text{FBY} \times (1 - \text{CFY})$ FBY is taken as $\text{FBY} \times \text{CFY} + f_a$ but shall not exceed $\text{FBY} \times \text{CLY}$

- b. When $\text{CF} \geq 1.00$, the effect of CF and CL are cumulative FBZ is taken as $\text{FBZ} \times \text{CFZ} \times \text{CLZ}$ FBY is taken as $\text{FBY} \times \text{CFY} \times \text{CLY}$

Minimum Eccentricity

The program checks against min. eccentricity in following cases:

- The member is a FRAME member and not a truss member and under compression.
- The value of actual axial compressive stress does not exceed 30% of the allowable compressive stress.
- The actual moments about both axes are less than moments that would be caused due to min. eccentricity. In this approach, the moment due to min. eccentricity is taken as the compressive load times an eccentricity of 1 in. or 0.1 x depth whichever is larger.

In case of min. eccentricity,

$$f_{bz} = f_a \times (6 + 1.5 \times \text{JZ}) / (\text{EZ} / \text{ZD})$$

$$f_{by} = f_a \times (6 + 1.5 \times \text{JY}) / (\text{EY} / \text{YD})$$

the following conditions are checked :

$$f_a / \text{FA} + f_{bz} / (\text{FBZ} - \text{JZ} \times f_a) \leq \text{RATIO}$$

$$f_a / \text{FA} + f_{by} / (\text{FBY} - \text{JY} \times f_a) \leq \text{RATIO}$$

Shear Stresses

Horizontal stresses are calculated and checked against allowable values:

$$f_{vz} = 3 \times \text{VY} / (2 \times \text{Area} \times \text{NSF}) \leq \text{FVZ}$$

$$f_{vy} = 3 \times \text{VZ} / (2 \times \text{Area} \times \text{NSF}) \leq \text{FVY}$$

4.3 Input Specification

A typical set of input commands for STAAD.Pro timber design per AITC 1984 is listed below:

```
UNIT KIP INCH
PARAMETER
```

```

CODE TIMBER
GLULAM 1:16F-V3-DF/DF MEMB 1 TO 14
GLULAM 1:24F-V5-SP/SP MEMB 15 TO 31
GLULAM 20F-V1-DF/WW MEMB 32 TO 41
LAMIN 1.375 LY 168.0 MEMB 5 9 15 TO 31
LZ 176.0 MEMB 1 TO 4 6 7 8 10 TO 14
LUZ 322.6 ALL
LUY 322.6 ALL
WET 1.0 ALL
CDT 1.33
NSF 0.85
BEAM 1.0 ALL
CHECK CODE 1 TO 14
SELECT MEMB 15 TO 31

```

4.3.1 Explanation of Input Commands and Parameters

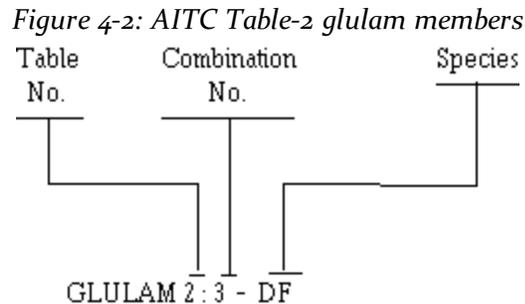
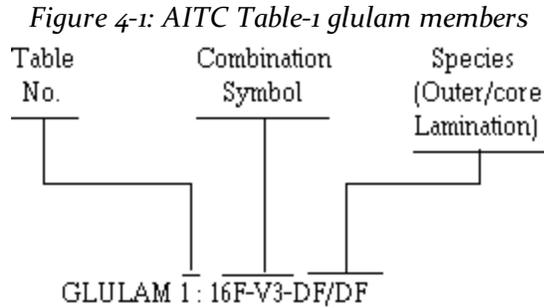
Specify **PARAMETER** and then **CODE TIMBER** to start **TIMBER DESIGN** before specifying the input parameters. The user must provide the timber grade (**GLULAM GRADE**) for each member he intends to design. The parameters can be specified for all or specified list of members. If a parameter is not specified, the default value is assigned to it. See following **INPUT PARAMETERS LIST TABLE** for description and default values of the parameters.

4.3.2 Glulam Grade and Allowable Stresses from Table

The allowable stresses for **GLULAM** members are read in from Table-1 and Table-2 of AITC for design values for Structural Glued Laminated Timber. The structural members are to be specified in the following manner:

Section 4 American Timber Design

4.4 Naming Conventions for Sections



For TABLE-2 members, the applicable stress values are selected based on the depth and the number of laminations.

Note: The lamination thickness (in inches) can be specified. Typical values are 1 3/8 inches or 1 1/2 inches. If not specified, a default value of 1 1/2 inches assumed by the program.

4.4 Naming Conventions for Sections

The following conventions are used to describe timber sections in STAAD.Pro

4.4.1 Dimensional Lumber sections

As can be seen from Tables 8.3 through 8.6 of the AITC 1994 manual, one or more of the following attributes have to be considered while choosing a section :

- Species
- Commercial Grade
- Size classification
- Nominal size of the section
- Grading rules agency

STAAD uses a naming convention that incorporates all of the above. Shown below is the name of a section that has characteristics as shown. It may be found on page 8-637 of the AITC 1994 manual.

Species: Douglas Fir Larch

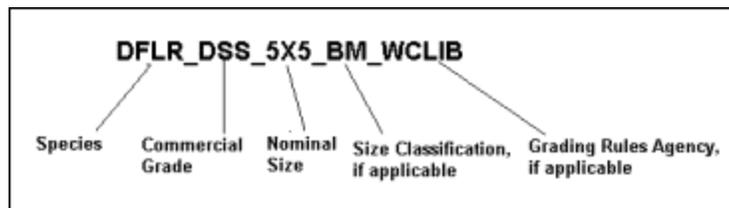
Commercial Grade: Dense Select Structural

Size Classification: Beams

Nominal size: 5" x 5"

Grading Rules Agency: WCLIB

Figure 4-3: Timber naming conventions



Implementation of Glulam Properties

For Glulam sections, each glulam designation has an associated value of Modulus of Elasticity and a set of allowable stresses. However, these values are not dependent on the size of the cross section. For example, a 3-1/8" x 6" 24F-V8 DF/DF beam and a 6-3/4" x 30" 24F-V8 DF/DF beam both have an E of 1,600 ksi and an allowable bending stress in the tension zone, F_{bx} , of 2,400 psi.

Therefore, in STAAD's glulam database, the section sizes are not linked to the glulam type. Users may specify any cross-section size they choose and pick the desired glulam type. The Modulus of Elasticity and allowable stresses associated with that glulam are assigned to the member. The material properties for the Glulam database are taken from Table 1 of AITC 117-93 – Design Standard Specifications for Structural Glued Laminated Timber of Softwood Species. This publication has been reproduced in the AITC 1994 manual starting from page 8-843.

Example for Dimensional Timber

```
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC DFLN_SS_4X4
E 273600
POISSON 0.15
```

Section 4 American Timber Design

4.4 Naming Conventions for Sections

```
DENSITY 0.025
ALPHA 5.5E-006
END DEFINE MATERIAL
MEMBER PROPERTY AITC
3 4 7 8 TABLE ST DFLN_SS_4X4
CONSTANTS
MATERIAL DFLN_SS_4X4 MEMB 3 4 7 8
```

4.4.2 Glulam sections

The STAAD name for glulam sections incorporates

- Combination Symbol
- Species-Outer Laminations/Core Laminations

Shown here is a typical section listed in page 8-854 of the AITC manual.

GLT-24F-V11_DF/DFS

Implementation of Material Constants

As explained in the previous paragraphs, for sawn lumber as well as glulam sections, E is built into the database and gets automatically assigned to the member along with the section dimensions. Density, Poisson's ratio and Alpha (coefficient of thermal expansion) have to be assigned separately. If they are not assigned, the analysis engine will use default values for those.

Example for Glulam Timber:

```
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC GLT-24F-V8_WET_DF/DF
E 191923
POISSON 0.15
DENSITY 0.025
ALPHA 5.5E-006
END DEFINE MATERIAL
MEMBER PROPERTY AITC
8 PRIS YD 1.5 ZD 0.427083
```

CONSTANTS

MATERIAL GLT-24F-V8_WET_DF/DF MEMB 8

4.5 Design Parameters

The timber design parameters for the AITC codes.

4.5.1 AITC 1994 Parameters

Table 4-2: AITC 1994 Timber Design Properties

Parameter Name		Default Value	Description
STAAD	AITC 1994 Code		
<u>CB</u>	C_b	1.0	Bearing Area Factor, Table 4.13
<u>CFB</u>	C_F	1.0	Size Factor for Allowable Bending Stress, see Table 8.3, 8.4, 8.5, 8.6, 8.7
<u>CFC</u>	C_F	1.0	Size Factor for Allowable Compression Parallel to Grain, see Table 8.3, 8.4, 8.5, 8.6, 8.7
<u>CFT</u>	C_F	1.0	Size Factor for Allowable Tension Parallel to Grain, see Table 8.3, 8.4, 8.5, 8.6, 8.7
<u>CFU</u>	C_{fu}	1.0	Flat Use Factor, see Table 4.9
<u>CSS</u>	C_H	1.0	Shear Stress Factor, Section 4.5.14

Section 4 American Timber Design

4.5 Design Parameters

Parameter Name		Default Value	Description
STAAD	AITC 1994 Code		
<u>CMB</u>	C_M	1.0	Wet service Factor for Allowable Bending Stress, see Table 4.8
<u>CMC</u>	C_M	1.0	Wet service Factor for Allowable Compression Parallel to Grain, see Table 4.8
<u>CME</u>	C_M	1.0	Wet service Factor for Modulus of Elasticity, see Table 4.8
<u>CMP</u>	C_M	1.0	Wet service Factor for Allowable Compression Perpendicular to Grain, see Table 4.8
<u>CMT</u>	C_M	1.0	Wet service Factor for Allowable Tension Parallel to Grain, see Table 4.8
<u>CMV</u>	C_M	1.0	Wet service Factor for Allowable Shear Stress Parallel to Grain, see Table 4.8
<u>CR</u>	C_r	1.0	Repetitive Member Factor, see Section 4.5.10
<u>CSF</u>	C_F	1.0	Form Factor, see Section 4.5.12
<u>CTM</u>	C_t	1.0	Temperature Factor, see Table 4.11

Parameter Name		Default Value	Description
STAAD	AITC 1994 Code		
<u>CTT</u>	C_T	1.0	Buckling Stiffness Factor, see Section 4.5.15
<u>KB</u>	K_b	1.0	Buckling Length Coefficient to calculate Effective Length
<u>KBD</u>	K_{bd}	1.0	Buckling Length Coefficient for Depth to calculate Effective Length
<u>KBE</u>	K_{bE}	0.609	Euler Buckling Coefficient for Beams, see Section 5.4.11
<u>KCE</u>	K_{cE}	1.0	Euler Buckling Coefficient for Columns, see Section 5.8.2
<u>KEY</u>	K_{ey}	1.0	Buckling Length Coefficient in Y Direction
<u>KEZ</u>	K_{ez}	1.0	Buckling Length Coefficient in Z Direction
<u>KL</u>	K_1	1.0	Load Condition Coefficient, Table 4.10
<u>LZ</u>	LZ	Member Length	Effective shear length in the z direction for Column Stability Check, $L_e=K_e*L$
<u>LY</u>	LY	Member Length	Effective shear length in the y direction for Column Stability Check, $L_e=K_e*L$
<u>LUZ</u>	LUZ	Member Length	Member length in the z direction for Beam Stability Check, $L_u=K_b*l+K_{bd}*d$

Section 4 American Timber Design

4.5 Design Parameters

Parameter Name		Default Value	Description
STAAD	AITC 1994 Code		
<u>LUY</u>	LUY	Member Length	Member length in the y direction for Beam Stability Check, $L_u = K_b \cdot l + K_{bd} \cdot d$
<u>CDT</u>	CDT	1.0	Load Duration Factor
<u>CCR</u>	CCR	1.0	Curvature factor (Section 4.5.11)
<u>INDEX</u>	INDEX	10	Exponent value in the Volume Factor Equation (Section 4.5.6)
<u>CV</u>	CV	1.0	Volume Factor (Section 4.5.6)
<u>CC</u>	CC	0.8	Variable in Column Stability Factor, C_p (Section 5.8.2, Eqn 5-14)
<u>SRC</u>	SRC	1.0	Slenderness ratio of Compression member
<u>SRT</u>	SRT	1.0	Slenderness ratio of Tension member
<u>RATIO</u>		1.0	Permissible ratio of actual to allowable stress
<u>BEAM</u>		1.0	0 = Design for end forces or locations specified by section command. 1 = Calculate moments at 12 pts along the beam and use the maximum for design.

4.5.2 AITC 1984 Parameters

Table 4-3: AITC 1985 Timber Design Parameters

Parameter Name	Default Value	Description
<u>BEAM</u>	1.0	0.0 = design for end forces or at locations specified by section command. 1.0 = calculate moments at twelfths sections along the beam and use the max. for design.
<u>CCR</u>	1.0	Curvature factor
<u>CDT</u>	1.0	Duration of load factor
<u>CSF</u>	1.0	Form factor
<u>CTM</u>	1.0	Temp. factor
<u>LAMINATION</u>	1.50 inch	Thickness of lamination in inch (1.50 or 1.375)
<u>LUZ</u>	$1.92 * L$	Unsupported effective length for beam in z.
<u>LUY</u>	$1.92 * L$	Unsupported effective length for beam in y.
<u>LY</u>	Member Length	Same as above in y-axis.
<u>LZ</u>	Member Length	Effective length of the column in z-axis.
<u>NSF</u>	1.0	Net section factor for tension members. (both shear and tension stresses are based on sectional area x nsf)
<u>RATIO</u>	1.0	Permissible ratio of actual to allowable stresses.

Section 4 American Timber Design

4.6 Member Design Capabilities

Parameter Name	Default Value	Description
<u>WET</u>	0.0	0.0 - dry condition 1.0 - wet condition wet use factors are in-built

4.6 Member Design Capabilities

STAAD.Pro is capable of performing member design functions for both 1984 and 1994 editions of AITC.

Hint: The User Interface can be used to easily assign design commands to members.

4.6.1 Code Checking

The **CHECK CODE** command enables the user to check the adequacy of the size (**YD X ZD**) provided in the **MEMBER PROPERTIES** for the most critical forces and moments. The program prints whether the member has passed or failed, the critical conditions and the value of the ratio.

4.6.2 Member Selection

Member selection is limited to AITC 1984.

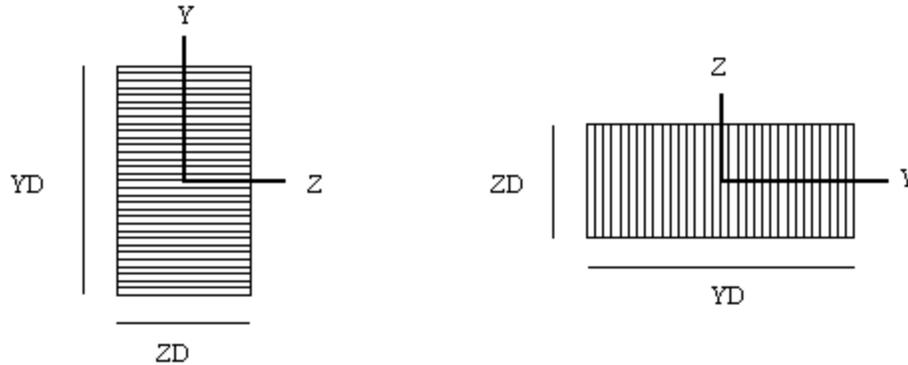
The **SELECT MEMBER** command starts with the min. permissible depth (or min. depth provided thru **DMIN** parameter) and checks the code. If the member fails with this depth, the thickness is increased by one lamination thickness and the code requirements are checked again. The process is continued till the section passes all the code requirements. This ensures the least weight section for the member. If the depth of the section reaches max. allowable or available depth and the member still fails, you can have the following options for redesign:

- a. Change the width or increase the max. allowable depth (**DMAX**)
- b. Change the timber grade
- c. Change the design parameters

4.7 Orientation of Lamination

Laminations are always assumed to lie along the local Z-plane of the member. In the **MEMBER PROPERTIES** section, **YD** always represents the depth of the section across the grain and **ZD** represents the width along the grain. This is in accordance with the sign convention conforming to **SET Y UP** (default).

Figure 4-4: Orientation of lamination



4.8 Tabulated Results of Member Design

For **CODE CHECKING** and/or **MEMBER SELECTION** the output results are printed as shown in the previous section. The items are explained as follows:

- MEMBER refers to the member number for which the design is performed.
- TABLE refers to the size of the **PRISMATIC** section (B X D or ZD X YD).
- RESULT prints whether the member has **PASSed** or **FAILed**.
- CRITICAL COND refers to the **CLAUSE** or **FORMULA NO.** from the **TIMBER CONSTRUCTION MANUAL** (3rd. Edition, AITC-1985) which governed the design. See following table:

Table 4-4: Critical conditions evaluated per AITC code

Critical Condition	Governing Criteria
Clause 5-19	Axial Compression and Bending with MINIMUM ECCENTRICITY.

Section 4 American Timber Design

4.8 Tabulated Results of Member Design

Critical Condition	Governing Criteria
Clause 5-18	Axial Compression and Bending
Clause 5-42	Axial Tension and Bending
Clause 5-24	Horizontal Shear
Clause 5.40	Lateral stability for net compressive stress in case of Tension and Bending.

- e. **RATIO** prints the ratio of the actual stresses to allowable stresses for the critical condition. This ratio is usually the cumulative ratio of stresses in the interaction formula. In case of shear governing the design, it means the ratio of the actual shear stress to allowable shear stress. If this value exceeds the allowable ratio (default 1.0) the member is FAILED.
- f. **LOADING** provides the load case number that governed.
- g. **FX**, **MY** and **MZ** provide the design axial force, moment in local Y axes and moment in local Z axes respectively. **FX** value is followed by a letter **C** or **T** to denote **COMPRESSION** or **TENSION**.
- h. **LOCATION** specifies the actual distance from the start of the member to the section where design forces govern in case **BEAM** command or **SECTION** command is specified.

OUTPUT parameters that appear within the box are explained as follows:

- a. **MEMB** refers to the same member number for which the design is performed.
- b. **GLULAM GRADE** refers to the grade of the timber.
- c. **LAM** refers to lamination thickness provided in the input or assumed by the program. See **INPUT PARAMETERS** section.
- d. **LZ**, **LY**, **LUZ** and **LUY** are the effective lengths as provided or calculated. See **INPUT PARAMETERS** section.
- e. **JZ** and **JY** are the modifiers for the **P-DELTA** effect about Z-axis and Y-axis respectively. These are calculated by the program.

- f. CDT, CSF, WET, CCR, CTM are the allowable stress modifiers explained in the INPUT PARAMETERS section.
- g. CFZ and CFY are values of the size factors in the Z-axis and Y-axis respectively. CLZ and CLY represent the factors of lateral stability for beams about Z-axis and Y-axis respectively. These values are printed to help the user see the intermediate design values and re-check the design calculations.
- h. f_a , f_{bz} , f_{by} , f_{vz} and f_{vy} are the actual axial stress, bending stresses about Z and Y axes and horizontal shear stresses about Z and Y axes respectively. If the bending moments about both axes are less than the eccentric moments based on min. eccentricity then bending stresses are calculated based on the min. eccentricity. Refer DESIGN OPERATIONS section for details.
- i. FA, FBZ, FBY, FVZ, and FVY are the final allowable axial, bending (Z and Y axes) and horizontal shear (Z and Y axes) stresses. See "Allowable Stress per AITC Code" on page 250 for details.

4.8.1 Example Glulam Member Design

```

          STAAD.Pro CODE CHECKING - (AITC)
          *****
ALL UNITS ARE - POUN FEET (UNLESS OTHERWISE NOTED)
MEMBER      TABLE      RESULT/      CRITICAL COND/      RATIO/      LOADING/
              FX              MY              MZ              LOCATION
=====
   1      10.750X16.500  GLULAM GRADE:GLT-24F-V8_DF/DF
                   PASS      CL.5.9.2              0.014              3
                   4583.17 C              0.00              1310.87      12.0000
-----
| LEZ = 144.000 LEY = 144.000 LUZ = 144.000 LUY = 144.000 INCHES
|
| CD = 1.000 CMB = 1.000 CMT = 1.000 CMC = 1.000 CMP = 1.000
| CMV = 1.000 CME = 1.000 CFB = 1.000 CFT = 1.000 CFC = 1.000
| CFU = 1.000 CR = 1.000 CTT = 1.000 CC = 1.000 CF = 1.000
| CT = 1.000 CH = 1.000 CB = 1.000 CI = 1.000 CV = 0.000
| CLY = 0.999 CLZ = 0.997 CP = 0.934 c = 0.900 E' = 1600000.122 PSI
|
| ACTUAL STRESSES : (POUND INCH)
|          fc =          25.839 ft =          0.000
|          f_cby =          0.000 f_cbz =          32.249
|          fv =          0.000
| ALLOWABLE STRESSES: (POUND INCH)
|          FC =          1541.320 FT =          0.000
|          FCBY =          1448.972 FCBZ =          2340.789
|          FCEY =          0.000 FCEZ =          8780.903
|          FBE =          3727.248
|          FTB =          0.000 F**TB =          0.000
|          FV =          0.000 SLENDERNESS =          50.000
|-----

```

4.9 Examples

The following conventions are used to describe timber sections in STAAD.Pro

4.9.1 Example for dimensional lumber

```
STAAD PLANE EXAMPLE FOR DIMENSIONAL LUMBER
UNIT FEET POUND
JOINT COORDINATES
1 0 0 0; 2 6 0 0; 3 12 0 0; 4 18 0 0;
5 24 0 0; 6 6 3 0; 7 12 6 0; 8 18 3 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 1 6; 6 6 7; 7 7 8; 8 8 5;
9 2 6; 10 3 7; 11 4 8; 12 6 3; 13 3 8;
UNIT FEET POUND
DEFINE MATERIAL START
ISOTROPIC DFLR_SS_2X4
E 2.736E+008
POISSON 0.15
DENSITY 25
ALPHA 5.5E-006
ISOTROPIC DFLR_SS_3X6
E 2.736E+008
POISSON 0.15
DENSITY 25
ALPHA 5.5E-006
END DEFINE MATERIAL
MEMBER PROPERTY AITC
1 TO 4 9 TO 11 TABLE ST DFLR_SS_2X4
5 TO 8 12 13 TABLE ST DFLR_SS_3X6
CONSTANTS
MATERIAL DFLR_SS_2X4 MEMB 1 TO 4 9 TO 11
MATERIAL DFLR_SS_3X6 MEMB 5 TO 8 12 13
MEMBER RELEASE
9 TO 13 START MP 0.99
```

```
9 TO 13 END MP 0.99
6 END MP 0.99
7 START MP 0.99
SUPPORTS
1 PINNED
5 FIXED BUT FX MZ
UNIT FEET POUND
LOAD 1 DEAD+LIVE LOAD
SELFWEIGHT Y -1
MEMBER LOAD
1 TO 4 UNI GY -30
5 TO 8 UNI GY -40
LOAD 2 SNOW LOAD
MEMBER LOAD
5 TO 8 UNI GY -50
LOAD 3 WIND LOAD
MEMBER LOAD
5 6 UNI Y -30
7 8 UNI Y 25
LOAD COMB 11 D+L+SNOW
1 1.0 2 1.0
LOAD COMB 12 D+L+SNOW+WIND
1 1.0 2 1.0 3 1.0
PERFORM ANALYSIS PRINT STATICS CHECK
PARAMETER
CODE AITC
BEAM 1.0 ALL
CHECK CODE ALL
FINISH
```

4.9.2 Example for Glulamined lumber

```
STAAD PLANE EXAMPLE FOR GLULAM DESIGN
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 12 0 0; 3 24 0 0; 4 36 0 0; 5 0 12 0; 6 6 10
0; 7 18 6 0; 8 30 2 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 5 6; 5 6 7; 6 7 8; 7 8 4; 8 1 5; 9
2 6; 10 3 7; 11 1 6;
12 2 7; 13 3 8;
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC GLT-24F-V8_DF/DF
E 1600
POISSON 0.15
DENSITY 1.44676E-005
ALPHA 5.5E-006
END DEFINE MATERIAL
MEMBER PROPERTY
1 TO 7 PRIS YD 16.5 ZD 10.75
8 TO 13 PRIS YD 10.5 ZD 8.75
CONSTANTS
MATERIAL GLT-24F-V8_DF/DF MEMB 1 TO 13
SUPPORTS
1 4 PINNED
UNIT POUND FEET
LOAD 1 DEAD
SELFWEIGHT Y -1
LOAD 2 LIVE
MEMBER LOAD
```

```
1 TO 3 UNI GY -100
4 TO 7 UNI GY -100
LOAD COMB 3
1 1.0 2 1.0
PERFORM ANALYSIS PRINT STATICS CHECK
PARAMETER
CODE AITC
CMT 1 ALL
RATIO 0.9 ALL
CHECK CODE ALL
FINISH
```


Section 5

Commands and Input Instructions

5.1 Command Language Conventions	275
5.2 Problem Initiation and Model Title	280
5.3 Unit Specification	282
5.4 Input/Output Width Specification	284
5.5 Set Command Specification	284
5.6 Data Separator	294
5.7 Page New	295
5.8 Page Length/Eject	295
5.9 Ignore Specifications	296
5.10 No Design Specification	296
5.11 Joint Coordinates Specification	296

5.12 Member Incidences Specification	300
5.13 Elements and Surfaces	303
5.14 Plate Element Mesh Generation	309
5.15 Redefinition of Joint and Member Numbers	317
5.16 Entities as Single Objects	318
5.17 Rotation of Structure Geometry	324
5.18 Inactive/Delete Specification	324
5.19 User Steel Table Specification	326
5.20 Member Property Specification	339
5.21 Element/Surface Property Specification	368
5.22 Member/Element Releases	370
5.23 Axial Member Specifications	375
5.24 Element Plane Stress and Ignore Inplane Rotation Specification	382
5.25 Member Offset Specification	383
5.26 Specifying and Assigning Material Constants	385
5.27 Support Specifications	405
5.28 Rigid Diaphragm Modeling	420
5.29 Draw Specifications	427
5.30 Miscellaneous Settings for Dynamic Analysis	427
5.31 Definition of Load Systems	429
5.32 Loading Specifications	539
5.33 Reference Load Cases - Application	669
5.34 Frequency Calculation	670
5.35 Load Combination Specification	672
5.36 Calculation of Problem Statistics	676
5.37 Analysis Specification	676
5.38 Change Specification	709
5.39 Load List Specification	711
5.40 Load Envelope	712
5.41 Section Specification	713

5.42 Print Specifications	714
5.43 Stress/Force output printing for Surface Entities	722
5.44 Printing Section Displacements for Members	724
5.45 Printing the Force Envelope	726
5.46 Post Analysis Printer Plot Specifications	727
5.47 Size Specification	727
5.48 Steel and Aluminum Design Specifications	728
5.49 Code Checking Specification	731
5.50 Group Specification	734
5.51 Steel and Aluminum Take Off Specification	736
5.52 Timber Design Specifications	737
5.53 Concrete Design Specifications	739
5.54 Footing Design Specifications	742
5.55 Shear Wall Design	742
5.56 End Run Specification	745

This section of the manual describes in detail various commands and related instructions for STAAD. The user utilizes a command language format to communicate instructions to the program. Each of these commands either supplies some data to the program or instructs it to perform some calculations using the data already specified. The command language format and conventions are described in Section 5.1. This is followed by a description of the available commands.

Although the STAAD input can be created through the Modeling mode, it is important to understand the command language. With the knowledge of this language, it is easy to understand the problem and add or comment data as necessary. The general sequence in which the commands should appear in an input file should ideally follow the same sequence in which they are presented in this section. The commands are executed in the sequence entered. Obviously then the data needed for proper execution of a command must precede the command (e.g., Print results after Perform Analysis). Otherwise, the commands can be provided in any order with the following exceptions.

- i. All design related data can be provided only after the analysis command.
- ii. All load cases and load combinations must be provided together, except in

a case where the **CHANGE** command is used (refer to section 5.38).
Additional load cases can be provided in the latter part of input.

All input data provided is stored by the program. Data can be added, deleted or modified within an existing data file.

In STAAD.Pro 2006 and earlier, all analytical calculations such as joint displacements, eigenvalues and eigenvectors were calculated using an analysis engine which for identification purposes is known as the Basic Solver. This engine has been able to handle the analytical requirements for a vast majority of STAAD models that users have created in the recent years.

As computer resources such as processor speed, memory and disk space have grown, STAAD users are also creating larger models. As a result, numerically faster algorithms and solution techniques have become necessary. Also, new features such as pushover analysis and buckling analysis which are outside the scope of the standard engine have made it necessary to introduce a new engine which is known as the Advanced Solver. This new Solver is available as an alternative engine effective from STAAD.Pro 2007, Build 1001.

The Advanced Solver

As described above, the Advanced solver is a new addition to the STAAD Analysis Engine (note 1) which can be used for solving both static and dynamic problems. It is part of the STAAD engine with no special command required to run it. It is automatically activated if a suitable license is available (note 2), however, this can be turned off and the Basic Solver used by including the option:

SET STAR 0

This command must be included in the header information block at the start of the file and before the first JOINT command block.

The engine can operate in two modes, **in-core** and **out-of-core**. The in-core solver will be used for models with under 20000 joints and the out-of-core solver for models over 20000 joints. In most situations, the in-core mode will provide the quickest solution, but where there is insufficient memory available, then the engine will use the out-of-core mode. Again, selection of the mode is automatically chosen by the analysis, but can be over-ridden.

The full set of overrides for the advanced engine is:

SET STAR -3 use in-core solver regardless of size

SET STAR 4 use out-of-core solver regardless of size

SET STAR 3 default

SET STAR 0 use Basic STAAD solver

Notes

1. The Advanced Solver is *not* available for use with a Stardyne Analysis.
2. To use this feature requires access to a **STAAD Advanced license**. If you do not currently have this feature, please contact your account manager.
3. Global Euler Buckling analysis is different between the two solvers.

5.1 Command Language Conventions

This section describes the command language used in STAAD. First, the various elements of the language are discussed and then the command format is described in details.

5.1.1 Elements of STAAD Commands

Integer Numbers

Integer numbers are whole numbers written without a decimal point. These numbers are designated as i_1, i_2, \dots , etc., and should not contain any decimal point. Negative signs (-) are permitted in front of these numbers. Omit the sign for positive. No spaces between the sign and the number.

Floating Point Numbers

These are real numbers which may contain a decimal portion. These numbers are designated as f_1, f_2, \dots etc.. Values may have a decimal point and/or exponent. When specifying numbers with magnitude less than 1/100, it is advisable to use the E format to avoid precision related errors. Negative signs (-) are permitted in front of these numbers. Omit the sign for positive. No spaces between the sign and the number. Limit these to 24 characters.

Example

```
5055.32  0.73  -8.9  732
5E3     -3.4E-6
```

etc.

The decimal point may be omitted if the decimal portion of the number is zero.

Section 5 Commands and Input Instructions

5.1 Command Language Conventions

Alphanumeric

These are characters, which are used to construct the names for data, titles or commands. Alphabetic characters may be input in upper or lower case letters. No quotation marks are needed to enclose them.

Example

```
MEMBER PROPERTIES  
1 TO 8 TABLE ST W8X35
```

Repetitive Data

Repetitive numerical data may be provided in some (but not all) input tables such as joint coordinates by using the following format:

n*f

Where:

n = number of times data has to be repeated

f = numeric data, integer and floating point

Example

```
JOINT COORDINATES  
1 3*0.
```

This joint coordinate specification is same as:

```
1 0. 0. 0.
```

5.1.2 Command Formats

Free-Format Input

All input to STAAD is in free-format style. Input data items should be separated by blank spaces (not commas) from the other input data items. Quotation marks are never needed to separate any alphabetic words such as data, commands or titles. Limit a data item to 24 characters.

Commenting Input

For documentation of a STAAD data file, the facility to provide comments is available. Comments can be included by providing an asterisk (*) mark as the first non-blank character in any line. The line with the comment is "echoed" in the output file but not processed by the program.

Example

```
JOINT LOAD
* THE FOLLOWING IS AN EQUIPMENT LOAD
2 3 7 FY 35.0
ETC.
```

Meaning of Underlining in the Manual

Exact command formats are described in the latter part of this section. Many words in the commands and data may be abbreviated. The full word intended is given in the command description with the portion actually required (the abbreviation) underlined.

For example, if the word MEMBER is used in a command, only the portion **MEMB** need be input. It is clearer for others reading the output if the entire word is used, but an experienced user may desire to use the abbreviations.

Meaning of Braces and Parenthesis

In some command formats, braces enclose a number of choices, which are arranged vertically or separated by a | character. One and only one of the choices can be selected. However, several of the listed choices may be selected if an asterisk (*) mark is located outside the braces.

Example

```
{XY | YZ | XZ}
```

In the above example, you must make a choice of XY or YZ or XZ.

Note: In some instances, the choices will be explicitly defined using "or" for clarification.

Example

Section 5 Commands and Input Instructions

5.1 Command Language Conventions

***{FX | FY | FZ}**

Here, you can choose one or all of the listing (FX, FY and FZ), in any order.

Parentheses, (), enclosing a portion of a command indicate that the enclosed portion is optional. The presence or absence of this portion affects the meaning of the command, as is explained in the description of the particular command.

Example

```
PRINT (MEMBER) FORCES
```

```
PERFORM ANALYSIS (PRINT LOAD DATA)
```

In the first line, the word **MEMBER** may be omitted with no change of the meaning of the command. In the second line, the **PRINT LOAD DATA** command may also be omitted, in which case the load data will not be printed.

Multiple Data Separator

Multiple data can be provided on a single line, if they are separated by a semicolon (;) character. One restriction is that a semicolon can not separate consecutive commands. They must appear on separate lines.

Example

```
MEMBER INCIDENCES
```

```
1 1 2; 2 2 3; 3 3 4
```

etc.

Possible Error:

```
PRINT FORCES; PRINT STRESSES
```

In the above case, only the **PRINT FORCES** command is processed and the **PRINT STRESSES** command is ignored.

Listing Data

In some STAAD command descriptions, the word "list" is used to identify a list of joints, members/elements, or loading cases. The format of a list can be defined as follows:

```
list = *{ i1 i2 i3 ... | i1 TO i2 (BY i3) | X or Y or Z }
```

TO means all integers from the first (i_1) to the second (i_2) inclusive. **BY** means that the numbers are incremented by an amount equal to the third data item (i_3). If **BY** i_3 is omitted, the increment will be set to one. Sometimes the list may be too long³ to fit on one line, in which case the list may be continued to the next line by providing a hyphen preceded by a blank. Also, only a list may be continued and not any other type of data.

Instead of a numerical list, a single group-name may be entered if that group was previously defined.

Instead of a numerical list, the specification X (or Y or Z) may be used. This specification will include all **MEMBERS** parallel to the global direction specified. Note that this is not applicable to **JOINTS** or **ELEMENTS**.

Note: ALL, BEAM, PLATE, SOLID. Do not use these unless the documentation for a command specifically mentions them as available for that command. **ALL** means all members and elements, **BEAM** means all members, etc.

Continuing a command to the next line

Only lists may be continued to the next line by ending the line with a blank and hyphen (see above) with few exceptions: Multilinear spring supports, Supports, Master/Slave. Others have special types of continuations. Please follow the command descriptions.

Example

```
2 4 7 TO 13 BY 2 19 TO 22 -
28 31 TO 33 FX 10.0
```

This list of items is the same as:

```
2 4 7 9 11 13 19 20 21 22 28 31 32 33 FX 10.0
```

Possible Error:

```
3 5 TO 9 11 15 -
FX 10.0
```

Section 5 Commands and Input Instructions

5.2 Problem Initiation and Model Title

In this case, the continuation mark for list items is used when list items are not continued. This will result in an error message or possibly unpredictable results.

5.1.3 Listing of Objects by Specification of Global Ranges

Used to specify lists of objects (e.g., joints, members, and/or elements) by providing global ranges. The general format of the specification is as follows.

General Format

```
{ XRANGE | YRANGE | ZRANGE } f1, f2
```

Where:

XRANGE, YRANGE, ZRANGE = direction of range (parallel to global X, Y, Z directions respectively)

f1, f2 = values (in current unit system) that defines the specified range.

Notes

1. Only one range direction (XRANGE, YRANGE etc.) is allowed per list. (Exceptions: Area/Floor load and Master/Slave).
2. No other items may be in the list.
3. The values defining the range (f1, f2) must be in the current unit system.

Example

```
MEMBER TRUSS
XRANGE 20. 70.
CONSTANTS
E STEEL YRANGE 10. 55.
```

In the above example, a XRANGE is specified with values of 20. and 70. This range will include all members lying entirely within a range parallel to the global X-axis and limited by X=20 and X=70.

5.2 Problem Initiation and Model Title

This command initiates the STAAD run and is also used to specify the type of the structure and provide an optional title.

Any STAAD input file must start with the word **STAAD**. Following type specifications are available:

- PLANE= Plane frame structure
- SPACE= Space frame structure
- TRUSS= Plane or space truss structure
- FLOOR= Floor structure

General Format

STAAD { PLANE | SPACE | TRUSS | FLOOR } (any title a₁)

Where:

a₁ = Any title for the problem. This title will appear on the top of every output page. To include additional information in the page header, use a comment line containing the pertinent information as the second line of input.

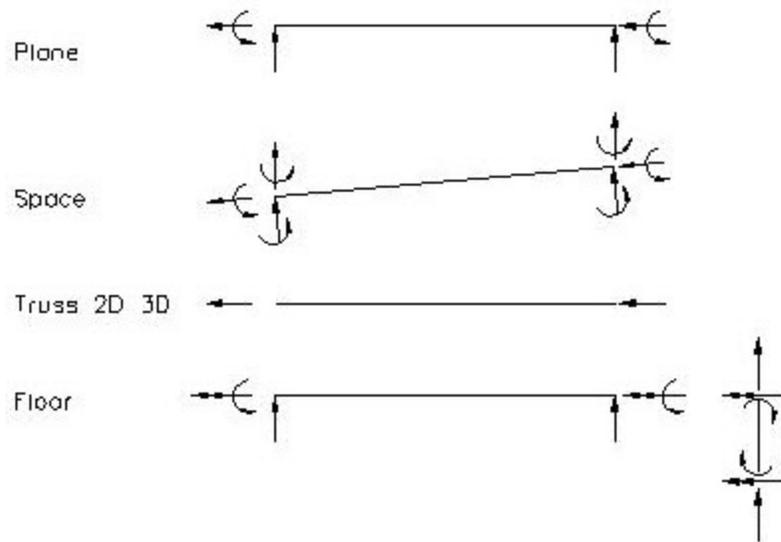
Notes

1. Care must be taken about choosing the type of the structure. The choice is dependent on the various degrees of freedom that need to be considered in the analysis. The following figure illustrates the degrees of freedoms considered in the various type specifications. Detailed discussions are available in Section 1.3. PLANE indicates the XY plane for Y up and the XZ plane for Z up. FLOOR indicates the XZ floor for Y up and the XY floor for Z up.

Figure 5-1: Structure type A) Plane, B) Space, C) Truss 2D or 3D, and D) Floor

Section 5 Commands and Input Instructions

5.3 Unit Specification



2. The optional title provided by you are printed on top of every page of the output. You can use this facility to customize his output.

Limits

The following limits to model size are effective for STAAD.Pro V8i (SELECTseries 2) (release 20.07.07).

1. Joint number 1 to 999999
2. Number of joints: 200000*
3. Member/Element numbers: 1 to 999999
4. Number of Members, Plates, and Solids: 225000*
5. Load Case numbers: 1 to 99999
6. Number of primary and combination cases: 10101
7. Number of modes and frequencies: 2700
8. Number of load cases that may be combined by a Repeat Load or Load Combination command: 550

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

5.3 Unit Specification

This command allows you to specify or change length and force units for input and output.

General Format

UNIT *{ length-unit, force-unit}

length-unit = { INCHES | FEET or FT or FO | CM | METER | MMS
| DME | KM }

force-unit = { KIP | POUND | KG | MTON | NEWTON | KNS | MNS
| DNS }

Note: DME denotes Decimeters. MNS denotes mega Newtons (1000 KiloNewtons) and DNS denotes DecaNewtons (10 Newtons). MTON denotes Metric Ton (1000 kilograms). All other units are self explanatory.

Description

The UNIT command can be specified any number of times during an analysis. All data is assumed to be in the most recent unit specification preceding that data. Also, the input-unit for angles is always degrees. However, the output unit for joint rotations (in joint displacement) is radians. For all output, the units are clearly specified by the program.

Example

```
UNIT KIP FT
UNIT INCH
UNIT METER KNS
UNIT CM MTON
```

Notes

- This command may be used as frequently as needed to specify data or generate output in the desired length and/or force units. Mixing of different unit systems (Imperial, Metric, SI etc.) is allowed.
- This command may be anywhere a primary level command (e.g., JOINT COORD, MEMBER INCIDENCE, etc.) can be. In addition, it may also be wherever a first level load command may be.

Section 5 Commands and Input Instructions

5.4 Input/Output Width Specification

- c. Exceptions: The second level load commands (MEMBER LOAD, JOINT LOAD and FLOOR LOAD), allow the UNIT command to be entered on a separate line, not within a continuation or between semi-colons.

5.4 Input/Output Width Specification

These commands may be used to specify the width(s) of the lines of output file (s).

For INPUT width, 79 is always used. The program can create output using two different output widths - 72 (default) and 118. The 72-character width may be used for display on most CRTs and for printing on “portrait” wide paper. The 118-character width may be used for printing on “landscape” wide paper.

Note: This is a customization facility that may be used to improve the presentation quality of the output documents.

General Format

{INPUT | OUTPUT} WIDTH i_1

Where:

$i_1 = 72$ or 118 depending on narrow or wide output. 72 is the default value.

5.5 Set Command Specification

This command allows the user to set various general specifications for the analysis/design run.

General Format

SET { NL i_1 | {DISPLACEMENT i_2 | PDELTA TOL i_9 } | SDAMP i_3 | WARP i_4 | ITERLIM i_5 | PRINT i_7 | NOPRINT DIRECT | SHEAR | ECHO { ON | OFF } | GUI i_6 | Z UP | DEFLECTION CUTOFF f_1 | FLOOR LOAD TOLERANCE F_2 }

Where:

i_1 = Maximum number of primary load cases (NL)

i_3 = Damping ratio to be used for all springs in computing the modal composite damping in dynamics.

i_4 = Warping restraint ratio to be used for I section members in computing the torsional rigidity. If 0.0 then no warping restraint, normal default option; If 1.0 then full warping restraint. C_w , the warping constant, will be computed and used in the torsional rigidity calculation. Values between 0.0 and 1.0 will result in a partial warping restraint.

i_5 = Maximum number of tension/compression iterations.

i_6 = 1, Bypass forming data and files needed for post-processing.

i_7 = Used to suppress some warning messages or to include additional output. Refer to table 5-1 for values.

Note: The following **SET** commands contain values with associated units and should appear after a **UNIT** command and before the first **JOINT** command.

Note: Both i_2 and i_9 are the P-Delta Convergence criteria input. There are two methods available, enter **SET DISPLACEMENT i_2** or **SET PDELTA TOL i_9** . Enter only one.

i_2 = If the change in the Euclidean norm of the displacement vector from one **PDELTA** iteration to the next is less than this convergence tolerance value; then the iteration has converged for the case being analyzed.

i_9 = If the maximum absolute change in displacement of each dof from one **PDELTA** iteration to the next is less than this convergence tolerance value; then the iteration has converged for the case being analyzed.

f_1 = If the absolute value of the maximum section displacement is less than f_1 after two iterations; then it is converged. Rapidly diverging minor axis displacement will not occur until after two iterations. f_1 is in current length units.

f_2 = tolerance value used

Description

Table 5-1: Commonly used SET commands

Command	Description
SET NL	<p>The SET NL command is used in a multiple analysis run if the user wants to add more primary load cases after one analysis has been performed. Specifically, for those examples, which use the CHANGE command (section 5.38), if the user wants to add more primary load cases, the NL value should be set to the maximum number (or slightly more) with the SET NL command. The program will then be able to set aside additional memory space for information to be added later. This command should be provided before any joint, member or load specifications. The value for i_1 should not be much greater than the maximum number of primary load cases.</p>

Command	Description
SET DISPLACEMENT or PDELTA TOL	<p>For PDELTA ANALYSIS with CONVERGE option there are two convergence methodologies to choose from (refer to section 5.37.2 for additional information)</p> <ol style="list-style-type: none"> 1. The SET DISPLACEMENT i_2 command is used to specify the convergence tolerance. If the Euclidean norm RMS displacement of two consecutive iterations changes less than the value entered, then that load case is converged. This command should be placed before the JOINT COORDINATE specification. The default tolerance value, i_2, is equal to the maximum span of structure divided by 120. The convergence tolerance for the Euclidean norm is difficult to know, so using this option is not recommended. 2. The SET PDELTA TOL i_9 command selects this second method. Default tolerance value, i_9, is 0.01 inch. If the maximum change in each displacement dof from two consecutive iterations is less than $ftol$, then that load case is converged. This command should be placed before the JOINT COORDINATE specification and after a UNIT command.

Section 5 Commands and Input Instructions

5.5 Set Command Specification

Command	Description
SET SDAMP	The SET SDAMP command will allow the damping of springs to be considered in computing the composite modal damping for each mode in a dynamic solution. This command is not used unless CDAMP ratios are also entered for the members and elements in the CONSTANTS command. Composite damping is generally only used if there are many modes in the dynamic solution and there are a wide range of damping ratios in the springs, members, or elements.
SET WARP	The SET WARP command will allow the I section member end warping restraint to be considered in calculating the torsional stiffness rigidity. Full or partial or no warping restraint are allowed.

Command	Description
SET ITERLIM	<p>The SET ITERLIM command is for raising the maximum iteration limit above the default of 10 in tension/compression iterations. Since this iterative procedure will not necessarily converge, this option of more iterations may not help and should be used with caution. The minimum iteration limit that may be entered is 3.</p> <p>After any tension/compression analysis, the output file (file extension .ANL) should be scanned for warnings of non-convergence. Do not use results from non-converged cases.</p>

Section 5 Commands and Input Instructions

5.5 Set Command Specification

Command	Description
SET PRINT	<p>The following values can be used to suppress the described warnings or include the described additional results in the output:</p> <ol style="list-style-type: none">1. Omit zero stiffness message, Rotational zero stiffness message due to solids, and "Node not connected. OK if master/slave" message.2. Omit Member in list does not exist message.3. Omit joint not connected message.5. Turn off floor load message.10. Turns on some iteration messages in direct analysis.17. Write rotational masses to mass text file; otherwise only translational masses written. Print scaled modal results for RSA and some force data by floor for RSA.
SET NOPRINT DIRECT	<p>Used to turn off the tau-b details in the output file when running a Direct Analysis.</p>
SET SHEAR	<p>The SET SHEAR command is for omitting the additional pure shear distortion stiffness terms in forming beam member stiffnesses. With this command you can exactly match simple text-book beam theory results.</p>

Command	Description
SET ECHO	The SET ECHO ON command will activate and the SET ECHO OFF command will deactivate the echoing of input file commands in the output file. In the absence of the SET ECHO command, input file commands will be echoed back to the output file.
SET GUI	After the calculations are completed, and before the Analysis window is closed, the program creates several files for the purpose of displaying results in the post processing mode. In large models, this can be a time consuming process. If the user's goal is to look at results in the output file only (file extension .ANL) and does not intend to go into the post-processing mode, he/she could instruct the program to skip the process of creating those files. The SET GUI 1 may be specified immediately before the FINISH command, or somewhere near the beginning of the file after STAAD SPACE .

Section 5 Commands and Input Instructions

5.5 Set Command Specification

Command	Description
SET Z	<p>By default, the Y-axis is the vertical axis. However, the SET Z UP command may be used to model situations where Z-axis represents the vertical axis (direction of gravity load) of the structure. This situation may arise if the input geometry is created through some CAD software. This command will affect the default BETA angle specification. However, BETA can be set to a certain value for all members parallel to a particular global axis by using the MEMBER X (or Y or Z) type of listing. For additional information, see the CONSTANTs specification (Section 5.26).</p> <p>The SET Z UP Command directly influences the values of the following input:</p> <ol style="list-style-type: none">a. JOINT COORDINATEb. Input for the PERFORM ROTATION Commandc. BETA ANGLE <p>The following features of STAAD cannot be used with the SET Z UP command:</p> <p>Automatic Generation of Spring Supports for Mat Foundations</p>

Command	Description
SET STAR n	Instructs the program which solver to use. Refer to Section 5. for additional information on using the Advanced Solver.
SET DEFLECTION CUTOFF	Used to arrest huge displacements in minor axis due to small delta effects.
SET FLOOR LOAD TOLERANCE	Used to specify the tolerance for out of plane nodes in a floor load.

The following table contains a list of other rarely used SET commands

Table 5-2: Infrequently used SET commands

Command	Description
SET <u>DATA</u> CHECK	Ignored
SET <u>RUN</u>	Ignored
SET <u>COMPRESS</u>	Turn OFF file compression
SET <u>SOLUTION</u> <u>INCORE</u>	Use determinant search for frequencies for small problems.
SET <u>SOLVER</u>	should ignore
SET <u>EXM</u>	should ignore (Extended Memory)
SET <u>NJ</u>	should ignore
SET <u>MN</u>	should ignore
SET <u>CONNECTIVITY</u>	should ignore
SET <u>MASS</u>	= 1, Use generated moments at masses.
SET <u>MODAL</u>	should ignore

Section 5 Commands and Input Instructions

5.6 Data Separator

Command	Description
SET <u>THISTORY</u>	=2, Use exact force integration in time history.
SET <u>INTERPOLATION</u>	LIN or LOG for spectra
SET <u>DISPLACEMENT METHOD</u>	should ignore
SET <u>s1</u>	s1 = file extension for L43
SET <u>BUBBLE</u>	= 1, Do not use bubble fns in solids
SET <u>NOSECT</u>	No section results will be calculated
SET <u>TMH</u>	should ignore
SET <u>SSVECT</u>	To instruct the program to use a different initial set of trial vectors for eigen-solution. May be used if eigen extraction fails.
SET <u>INCLINED REACTION</u>	To obtain reactions at inclined supports in the inclined axis system.
SET <u>GROUP DUPLICATES</u>	Followed by an integer value which is used to specify the number of groups to which one model entity (node, member, plate, or solid object) may belong. Must be in the range of four to 100, inclusive. The default value is 10 groups.

5.6 Data Separator

This command may be used to specify the desired separator character that can be used to separate multiple lines of data on a single line of input.

The semicolon (;) is the default character which functions as the separator for

multiple line data on one line. However, this separator character can be changed by the **SEPARATOR** command to any character.

Note: Comma (,) or asterisk (*) may *not* be used as a separator character.

General Format

SEPARATOR a₁

5.7 Page New

This command may be used to instruct the program to start a new page of output.

With this command, a new page of output can be started. This command provides the flexibility, the user needs, to design the output format.

Note: The presentation quality of the output document may be improved by using this command properly.

General Format

PAGE NEW

5.8 Page Length/Eject

These commands may be used to specify the page length of the output and the desired page eject character.

General Format

PAGE { LENGTH i | EJECT a₁ }

Where:

i = The page length in STAAD output is based on a default value of 60 lines . However, the user may change the page length to any number i (number of lines per page) desired.

a₁ = Standard page eject character (CNTRL L for PCs and 1 for Mini/Mfrm) is embedded in the STAAD program. The PAGE EJECT command with the input of the character a₁ will alter the default page eject character in the program. A blank character will suppress page ejection.

5.9 Ignore Specifications

This command allows you to provide member lists in a convenient way without triggering error messages pertaining to non-existent member numbers.

The **IGNORE LIST** command may be used if you want the program to ignore any nonexistent member(s) that may be included in a member list specification. For example, for the sake of simplicity, a list of members may be specified as MEMB 3 TO 40 where members 10 and 11 do not exist. An error message can be avoided in this situation by providing the **IGNORE LIST** command anywhere in the beginning of input. A warning message, however, will appear for each nonexistent member.

General format

IGNORE LIST

5.10 No Design Specification

This command allows you to declare that no design operations will be performed during the run. The memory reserved for design will be released to accommodate larger analysis jobs.

STAAD always assumes that at some point in the input, you may want to perform design for steel, concrete, etc. members. These design processes require more computer memory. If memory availability is a problem, the above command may be used to eliminate extra memory requirements.

General Format

INPUT NODESIGN

5.11 Joint Coordinates Specification

These commands allow the user to specify and generate the coordinates of the joints of the structure. The **JOINT COORDINATES** command initiates the specification of the coordinates. The **REPEAT** and **REPEAT ALL** commands allow easy generation of coordinates using repetitive patterns.

General Format

JOINT COORDINATES (CYLINDRICAL (REVERSE)) (NOCHECK)

band-spec

$i_1, x_1, y_1, z_1, (i_2, x_2, y_2, z_2, i_3)$

```

REPEAT   n, xi1, yi1, zi1, (xi2, yi2, zi2, ..., xin, yin, zin)
REPEAT ALL   n, xi1, yi1, zi1, (xi2, yi2, zi2, ..., xin, yin,
zin)

```

n is limited to 150

```
JTORIG xOrigin yOrigin zOrigin
```

```
band-spec = (NOREDUCE BAND)
```

Where:

NOCHECK = Do not perform check for multiple structures or orphan joints.

Description

The command **JOINT COORDINATES** specifies a Cartesian Coordinate System. Joints are defined using the global X, Y and Z coordinates. The command **JOINT COORDINATES CYLINDRICAL** specifies a Cylindrical Coordinate System. Joints are defined using the r, q and z coordinates. **JOINT COORDINATES CYLINDRICAL REVERSE** specifies a Reverse Cylindrical Coordinate system. Joints are defined using the r, y and q coordinates. See "Global Coordinate System" on page 10 for details and figures.

JTORIG causes the program to use a different origin than (o, o, o) for all of the joints entered with this **JOINT COORDINATES** command. It is useful in instances such as when the center of cylinder is not at (o, o, o) but at a different point in space. The **JTORIG** command should be entered on a separate command line. Basically after the joint coordinates are entered or generated, then the xOrigin, yOrigin, and zOrigin values are added to the coordinates. For example a cylinder could be generated about the Y axis then moved by this command to its proper place. To create multiple offset structural parts, enter additional **JOINT COORDINATES** commands, each one followed by its **JTORIG** command. An example showing the use of this command is provided later in this section.

The multiple **JOINT COORDINATES** command concept allows **UNIT** changes and **PERFORM ROTATION** commands in between; such that these commands would apply to a selected portion of the joints. However, the **PERFORM ROTATION** command applies to all prior defined joints, not just those in the previous **JOINT COORDINATE** command.

NOREDUCE BAND causes the program to execute without performing a bandwidth reduction.

Example

Section 5 Commands and Input Instructions

5.11 Joint Coordinates Specification

JOINT COORDINATES NOREDUCE BAND

The **REPEAT** command causes the previous line of input to be repeated 'n' number of times with specified coordinate increments. The **REPEAT ALL** command functions similar to the **REPEAT** command except that it repeats all previously specified input back to the most recent **REPEAT ALL** command, or all joint data if no previous **REPEAT ALL** command has been given.

Note: Use **REPEAT ALL 0** to start a section of data to be repeated if necessary. When using the **REPEAT** and **REPEAT ALL** commands, joint numbering must be consecutive and should begin with 1.

* i_1 = The joint number for which the coordinates are provided. Any integer number (maximum 6 digits) within the limit (see section 5.2 for limit) is permitted.

x_1, y_1 and z_1 = X, Y & Z (R, & Z for cylindrical or R, Y & for cylindrical reverse) coordinates of the joint.

For **PLANE** analyses z_1 is an optional data item when defining input for individual joints. z_1 is always required for joint generation. The following are used only if joints are to be generated.

* i_2 = The second joint number to which the joint coordinates are generated.

x_2, y_2 , and z_2 = X, Y & Z (R, & Z for cylindrical or R, Y & for cylindrical reverse) coordinates of the joint i_2 .

i_3 = Joint number increment by which the generated joints will be incremented. Defaults to 1 if left out.

n = Number of times repeat is to be carried out. Note that "n" cannot exceed 150 in any one single **REPEAT** command.

xi_k, yi_k & zi_k = X, Y & Z (R, & Z [R, Y &]) coordinate increments for kth repeat.

The X, Y and Z (R, & Z [R, Y &]) coordinates will be equally spaced between i_1 and i_2 .

* The **REPEAT** command uses the highest joint number entered so far plus one for the intermediate generated joint numbers.

Example 1

```

JOINT COORDINATES
1  10.5  2.0  8.5
2  0.0  0.0  0.0
3  5.25  0.0  8.5  6  50.25  0.0  8.5

```

In this example, X Y Z coordinates of joints 1 to 6 are provided. Note that the joints between 3 & 6 will be generated with joints equally spaced from 3 to 6. Hence, joint 4 will have coordinates of 20.25 0.0 8.5 and joint 5 will have coordinates of 35.25 0.0 8.5.

Example 2

```

JOINT COORDINATES
1  0.0  0.0  0.0  4  45  0.0  0.0
REPEAT  4  0.0  0.0  15.0
REPEAT ALL  10  0.0  10.0  0.0

```

Here, the 220 joint coordinates of a ten story 3 X 4-bay structure are generated. The **REPEAT** command repeats the first input line 4 times, incrementing each Z coordinate by 15. Thus, the first 2 lines are sufficient to create a "floor" of twenty joints.

```

1  0.  0.  0.  ; 2  15.  0.  0.  ; 3  30.  0.  0.  ;
4  45.  0.  0.
5  0.  0.  15.  ; 6  15.  0.  15.  ; 7  30.  0.  15.
; 8  45.  0.  15.
.....      .....      .....      .....
17  0.  0.  60.  ; 18  15.  0.  60.  ; 19  30.  0.
60.  ; 20  45.  0.  60.

```

The **REPEAT ALL** command repeats all previous data (i.e., the 20 joint "floor") ten times, incrementing the Y coordinate by 10 each time. This creates the 200 remaining joints of the structure.

Section 5 Commands and Input Instructions

5.12 Member Incidences Specification

Example 3

```
21 0.0 10.0 0.0 ; 22 15.0 10.0 0.0 ; ... ;
40 45.0 10.0 60.0 ; 41 0.0 20.0 0.0 ; ... ;
200 45.0 90.0 60.0 ; 201 0.0 100.0 0.0 ; ... ;
219 30.0 100.0 60.0 ; 220 45.0 100.0 60.0
```

The following examples illustrate various uses of the **REPEAT** command.

```
REPEAT 10 5. 10. 5.
```

The above **REPEAT** command will repeat the last input line 10 times using the same set of increments (i.e., $x = 5.$, $y = 10.$, $z = 5.$)

```
REPEAT 3 2. 10. 5. 3. 15. 3. 5. 20. 3.
```

The above **REPEAT** command will repeat the last input line three times. Each repeat operation will use a different increment set.

```
REPEAT 10 0. 12. 0. 15*0 0. 10. 0. 9*0
```

The above **REPEAT** command will repeat the last input line 10 times; six times using x , y and z increments of 0., 12. and 0., and four times using increments of 0., 10. and 0. Each x , y and z value of 0 represents no change from the previous increment. To create the 2nd through 6th repeats, five sets of 0., 0. and 0. ($15*0$) are supplied. The seventh repeat is done with increments of 0., 10. and 0. The 8th through 10th repeats are done with the same increments as 7, and is represented as $9*0$.

Note: The **PRINT JOINT COORDINATE** command may be used to verify the joint coordinates provided or generated by **REPEAT** and **REPEAT ALL** commands. Also, use the Post Processing facility to verify geometry graphically.

5.12 Member Incidences Specification

This set of commands is used to specify members by defining connectivity between joints. **REPEAT** and **REPEAT ALL** commands are available to facilitate

generation of repetitive patterns.

The member/element incidences must be defined such that the model developed represents one single structure only, not two or more separate structures. STAAD is capable of detecting multiple structures automatically.

General Format

```

MEMBER  INCIDENCES
i1, i2, i3, ( i4, i5, i6 )
REPEAT n, mi, ji
REPEAT ALL n, mi, ji

```

Description

The **REPEAT** command causes the previous line of input to be repeated 'n' number of times with specified member and joint increments. The **REPEAT ALL** command functions similar to the **REPEAT** command except that it repeats all previously specified input back to the most recent **REPEAT ALL** command or to the beginning of the specification if no previous **REPEAT ALL** command has been issued.

Note: When using **REPEAT** and **REPEAT ALL** commands, member numbering must be consecutive.

i_1 = Member number for which incidences are provided. Any integer number (maximum six digits) is permitted.

i_2 = Start joint number.

i_3 = End joint number.

Note: Use **REPEAT ALL 0** to start a set of members that will be repeated if you don't want to repeat back to the last **REPEAT ALL**.

The following data are used for member generation only:

i_4 = Second member number to which members will be generated.

i_5 = Member number increment for generation.

Section 5 Commands and Input Instructions

5.12 Member Incidences Specification

i_6 = Joint number increment which will be added to the incident joints. (i_5 and i_6 will default to 1 if left out.)

n = Number of times repeat is to be carried out.

m_i = Member number increment

j_i = Joint number increment

The **PRINT MEMBER INFO** command may be used to verify the member incidences provided or generated by **REPEAT** and **REPEAT ALL** commands.

Hint: Use the Post Processing facility to verify geometry graphically.

Example 1

```
MEMBER INCIDENCES
1  1  2
2  5  7  5
7 11 13 13  2  3
```

In this example, member 1 goes from joint 1 to 2. Member 2 is connected between joints 5 and 7. Member numbers from 3 to 5 will be generated with a member number increment of 1 and a joint number increment 1 (by default). That is, member 3 goes from 6 to 8, member 4 from 7 to 9, member 5 from 8 to 10. Similarly, in the next line, member 7 will be from 11 to 13, member 9 will be from 14 to 16, 11 from 17 to 19 and 13 from 20 to 22.

Example 2

```
MEMBER INCIDENCES
1  1  21  20
21 21  22  23
REPEAT  4  3  4
36 21  25  39
REPEAT  3  4  4
REPEAT  ALL  9  51  20
```

This example creates the 510 members of a ten story 3 X 4-bay structure (this is a continuation of the example started in Section 5.12). The first input line creates the twenty columns of the first floor:

```
1 1 21 ; 2 2 22 ; 3 3 23 ; ... ; 19 19 39
; 20 20 40
```

The two commands (21 21 22 23 and REPEAT 4 3 4) create 15 members which are the second floor "floor" beams running, for example, in the east-west direction:

```
21 21 22; 22 22 23; 23 23 24
24 25 26; 25 26 27; 26 27 28
...     ...     ...
33 37 38; 34 38 39; 35 39 40
```

The next two commands (36 21 25 39 and REPEAT 3 4 4) function similar to the previous two commands, but here create the 16 second floor "floor" beams running in the north-south direction:

```
36 21 25; 37 22 26; 38 23 27; 39 24 28
40 25 29; 41 26 30; 42 27 31; 43 28 32
...     ...     ...     ...
48 33 37; 49 34 38; 50 35 39; 51 36 40
```

The preceding commands have created a single floor unit of both beams and columns, a total of 51 members. The REPEAT ALL now repeats this unit nine times, generating 459 new members and finishing the ten story structure. The member number is incremented by 51 (the number of members in a repeating unit) and the joint number is incremented by 20, (the number of joints on one floor).

5.13 Elements and Surfaces

This section describes the commands used to specify plates (i.e., shells), surfaces, and solids.

5.13.1 Plate and Shell Element Incidence Specification

This set of commands is used to specify ELEMENTs by defining the connectivity between JOINTs. REPEAT and REPEAT ALL commands are available to facilitate

Section 5 Commands and Input Instructions

5.13 Elements and Surfaces

generation of repetitive patterns.

The element incidences must be defined such that the model developed represents one single structure only, not two or more separate structures. STAAD is capable of detecting multiple structures automatically.

General format

ELEMENT INCIDENCES (SHELL)

$i_1, i_2, i_3, i_4, (i_5), (\text{ TO } i_6, i_7, i_8)$

REPEAT n, e_i, j_i

REPEAT ALL n, e_i, j_i

Description

ELEMENT INCIDENCES SHELL must be provided immediately after MEMBER INCIDENCES (if any) are specified. The REPEAT command causes the previous line of input to be repeated 'n' number of times with specified element and joint increments. The REPEAT ALL command functions similar to the REPEAT command, except that it repeats all previously specified input back to the most recent REPEAT ALL command; or to the beginning of the specification if no previous REPEAT ALL command had been issued. Use "REPEAT ALL o o o" to start a set of elements that will be repeated if you don't want to repeat back to the last REPEAT ALL.

i_1 = Element number (any number up to six digits). If MEMBER INCIDENCE is provided, this number must not coincide with any MEMBER number.

$i_2 \dots i_5$ = Clockwise or counterclockwise joint numbers which represent the element connectivity. i_5 is not needed for triangular (3 noded) elements.

The following data is needed if elements are to be generated:

i_6 = Last element number to which elements are generated.

i_7 = Element number increment by which elements are generated.
Defaults to 1 if omitted.

i_8 = Joint number increment which will be added to incident joints.
Defaults to 1 if omitted.

The following data is needed if REPEAT or REPEAT ALL commands are used to generate elements:

n = Number of times repeat is to be carried out.

e_i = Element number increment.

j_i = Joint number increment.

Example

```
ELEMENT INCIDENCE
1 1 2 7 6
2 3 4 8
3 8 9 11 10 TO 8
9 1 3 7 TO 14
```

Notes

The PRINT ELEMENT INFO command may be used to verify the element incidences provided or generated by REPEAT and REPEAT ALL commands.

Also, use the Post Processing facility to verify geometry graphically.

5.13.2 Solid Element Incidences Specification

4- through 8-noded elements, also known as solid elements, are described using the commands described below. Technical information on these elements is available in section 1.6.2 of this manual.

General Format

The element incidences for solid elements are to be identified using the expression SOLID to distinguish them from PLATE/SHELL elements.

ELEMENT INCIDENCES SOLID

$i_1, i_2, i_3, i_4, i_5, i_6, i_7, i_8, i_9, (TO i_{10}, i_{11}, i_{12})$

REPEAT n, e_i, j_i

REPEAT ALL n, e_i, j_i

Section 5 Commands and Input Instructions

5.13 Elements and Surfaces

Description

ELEMENT INCIDENCES SOLID must be provided immediately after MEMBER INCIDENCES (if any) are specified as well as after the ELEMENT INCIDENCES SHELL (if any).

i_1 = Element number

$i_2 \dots i_9$ = Joint number of the solid element

i_{10} = Last element number to be generated

i_{11} = Element number increment

i_{12} = Joint number increment

n = Number of times REPEAT is to be carried out

e_i = Element number increment

j_i = Joint number increment

Specify the four nodes of any of the faces of the solid element in a counter-clockwise direction as viewed from the outside of the element and then go to the opposite face and specify the four nodes of that face in the same direction used in specifying the nodes of the first face. The opposite face must be behind the first face, as defined by the right hand rule, i.e., the opposite (back) face points to the first (front) face, which points to the viewer.

Use **REPEAT ALL 0** to start a set of solids that will be repeated if you don't want to repeat back to the last REPEAT ALL.

Example

```
ELEMENT INCIDENCES SOLID
1 1 5 6 2 21 25 26 22 TO 3
4 21 25 26 22 41 45 46 42 TO 6
```

5.13.3 Surface Entities Specification

In order to facilitate rapid modeling of complex walls and slabs, a type of entity called **Surface** is available. At the modeling level, it corresponds to the entire structural part, such as a wall, floor slab or bridge deck. At the analysis level, it is first decomposed into a number of quadrilateral plate elements. Thus the Surface is a superelement for modeling purposes (it is composed from a number of plate

elements). Consequently, the user has the convenience of specifying only one large structural component per wall or slab, yet may maintain full control over the computational accuracy by setting the desired number of finite element divisions. Surfaces may include rectangular openings.

The attributes associated with the surface element, and the sections of this manual where the information may be obtained, are listed below:

Attributes	Related Sections
Surfaces Incidences	5.13.3
Openings in surface	5.13.3
Local Coordinates system for surfaces	1.6.3
Specifying sections for stress/force output	5.13.3
Property for surfaces	5.21.2
Material constants	5.26.3
Surface loading	5.32.3.4
Stress/Force output printing	5.42
Shear Wall Design	3.8.2, 5.55

General Format

SET DIVISION m

SURFACE INCIDENCE n1, ... , ni **SURFACE** s

DIVISION sd1, ... , sdj -

REOPENING x1 y1 z1 x2 y2 z2 x3 y3 z3 x4 y4 z4

DIVISION od1, ... , odk

Where:

m - number of segments to be generated between each pair of

Section 5 Commands and Input Instructions

5.13 Elements and Surfaces

adjacent nodes

n_1, \dots, n_i - node numbers defining the perimeter of the surface,

s - surface ordinal number,

sd_1, \dots, sd_j - number of divisions for each of the node-to-node distance on the surface perimeter,

$x_1 y_1 z_1 (\dots)$ - coordinates of the corners of the opening,

od_1, \dots, odk - divisions along edges of the opening.

The **SET DIVISION** command specifies a default number of generated meshing divisions for all node to node segments. In its absence, that number is taken as 10.

If the sd_1, \dots, sd_j or the od_1, \dots, odk list does not include all node-to-node segments, or if any of the numbers listed equals zero, then the corresponding division number is set to the default value of 10, or as previously input by the **SET DIVISION** command).

The **SURFACE INCIDENCES** command start the specifications of the elements. Commands **SUR 1** and **SUR 2** define Surface elements No. 1 and 2 with default boundary divisions and no openings. The **SUR 3** command defines Surface No. 3 with non-default edge divisions and one opening. The **DIV** command following **SUR 3** defines Surface element edge divisions. Non-default opening edge divisions are defined by the **DIV** command following the **RECO** command.

Notes

1. The surface definition must comprise a minimum of four nodal points forming corners of a rectangle. However, any number of additional nodes may be incorporated into the surface boundaries provided the nodes are collinear on edges they belong to. In addition, the user specifies the number of edge divisions that will be the basis for mesh generation. A single command per wall is used for this purpose. The program will subdivide all edges into the requested number of fragments and each of these fragments will become an edge of a plate element. However, if the original surface edges have additional nodal points between the corners, all node-to-node lengths of the surface edge will be divided into the same number of fragments.

2. Surface thickness and material constants are specified in a manner similar to that for plate elements, except that, currently, only a constant surface thickness is supported.
3. A support generation function allows quick assignment of support specifications to multiple nodal points.
4. Surface elements may be loaded by uniformly distributed loads in any global direction or by loads normal to the plane.
5. It is possible to obtain in-plane bending moments as well as stresses along any arbitrary line cutting the surface.

Example

```

SET DIVISION 12
SURFACE INCIDENCES
2 5 37 34 SUR 1
34 37 54 51 SUR 2
19 16 65 68 SUR 3 DIV 10 16 10 16 -
RECO 5.8 1.5 6.9 6.4 1.5 .6.9 6.4 0.5 6.9 5.8 0.5 6.9 DIV
5 10 5 10

```

This example illustrates definition of three Surface elements. The **SET DIVISION 12** command establishes a default number of boundary divisions for automatic mesh generation. This command will apply to outer edges of the elements as well as to the edges of openings, if present.

5.14 Plate Element Mesh Generation

There are several methods available in STAAD to model panel type entities like walls or slabs as an assembly of plate elements. This process is called meshing.

Two of those methods have a set of commands which can be provided in the STAAD input file. The first method, which is described in section 5.14.1 is based entirely on commands in the input file alone, and does not have any graphical interface for creation or modification.

Section 5 Commands and Input Instructions

5.14 Plate Element Mesh Generation

The second method is referred to as the Parametric mesh generator and is best used from STAAD's graphical screens. The aspect of this method, which enables commands to be written into the input file, is described in section 5.14.2.

5.14.1 Element Mesh Generation

This set of commands is used to generate finite element meshes. The procedure involves the definition of super-elements, which are subsequently divided into smaller elements.

Description

This is the second method for the generation of element incidences. If you need to divide a big element into a number of small elements, you may use this facility which generates the joint numbers and joint coordinates, the element numbers and the element incidences automatically. Use of this feature consists of two parts:

1. Definition of the super-element boundary points: A super-element may be defined by either 4 boundary points or 8 boundary points (see figure below). A boundary point is denoted by a unique alphabet (A-Z in upper case or a-z in lower case) and its corresponding coordinates. Hence, any 4 or 8 of the 52 characters may be used to define the super-element boundary. If 4 points are used to define the super-element, each side of the super-element will be assumed to have a straight edge connecting the 2 points defining that side. If 8 points are used, each side will be a smooth curve connecting the 3 points defining that side.
2. Generation of sub-elements: define the super-element using boundary points (4 or 8 as explained above) and specify the total number of sub-elements required.

General Format

```
DEFINE MESH  
Ai xi yi zi ( { CYL, RCYL } ( x0, y0, z0 ) )  
...  
Aj xj yj zj ( { CYL, RCYL } ( x0, y0, z0 ) )  
GENERATE ELEMENT { (QUADRILATERAL), TRIANGULAR }  
MESH Ai Aj ... n1 (n2)  
MESH Am An ... n3 (n4)
```

...

(up to 21 MESH input lines)

Where:

$A_i, A_j =$ Alphabets A - Z or alphabets a - z. That is max 52.

$x_i, y_i, z_i =$ Coordinates for boundary point A_i .

If **CYL** or **RCYL** is defined, above coordinates will be in cylindrical or reverse cylindrical coordinates system. Optional coordinates x_o, y_o and z_o will be the Cartesian coordinates for the origin of the cylindrical coordinates. Defaults to 0, 0, 0 if not provided. The 3 fields (x,y,z) may be replaced by a joint number whose coordinates have been defined in the **JOINT COORDINATE** command by entering A_i **JOINT jn** instead.

$A_i, A_j, A_k \dots =$ A rectangular super-element defined by four or eight boundary points.

$n_1 =$ Number of elements along the side A_i, A_j of the super-element.
(Must not exceed 28).

$n_2 =$ Number of elements along the side A_j, A_k of the super-element.
(Must not exceed 28).

If n_2 is omitted, that is, only n_1 is provided, then n_1 will indicate the total number of elements within the super-element. In this case, n_1 must be the square of an integer.

Limits

There is a limit of 21 Mesh commands. Up to 33000 joints may be generated and up to 67000 elements. Total number of joints in the model after this command is completed may not exceed 100,000.

Notes

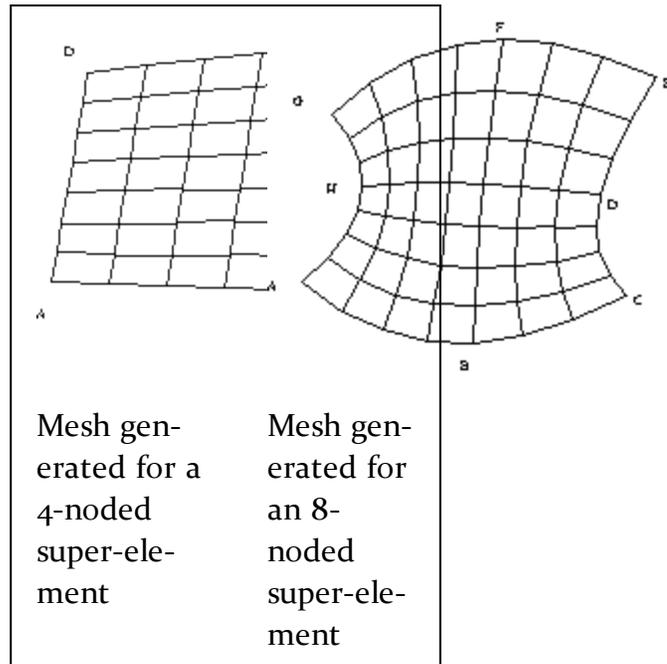
All coordinates are in current unit system. While using this facility you has to keep the following points in mind:

1. All super-elements must be 4-noded or 8-noded. Generated elements for 4-noded super-elements will retain the straight-line edges of the super-elements, while joints of elements generated from 8-noded super-elements will lie on a curved trajectory.

Section 5 Commands and Input Instructions

5.14 Plate Element Mesh Generation

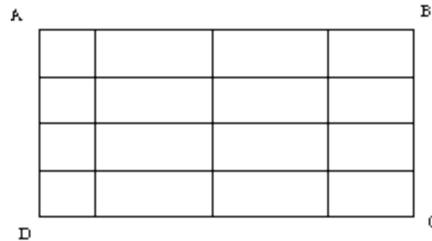
Figure 5-2: Mesh generation for super elements



2. Two super-elements, which have a common boundary, must have the same number of elements along their common boundary.
3. Sequence of super-elements - MESH commands define the super-elements. The sequence of this MESH command should be such that once one is defined, the next super-elements should be the ones connected to this. Therefore, for convenience, the first super-element should be the one, which is connected by the largest number of super-elements. In the example shown here for the tank, the bottom super-element is specified first.
4. This command must be used after the **MEMBER INCIDENCE** and **ELEMENT INCIDENCE** section and before the **MEMBER PROPERTIES** and **ELEMENT PROPERTIES** section. The elements that are created internally are numbered sequentially with an increment of one starting from the last member/element number plus one. Similarly the additional joints created internally are numbered sequentially with an increment of one starting from the last joint number plus one. It is advisable that users keep the joint numbers and member/element numbers in a sequence with an increment of one starting from one.
5. If there are members embracing a super-element which is being meshed, you must take care of the required additions/modifications in the **MEMBER INCIDENCE** section themselves since a few more new joints might appear

on the existing common boundary as a result of meshing the super-element. See the following figure:

Figure 5-3: Additional joints on a super element



Note: If a member exists between points A and B, the user must breakup this member into 4 parts. Members will not be meshed automatically.

6. The sub-elements will have the same direction (Clockwise or Anti-clockwise) as the super-elements. For a super-element bounded by four points A, B, C and D, if ABCD, BCDA etc. are in clockwise direction, CBAD or DCBA etc. are in anti-clock wise direction. If the particular super-element is denoted as ABCD, all the sub-elements in it will have a clockwise element incidence in this example.
7. Element incidences of the generated sub-elements may be obtained by providing the command 'PRINT ELEMENT INFORMATION' after the 'MESH...' command in the input file.
8. If the STAAD input file contains commands for **JOINT COORDINATES**, **MEMBER INCIDENCES**, **ELEMENT INCIDENCES**, and **MESH GENERATION**, they should be specified in the following order:

```

STAAD SPACE
UNIT . . .
JOINT COORDINATES
...
MEMBER INCIDENCES
...
ELEMENT INCIDENCES
...
DEFINE MESH

```

Section 5 Commands and Input Instructions

5.14 Plate Element Mesh Generation

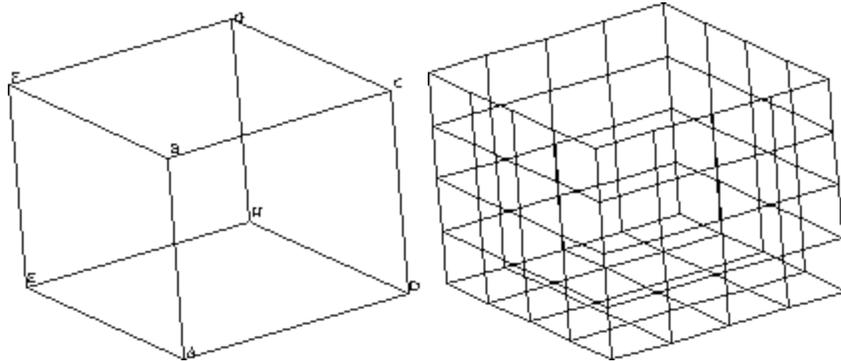
```
...  
GENERATE ELEMENT  
...
```

9. Newly created joints will be merged with existing joints if they are within 0.001 inches of each other.

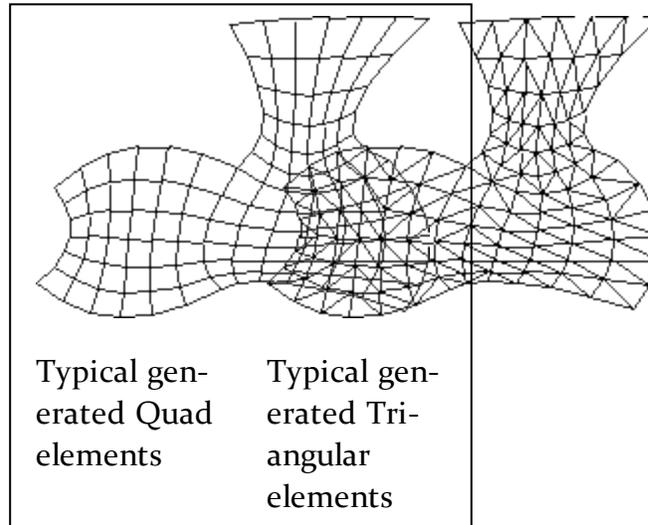
Example

The following section of input illustrates the use of **MESH GENERATION** facility, the user may compare this with the geometry inputs for Example Prob. No. 10 in the STAAD example manual:

Figure 5-4: Mesh generation used in Example Problem 10



```
STAAD SPACE TANK STRUCTURE WITH  
* MESH GENERATION  
UNIT . . .  
DEFINE MESH  
A 0 0 0 ; B 0 20 0 ; C 20 20 0  
D 20 0 0 ; E 0 0 -20 ; F 0 20 -20  
G 20 20 -20 ; H 20. 0. -20  
GENERATE ELEMENT  
MESH AEHD 16  
MESH EABF 16  
MESH ADCB 16  
MESH HEFG 16
```

MESH DHGC 16*Figure 5-5: Typical generated elements*

5.14.2 Persistency of Parametric Mesh Models in the STAAD Input File

There is a feature in STAAD.Pro's graphical model generation facilities called the Parametric Model. It is meant for creating a plate element mesh and is described in Section 2.2.2.5 of the STAAD Graphical Environment manual.

When a STAAD model has portions generated from a parametric mesh model, the parametric input (which is not otherwise part of the STAAD input data) is saved within a specially designated section of the STAAD input file. This gives you the flexibility to save mesh models at any time and make modifications at a later time, such as adding an opening or a density line.

It is important that this data not be modified or removed so as to preserve the parametric model. Any changes to the portions of the model marked between the <! and !> marks may have unintended consequences to the model.

Example

Special tag based commands have been introduced to support saving of parametric mesh models as part of the STAAD input file as shown below.

```
2072 1114 1113 1160; 2073 1045 1160 1113;
ELEMENT PROPERTY
```

Section 5 Commands and Input Instructions

5.14 Plate Element Mesh Generation

```
810 TO 1779 1821 TO 2073 THICKNESS 1
```

```
<! STAAD THIS ENTIRE SECTION OF THE INPUT FILE MUST NOT BE EDITED.  
PRO GENERATED DATA DO NOT MODIFY!!!
```

```
PARAMETRIC MODEL SECOND_FLOOR
```

```
MESH PARAM 0 3
```

```
MESH ORG 3 5 8
```

```
BOUNDARY 10
```

```
11 1 93 1 94 1 95 1 83 1 71 1 70 1 69 1 41 1 26 1
```

```
OPENING CIRC 72 360 96 43.2666 12
```

```
OPENING POLY 5
```

```
216 360 67.2 1 270 360 33.6 2 324 360 67.2 2 270 360  
100.8 2 216 360 100.8 2
```

```
DENSITY POINTS 2
```

```
180 360 168 1 360 360 168 1
```

```
DENSITY LINE 0 360 168 100 180 360 168 200
```

```
DENSITY LINE 180 360 168 1 360 360 168 1
```

```
DENSITY LINE 360 360 168 1 540 360 168 1
```

```
DENSITY LINE 180 360 0 1 180 360 168 1
```

```
DENSITY LINE 180 360 168 1 180 360 336 1
```

```
DENSITY LINE 360 360 0 1 360 360 168 1
```

```
DENSITY LINE 360 360 168 1 360 360 336 1
```

```
DENSITY LINE 54 360 302.4 1 162 360 201.6 1
```

```
DENSITY LINE 216 360 201.6 1 324 360 235.2 1
```

```
GENERATED PLATES ALL
```

```
END
```

```
<! STAAD PRO GENERATED DATA DO NOT MODIFY!!!
```

```
PARAMETRIC MODEL ROOF
```

```
MESH PARAM 60 3
```

```
MESH ORG 2 3 5
```

```
BOUNDARY 6
```

```
36 1 65 1 66 1 53 1 52 1 51 1
```

```

GENERATED PLATES ALL
END
!> END GENERATED DATA BLOCK
!> END GENERATED DATA BLOCK
DEFINE MATERIAL START
ISOTROPIC STEEL

```

5.15 Redefinition of Joint and Member Numbers

This command may be used to redefine **JOINT** and **MEMBER** numbers. Original **JOINT** and **MEMBER** numbers are substituted by new numbers.

General Format

```

SUBSTITUTE { { JOINT | MEMBER } { XRANGE | YRANGE | ZRANGE }
| COLUMN } f1, f2 START i

```

Where:

f_1 and f_2 are two range values of x, y, or z
 i is the new starting number.

Description

Joint and member numbers can be redefined in STAAD through the use of the **SUBSTITUTE** command. After a new set of numbers is assigned, input and output values will be in accordance with the new numbering scheme. You can design numbering schemes that will result in simple input specification as well as easy interpretation of results. For example, all joints in first floor of a building may be renumbered as 101, 102 ..., all second floor joints may be renumbered as 201, 202, etc.

Meaningful re-specification of **JOINT** and **MEMBER** numbers may significantly improve ease of interpretation of results.

This command may be in between incidence commands:

```

MEMBER INCIDENCE
SUBSTITUTE
ELEMENT INCIDENCE

```

Section 5 Commands and Input Instructions

5.16 Entities as Single Objects

Example

```
UNIT METER
SUBST JOINT YR  9.99  10.0 START  101
SUBST COLUMN START 901
```

Joints with Y coordinates ranging from 9.99 to 10 meters will have a new number starting from 101. Columns will be renumbered starting with the new number 901.

5.16 Entities as Single Objects

In the mathematical model, beams, columns, walls, slabs, block foundations, etc. are modeled using a collection of segments, which are known by the names members, plate elements, solid elements, etc. Hence, the bottom chord of a truss may be modeled using 5 members, with each member representing the segment between points where diagonals or vertical braces meet the bottom chord.

Often, it is convenient to cluster these segments under a single name so that assignment of properties, loads, design parameters, etc. is simplified. There are presently two options in STAAD for clustering entities - Group names and Physical members.

5.16.1 Listing of Entities (Members / Elements / Joints, etc.) by Specifying Groups

This command allows the user to specify a group of entities (e.g., members, joints, elements, etc.) and save the information using a 'group-name'. The 'group-name' may be subsequently used in the input file instead of a member/joint list to specify other attributes. This very useful feature allows avoiding of multiple specifications of the same member/joint list. Following is the general format required for the **GROUP** command.

General Format

```
START GROUP DEFINITION
```

followed by

```
(GEOMETRY)
```

```
_(group-name) member/element/solid-list
```

```
...
```

or

JOINT

_(group-name) joint-list

...

MEMBER

_(group-name) member-list

...

ELEMENT

_(group-name) element-list

SOLID

_(group-name) solid-list

...

FLOOR

_(group-name) member-list

...

followed by

END GROUP DEFINITION

Where:

joint-list = the list of joints belonging to the group. **TO**, **BY**, and **ALL** are permitted.

group-name = an alphanumeric name specified by the user to identify the group. The *group-name* must start with the '_' (underscore) character and is limited to 24 characters.

member-list = the list of members/joints belonging to the group **TO**, **BY**, **ALL**, **BEAM**, **PLATE**, and **SOLID** are permitted.

ALL means all members, plates, *and* solids; **BEAM** means all beams; **PLATE** all plates; and **SOLID** all solids.

Notes

1. The **GROUP** definition must start with the **START GROUP DEFINITION** command and end with the **END** command.
2. More than one **GROUP** name may be specified within the same definition

Section 5 Commands and Input Instructions

5.16 Entities as Single Objects

specification.

3. The words **JOINT**, **MEMBER**, **ELEMENT**, **FLOOR**, or **SOLID** may be provided if the you wish to identify the group name and lists with those specific items. However, if the group name and list is merely a means of grouping together more than one type of structural component under a single heading, the word **GEOMETRY** may be provided. In the absence of any of those words (**GEOMETRY**, **JOINT**, **MEMBER**, **ELEMENT**, **FLOOR**, or **SOLID**), the list is assumed to be that for **GEOMETRY**.
4. The same joint or member/element number may be included in up to four groups. Multiple definitions are useful for output but can be ambiguous for input data such as constants, section property, release, etc.
5. If two or more consecutively entered groups have the same name, then they will be merged. If not consecutive, the second entry of the same name will be ignored.
6. A member group may be used in lieu of a member-list with virtually any command which requires member lists, such as **MEMBER LOADS**, steel and concrete parameters, etc. There is one place however where a **MEMBER GROUP** will *not* suffice, and that is for defining panels during a **FLOOR LOAD** assignment.

In Section 5.32.4 of this manual, as explained under the topic “Applying floor load on members grouped under a FLOOR GROUP name”, a panel has to be specified using a **FLOOR GROUP**, not a **MEMBER GROUP**. A **FLOOR GROUP** is not accepted in lieu of a member-list for any other command.

7. The maximum number of group allowed in an input file is equal to the total number of member, plates, solid, and nodes times the number of duplicate entities allowed to be in different group, which is by default 10 (may be changed using the **SET GROUP DUPLICATE i** command in Section 5.5).

For example, if a model has 10 members, 3 plates and 2 solids, and 100 nodes. The maximum number of groups could be $10 \cdot (10 + 3 + 2 + 100) = 1150$.

Example 1

```
START GROUP DEFINITION
_TRUSS 1 TO 20 25 35
```

```

_BEAM 40 TO 50
END
MEMBER PROPERTIES
_TRUSS TA LD L40304
_BEAM TA ST W12X26

```

Example 2

```

START GROUP DEFINITION
JOINT
_TAGA 1 TO 10
MEMBER
_TAGB 40 TO 50
GEOMETRY
_TAGC 101 TO 135
END

MEMBER PROPERTIES
_TAGB TA LD L40304
_TAGC TA ST W12X26

```

5.16.2 Physical Members

STAAD allows grouping analytical predefined members into physical members using a special member group **PMEMBER**. This command defines a group of analytical, collinear members with same cross section and material property.

To model using **PMEMBER**, you need to model regular analytical members and then, group those together.

While creating a **PMEMBER**, the following pre-requisites apply:

1. Existence of the analytical members in the member-list.
2. Selected members should be interconnected.
3. The selected individual members must be collinear (adjacent analytical

Section 5 Commands and Input Instructions

5.16 Entities as Single Objects

members must lie within 5°).

4. Local axis of the individual members comprising the physical member should be identical (i.e., x, y and z are respectively parallel and in same sense).

Hint: You may select **Tools > Redefine Incidence** in the STAAD.Pro Graphical Environment for modifying analytical members which are pointing in the wrong direction

5. A member in one Physical Member Group should not be part of any other Physical Member Group.

Description

PMEMBER can be created either in the modeling mode or in the Steel Designer mode. Modeling mode and Steel Designer mode PMEMBERs will be labeled as M and D, respectively. Modeling mode PMEMBER will allow variable cross-sections. Steel Designer mode will allow importing of PMEMBERs created in the modeling mode.

To define a Physical Member, the following command is used after the **MEMBER INCIDENCE** command:

```
DEFINE PMEMBER  
{Member list} PMEMBER (pmember-no)
```

Example

```
JOINT COORDINATE  
1 0 0 0 6 10.0 0 0  
MEMBER INCIDENCE  
1 1 2 5  
DEFINE PMEMBER  
1 TO 5 PMEMB 1
```

To define the member property of a Physical Member, the following command is used:

```
PMEMBER PROPERTY  
{Pmember-list} PRIS ...
```

The Physical Member supports all types of member properties available in STAAD.

If multiple definitions of member properties for a particular analytical member is encountered (e.g., analytical member properties is defined twice, once via PMEMBER PROP command and again via the MEMBER PROP command, then the MEMBER PROP command will override the PMEMBER PROP definition.

To define the Material constants of a Physical Member, the following command is used:

```
PMEMBER CONSTANT
E CONCRETE pmember-list
DEN CONCRETE pmember-list
.....
```

Any member, which is a part of any PMEMBER is not allowed to be assigned constants explicitly.

A Physical Member can be loaded with Uniformly Distributed Load and Moment, Concentrated Load and Moment, and Trapezoidal Load. The command syntax is as follows:

```
PMEMBER LOAD
{Pmember List} UNI / CON / UMOM / UCON / TRAP f1 f2 f3 f4
```

Design parameters are available for use with PMEMBERS by using the PMEMB list. The following syntax is used:

```
{parameter} {value} PMEMB {Pmember-list}
```

Example

```
RATIO 1.05 PMEMB 1 2
```

Note: There is not option to specify **ALL** for a **PMEMB** list, except for **CODE CHECK** or **SELECT** commands.

After the analysis, the Post Analysis results of a PMEMBER can be seen by using the following command:

```
PRINT PMEMBER FORCE
```

This command will produce member forces for all the analytical members in the group.

5.17 Rotation of Structure Geometry

This command may be used to rotate the currently defined joint coordinates (and the attached members/elements) about the global axes. The rotated configuration is used for analysis and design. While specifying this command, the sense of the rotation should conform to the right hand rule.

General Format

```
PERFORM ROTATION *{ X d1 | Y d2 | Z d3 }
```

Where:

d₁, d₂, d₃ are the rotations (in degrees) about the X, Y, and Z global axes, respectively.

Example

```
PERFORM ROTATION X 20 Z -15
```

5.18 Inactive/Delete Specification

This set of commands may be used to temporarily inactivate or permanently delete specified **JOINTS** or **MEMBERS**.

General Format

```
INACTIVE { MEMBERS member-list | ELEMENTS element-list }  
DELETE { MEMBERS member-list | JOINTS joint-list }
```

Description

These commands can be used to specify that certain joints or members be deactivated or completely deleted from a structure. The **INACTIVE** command makes the members and elements temporarily inactive; the user must re-activate them during the later part of the input for further processing. The **DELETE** command will completely delete the members/elements from the structure; you cannot re-activate them. The Delete Joint command must be immediately after the Joint Coordinates. The **DELETE** commands must be provided immediately after all member/element incidences are provided and before any **INACTIVE** commands.

Notes

- a. The **DELETE MEMBER** command will automatically delete all joints associated with deleted members, provided the joints are not connected by any other active members or elements.
- b. This command will also delete all the joints, which were not connected to the structure in the first place. For example, such joints may have been generated for ease of input of joint coordinates and were intended to be deleted. Hence, if a **DELETE MEMBER** command is used, a **DELETE JOINT** command should not be used.
- c. The **DELETE MEMBER** command is applicable for deletion of members as well as elements. If the list of members to be deleted extends beyond one line, it should be continued on to the next line by providing a blank space followed by a hyphen (-) at the end of the current line.
- d. The **INACTIVE MEMBER** command cannot be used in situations where inactivating a member results in joints becoming unconnected in space.
- e. The inactivated members may be restored for further processes (such as an analysis or design for a 2nd set of load cases) by using the **CHANGE** command. See Section 5.37 and Example 4 for more information.
- f. The **DELETE MEMBER** command should be used to delete elements too. Specify the command as **DELETE MEMBER j**, where j is the element number of the element you wish to delete. In the example shown below, 29 to 34 and 43 are element numbers.
- g. Loads that have been defined on members declared as **INACTIVE** members will not be considered in the analysis. This applies to **SELFWEIGHT**, **MEMBER LOADS**, **PRESTRESS**, and **POSTSTRESS LOADS**, **TEMPERATURE LOADS**, etc.
- h. The **DELETE JOINT** command must be specified before all incidence commands such as **MEMBER INCIDENCE**, **ELEMENT INCIDENCE**, etc.

Example

```
INACTIVE MEMBERS 5 7 TO 10  
DELETE MEMBERS 29 TO 34 43
```

5.19 User Steel Table Specification

STAAD allows the user to create and use customized Steel Section Table (s) for Property specification, Code checking and Member Selection. This set of commands may be used to create the table(s) and provide necessary data.

General Format

```
START USER TABLE  
TABLE  $i_1$  ( $f_n$ )  
section-type  
section-name  
property-spec  
END
```

Where:

i_1 = table number (1 to 99). During the analysis process, the data in each user provided table is stored in a corresponding file with an extension .Uo?. For example, the data of the 5th table is stored in .Uo5. The first part of the input file name is the same as that of the STAAD input file. These files are located in the same working directory as the input file. Hence, they may later be used as external user provided tables for other input files.

f_n = vexternal file name containing the section name and corresponding properties (up to 72 characters).

section-type = a steel section name including: WIDE FLANGE, CHANNEL, ANGLE, DOUBLE ANGLE, TEE, PIPE, TUBE, GENERAL, ISECTION & PRISMATIC.

section-name = Any user designated section name, use 1 to 36 characters. First three characters of Pipes and Tubes must be PIP and TUB respectively. Only alphanumeric characters and digits are allowed for defining section names. (Blank spaces, asterisks, question marks, colon, semi-colon etc. are not permitted.)

property-spec = Properties for the section. The requirements are different for each section type as follows. Shear areas AY and AZ must be provided to ensure proper shear stress or shear strength calculations during design.

The default length units for properties are the current units. If UNIT command is entered within the User Table in the input file then those units become the current units. However, a UNIT command on an external file only affects that file and has no effect on the units in subsequent input file commands. The user may specify the desired length unit by using the UNIT command as the first command in the table (see example following this description).

If data is from input file, then use up to 3 lines of input per property-spec (end all but last with a hyphen, -). If data is from external file, then use only one line, but it can be up to 250 characters long.

Example

```

START USER TABLE
TABLE 1
UNIT INCHES KIP
WIDE FLANGE
P24X55-ABCDEFGHIJKLMNPOQRSTUVWXYZ111
16.2 23.57 0.375 7.005 0.505 1350 29.1 1.00688 8.83875
7.07505
P24X56
18.3 20.99 .4 8.24 .615 1330 57.5 1.83 0.84 7.0
MEMBER PROPERTY
27 UPTABLE 1 P24X55-ABCDEFGHIJKLMNPOQRSTUVWXYZ111
39 UPTABLE 1 P24X56

```

5.19.1 Wide Flange

AX

Cross section area

D

Depth of the section

TW

Thickness of web

WF

Width of the top flange (or both flanges when **WF1** is not specified)

Section 5 Commands and Input Instructions

5.19 User Steel Table Specification

TF

Thickness of top flange (or both flanges when **WF1** is not specified)

IZ

Moment of inertia about local z-axis (usually strong axis)

IY

Moment of inertia about local y-axis

IX

Torsional constant

AY

Shear area in local y-axis. If zero, shear deformation is ignored in the analysis

AZ

Same as above except in local z-axis

WF1

Width of the bottom flange

TF1

Thickness of bottom flange

The following option parameters are used to include a composite concrete slab. If included, these must be on a separate line following a dash, -, line end on the end of the required section parameters.

CFR

Width of the composite slab to the left of the web center line

CFL

Width of the composite slab to the right of the web center line

CFT

Thickness of the composite slab

MR

Modular ratio of the concrete in the composite slab

The following option parameters are used to include a bottom flange cover plate.

Hint: Bottom cover plates can only be added in association with a composite slab.

BPR

Width of the additional bottom flange plate to the left of the web center line

BPL

Width of the additional bottom flange plate to the right of the web center line

BPT

Thickness of the additional bottom flange plate

Example

```

START USER TABLE
TABLE 1
UNIT MMS
WIDE FLANGE
UNEQUAL_FLANGE_I
16855 600 10 405 15 1.10087E+009 1.21335E+008 1.13626E+006 6000 7450 300
17
UNEQUAL_FLANGE_COMP_I
-16855 600 10 405 15 1.10087E+009 1.21335E+008 1.13626E+006 6000 7450 -
300 17
250 350 75 9.1
UNEQUAL_FLANGE_COMP_BOTPLT_I
-16855 600 10 405 15 1.10087E+009 1.21335E+008 1.13626E+006 6000 7450 -
300 17
-250 350 75 9.1
120 100 25
END

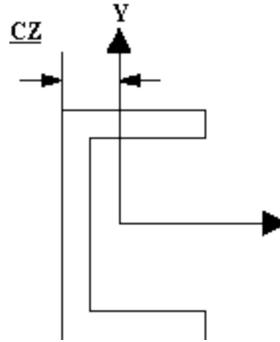
```

5.19.2 Channel

Figure 5-6: Channel section

Section 5 Commands and Input Instructions

5.19 User Steel Table Specification



AX

Cross section area

D

TW

WF

TF

IZ

IY

IX

CZ

AY

AZ

5.19.3 Angle

D

Depth of angle

WF

Width of angle

TF

Thickness of flanges

R

Radius of gyration about principal axis, shown as $r(Z-Z)$ in the AISC manual (this must not be zero)

AY

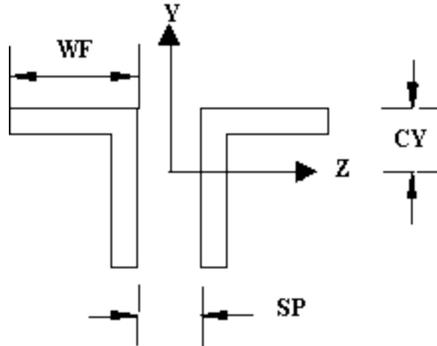
Shear area long Y axis

AZ

Shear area along Z axis

5.19.4 Double Angle

Figure 5-7: Double angle section



D

WF

TF

SP

IZ

IY

IX

CY

AY

AZ

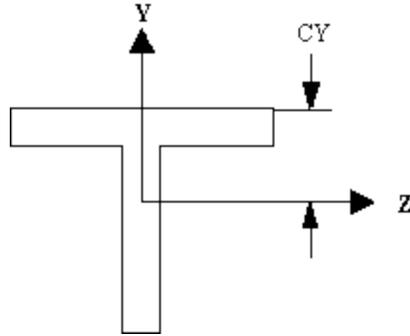
Note: The parameter **RVV** is defined as the radius of gyration about the minor principal axis for single angles - z-z axis for “TA ST” angles, and y-y axis for “TA RA” angles.

5.19.5 Tee

Figure 5-8: Tee section

Section 5 Commands and Input Instructions

5.19 User Steel Table Specification



AX

D

WF

TF

TW

IZ

IY

IX

CY

AY

AZ

5.19.6 Pipe

OD

Outer diameter

ID

Inner diameter

AY

AZ

5.19.7 Tube

AX

D

WF

TF

IZ

IY

IX

AY

AZ

5.19.8 General

The following cross-sectional properties should be used for this section-type. This facility allows the user to specify a built-up or unconventional steel section.

Provide both the Y and Z parameters for design or code checking.

AX

Cross section area

D

Depth of the section

TD

Thickness associated with section element parallel to depth (usually web).
To be used to check depth/thickness ratio

B

Width of the section

TB

Thickness associated with section element parallel to flange. To be used to
check width/thickness ratio

IZ

Moment of inertia about local z-axis

IY

Moment of inertia about local y-axis

IX

Torsional Constant

SZ

Section modulus about local z-axis

Section 5 Commands and Input Instructions

5.19 User Steel Table Specification

SY

Section modulus about local y-axis

AY

Shear area for shear parallel to local y-axis

AZ

Shear area for shear parallel to local z-axis

PZ

Plastic modulus about local z-axis

PY

Plastic modulus about local y-axis

HSS

Warping constant for lateral torsional buckling calculations

DEE

Depth of web. For rolled sections, distance between fillets should be provided

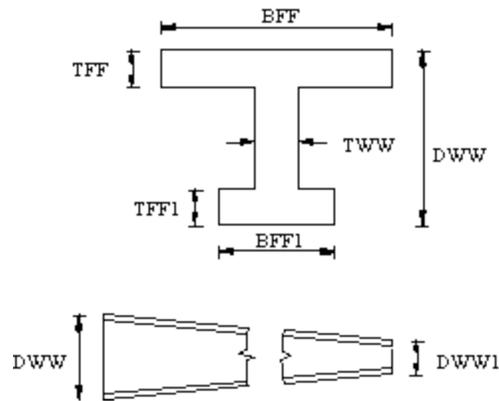
Note: Properties PZ, PY, HSS and DEE must be provided for code checking/member selection per plastic and limit state based codes (AISC LRFD, British, French, German and Scandinavian codes). For codes based on allowable stress design (AISC-ASD, AASHTO, Indian codes), zero values may be provided for these properties.

Note: STAAD.Pro can perform some code checks on a General section where as a Prismatic section (Section 5.20.2) is for analysis only.

5.19.9 I Section

This section type may be used to specify a generalized I-shaped section. The cross-sectional properties required are listed below. This section can be used to specify tapered I-shapes.

Figure 5-9: I section and tapered web

**DWW**

Depth of section at start node

TWW

Thickness of web

DWW₁

Depth of section at end node

BFF

Width of top flange

TFF

Thickness of top flange

BFF₁

Width of bottom flange

TFF₁

Thickness of bottom flange

AYF

Shear area for shear parallel to Y-axis (see following notes)

AZF

Shear area for shear parallel to Z-axis (see following notes)

XIF

Torsional constant (IX or J) (see following notes)

Section 5 Commands and Input Instructions

5.19 User Steel Table Specification

Notes

DWW should never be less than DWW₁. Therefore, you must provide the member incidences accordingly.

You are allowed the following options for the values A_{YF}, A_{ZF} and X_{IF}.

If positive values are provided, they are used directly by the program.

If zero is provided, the program calculates the properties using the following formula.

$$A_{YF} = D \times TWW$$

Where:

D = Depth at section under consideration

$$A_{ZF} = 0.66 ((BFF \times TFF) + (BFF1 \times TFF1))$$

$$X_{IF} = 1/3 ((BFF \times TFF^3) + (DEE \times TWW^3) + (BFF1 \times TFF1^3))$$

Where:

DEE = Depth of web of section

If negative values are provided, they are applied as factors on the corresponding value(s) calculated by the program using the above formula. The factor applied is always the absolute of the value provided, i.e., if the user provides the value of X_{IF} as -1.3, then the program will multiply the value of X_{IF}, calculated by the above formula, by a factor of 1.3.

5.19.10 Prismatic

The property-spec for the PRISMATIC section-type is as follows:

AX

Cross-section area

IZ

Moment of inertia about the local z-axis

IY

Moment of inertia about the local y-axis

IX

Torsional constant

AY

Shear area for shear parallel to local y-axis.

AZ

Shear area for shear parallel to local z-axis.

YD

Depth of the section in the direction of the local y-axis.

ZD

Depth of the section in the direction of the local z-axis

Note: When listing multiple shapes, those section names must be provided in ascending order by weight since the member-selection process uses these tables and the iteration starts from the top.

Example

```

START USER TABLE
TABLE 1
UNIT . . .
WIDE FLANGE
W14X30
8.85 13.84 .27 6.73 .385 291. 19.6 .38 0 0
W21X50
14.7 20.83 .38 6.53 .535 984 24.9 1.14 7.92 0
W14X109
32. 14.32 .525 14.605 .86 1240 447 7.12 7.52 0
TABLE 2
UNIT . . .
ANGLES
L25255
2.5 2.5 0.3125 .489 0 0
L40404

```

Section 5 Commands and Input Instructions

5.19 User Steel Table Specification

```
4. 4. .25 .795 0 0  
END
```

5.19.11 Using Reference Table Files

The above example can also be input as follows:

```
START USER TABLE  
TABLE 1 TFILE1  
TABLE 2 TFILE2  
END
```

Where TFILE1 and TFILE2 are names of files which must be created prior to running STAAD, and where the file TFILE1 will contain the following:

```
UNIT . . .  
WIDE FLANGE  
W14X30  
8.85 13.84 .27 6.73 .385 291. 19.6 .38 0 0  
W21X50  
14.7 20.83 .38 6.53 .535 984 24.9 1.14 7.92 0  
W14X109  
32. 14.32 0.525 14.605 .86 1240 447 7.12 7.52 0
```

and the file TFILE2 will contain:

```
UNIT . . .  
ANGLES  
L25255  
2.5 2.5 .3125 .489 0 0  
L40404  
4. 4. .25 .795 0 0
```

Note: The User-Provided Steel Table(s) may be created and maintained as separate file(s). The same files may be used for all models using sections from these tables. These files should reside in the same directory where the input file is located. On each file the first table should contain a UNITS command.

5.20 Member Property Specification

This set of commands may be used for specification of section properties for frame members.

The options for assigning properties come under two broad categories:

1. Those which are specified from built-in property tables supplied with the program, such as for steel, aluminum and timber.
2. Those which are *not* assigned from built-in tables, but instead are specified on a project-specific basis, such as for concrete beams and columns, or custom-made sections for industrial structures.

Properties which are specified from built-in property tables

1. General format for standard steel (hot rolled):

```
MEMBER PROPERTIES { AMERICAN | AUSTRALIAN | BRITISH |
CANADIAN | CHINESE | DUTCH | EUROPEAN | FRENCH | GERMAN
| INDIAN | JAPANESE | KOREAN | MEXICAN | RUSSIAN
| SAFRICAN | SPANISH | VENEZUELAN }
member-list { TABLE type-spec section-name-in-table
(additional-spec) | ASSIGN profile-spec }
```

AMERICAN, BRITISH, EUROPEAN (etc.) option will instruct the program to read properties from the appropriate steel table. The default depends on the country of distribution.

- See "Assigning Properties from Steel Tables" on page 341 for type-specs and additional-specs.
- See "Assign Profile Specification" on page 350 for **ASSIGN** profile-spec.
- Section 2 of this manual and the sections on steel design for various countries in the International Codes manual contain information on the section types which can be assigned for the various countries named in the above list.

Section 5 Commands and Input Instructions

5.20 Member Property Specification

- See "Examples of Member Property Specification" on page 351 for examples.

The **MEMBER PROPERTY** command may be extended to multiple lines by ending all lines but the last with a space and hyphen (-).

2. General format for cold formed steel:

```
MEMBER PROPERTIES { BUTLER | COLD AMERICAN | COLD  
BRITISH | COLD INDIAN | KINGSPAN | LYSAGHT | RCECO }  
member-list TABLE ST section-name-in-table
```

Section 2 of this manual and the sections on steel design for various countries in the International Codes manual contain information on the section types which can be assigned for the various countries/organizations named in the above list.

3. General format for steel joist:

```
MEMBER PROPERTIES SJIJOIST  
member-list TABLE ST section-name-in-table
```

Section 1 of this manual contains information on the joist types which can be assigned from the Steel Joist Institute's tables.

4. General format for Aluminum:

```
MEMBER PROPERTIES ALUMINUM  
member-list TABLE ST section-name-in-table
```

The section on aluminum design in the International Codes manual contain information on the section types which can be assigned for the aluminum table in the above list.

5. General format for Timber:

```
MEMBER PROPERTIES { AITC | TIMBER CANADIAN }  
member-list TABLE ST section-name-in-table
```

Section 4 of this manual and the sections on timber design in the International Codes manual contain information on the section types which can be assigned for the above list.

Properties that are *not* specified from built-in property tables

```
MEMBER PROPERTIES  
member-list { PRISMATIC property -spec | TAPERED argument-  
list | UPTABLE i1 section-name }
```

- See "Prismatic Property Specification" on page 346 for specification of **PRISMATIC** properties.
- See "Tapered Member Specification" on page 349 for specification of **TAPERED** members.
- See "Property Specification from User Provided Table" on page 350 for specification from **UPTABLES**.
- See "Examples of Member Property Specification" on page 351 for examples.

The **MEMBER PROPERTY** command may be extended to multiple lines by ending all lines but the last with a space and hyphen (-).

5.20.1 Assigning Properties from Steel Tables

The following commands are used for specifying section properties from built-in steel table(s). The section type specification is followed by additional specifications as needed.

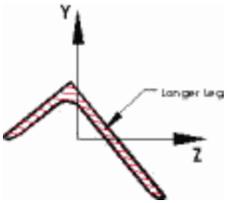
General Format

type-spec . table-name additional-spec.

type-spec = { ST | RA | D | LD | SD | T | CM | TC | BC | TB
| FR }

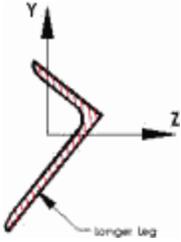
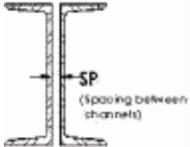
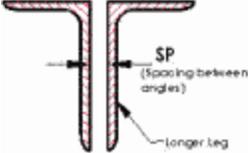
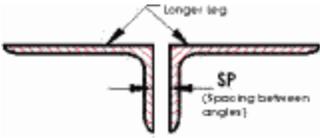
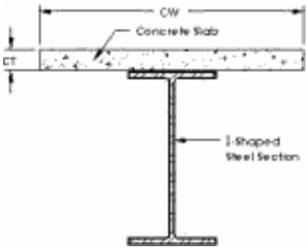
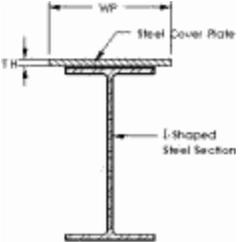
Where:

Table 5-3: Type-specs for various profile types

type-spec	Description	Diagram
ST	single section from the standard built-in tables	

Section 5 Commands and Input Instructions

5.20 Member Property Specification

type-spec	Description	Diagram
RA	single angle with reverse Y-Z axes (see Section 1.5.2)	
D	double profile. In the case of channels, back-to-back	
LD	double angle with long legs back-to-back	
SD	double angle with short legs back-to-back	
T	tee section cut from I shaped section	
CM	composite section, available for I shaped sections	
TC	section with top cover plate	

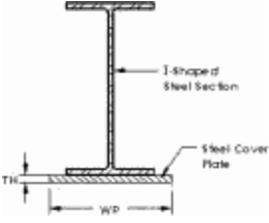
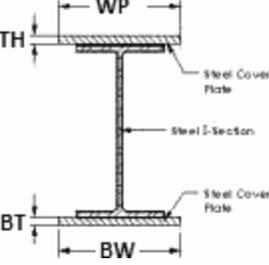
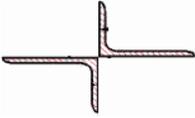
type-spec	Description	Diagram
BC	section with bottom cover plate	
TB	section with both top and bottom cover plate top plate dimensions are described using WP and TH parameter values and bottom plate dimensions are described using BW and BT parameter values	
FR	front-to-front (i.e., toe-to-toe) channels. Note: Spacing between the channels must be provided using the SP option mentioned in the additional spec specification described below.	
SA	double angle in a star arrangement (heel to heel)	

table-name = Table section name like W8X18, C15X33 etc.

The documentation on steel design per individual country codes contains information regarding their steel section specification also. For details on specifying sections from the American steel tables, see Section 2.2.1 of this manual.

Section 5 Commands and Input Instructions

5.20 Member Property Specification

additional-spec = * {SP f1 | WP f2 | TH f3 | WT f4 | DT f5 |
OD f6 | ID f7 | CT f8 | FC f9 | CW f10 | CD f11 | BW f12
| BT f13 }

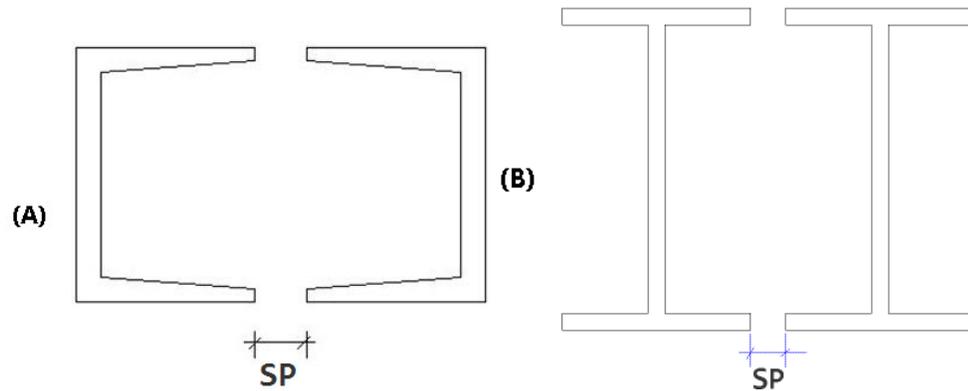
Where:

Table 5-4: Additional specifications for steel sections

Variable	Description
f1	Spacing between double angles or double channels (default is 0.0) Rib height for composite sections
f2	Width of top cover plate for I shaped sections with cover plate.
f3	Thickness of plates or tubes (top plate for I shaped sections)
f4	Width of tube.
f5	Depth of tube.
f6	Outer diameter of pipe.
f7	Inside diameter of pipe.
f8	Concrete thickness of the concrete for composite sections.
f9	Compressive strength of concrete for composite sections.
f10	Concrete width for composite sections.
f11	Concrete density for composite sections (default is 150 pounds per cubic ft.)
f12	Width of bottom cover plate for I shaped sections with cover plates
f13	Thickness of cover plates for I shaped sections with cover

See "Built-In Steel Section Library" on page 40 for more information.

Figure 5-10: Spacing for (A) FR double channels and (B) double wide-flange sections



See "Examples of Member Property Specification" on page 351 for an example.

Notes

All values f_{1-9} must be supplied in current units.

Some important points to note in the case of the composite section are:

1. The **CM** parameter can be assigned to I-shaped sections only. A **CM** (composite) section is one obtained by considering a portion of a concrete slab to act in unison with the I shaped steel section. FC is the strength or grade of concrete used in the slab. In the USA, FC is called the specified compressive strength of concrete. Typical values of FC range between 2.0 and 5.0 ksi, and 20 to 50 Mpa.
2. The width of the concrete slab (**CW**) (if not entered) is assumed to be the width of the top flange of the steel section + 16 times the thickness of the slab.
3. In order to calculate the section properties of the cross-section, the modular ratio is calculated assuming that:

$$E_s = \text{Modulus of elasticity of steel} = 29000 \text{ Ksi.}$$

$$E_c = \text{Modulus of elasticity of concrete} = 1802.5\sqrt{F_c} \text{ Ksi}$$

Where F_c (in Ksi) is defined earlier.

Some other general notes on this subject of member property designations are :

4. The **T** parameter stands for a T-shaped section obtained by cutting an I-shaped section at exactly its mid height level along the web. Hence, the area of a T shape is exactly half the area of the corresponding I shape. The depth of a T shape is half the depth of the I shape it was cut from.

Section 5 Commands and Input Instructions

5.20 Member Property Specification

What we refer to as I shaped sections are sections which look like the English alphabet I. The American Wide Flange, the British UB and UC sections, Japanese H sections, etc., all fall under this category. Consequently, the "T" shape cut from a Japanese H shape is one obtained by cutting the H shape at exactly its mid-height level of the web.

Not all I shaped sections have a corresponding T. This may be inferred by going through the section libraries of individual countries and organizations. In such cases, if a user were to specify such a T section, the program will terminate with the message that the section does not exist.

5. Steel Cover plates also can be added only to I shaped sections. Thus, the **TC**, **BC**, and **TB** are not applicable to any shape other than an I shape.

5.20.2 Prismatic Property Specification

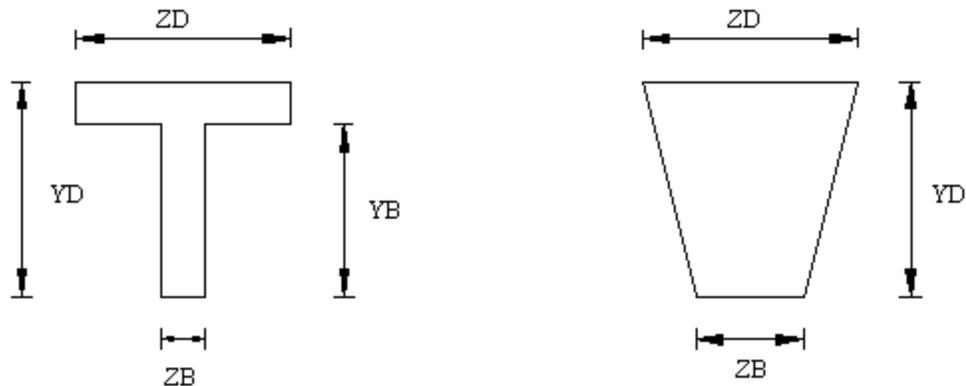
The following commands are used to specify section properties for prismatic cross-sections.

General Format

For the **PRISMATIC** specification, properties are provided directly (End each line but last with a hyphen "-") as follows:

```
property-spec = * { AX f1 | IX f2 | IY f3 | IZ f4 | AY f5 |  
AZ f6 | YD f7 | ZD f8 | YB f9 | ZB f10 }
```

Figure 5-11: Prismatic property nomenclature for a T and Trapezoidal section



Where:

AX f_1 = Cross sectional area of the member. Set to zero for TEE, Rectangular, Trapezoid, or circular.

IX f_2 = Torsional constant.

IY f_3 = Moment of inertia about local y-axis.

IZ f_4 = Moment of inertia about local z-axis (usually major).

AY f_5 = Effective shear area in local y-axis.

AZ f_6 = Effective shear area in local z-axis.

If any of the previous six parameters are omitted, it will be calculated from the YD, ZD, YB, and/or ZB dimensions.

YD f_7 = Depth of the member in local y direction. Used as the diameter of section for circular members.

ZD f_8 = Depth of the member in local z direction. If ZD is not provided and YD is provided, the section will be assumed to be circular.

YB f_9 = Depth of stem for T-section.

ZB f_{10} = Width of stem for T-section or bottom width for TRAPEZOIDAL section.

The values that STAAD calculates for the omitted terms can be obtained by specifying the command **PRINT MEMBER PROPERTIES**.

The values of many of the derived properties like shear areas (AY, AZ), section moduli (SY, SZ), etc. will be shown in the output file.

This command can be used regardless of the manner in which the properties are specified (e.g., PRISMATIC, user table, built-in table).

5.20.2.1 Prismatic Tapered Tube Property Specification

The following commands are used to specify section properties for prismatic tapered tube cross-sections. For the property types shown below, additional information can be obtained from Table 2.1 of the ASCE 72 document, 2nd edition.

General Format

property-spec = * { ROUND | HEXDECAGONAL | DODECAGONAL | OCTAGONAL | HEXAGONAL | SQUARE } START d_1 END d_2 THICK t

Where:

d_1 = Depth of section at start of member.

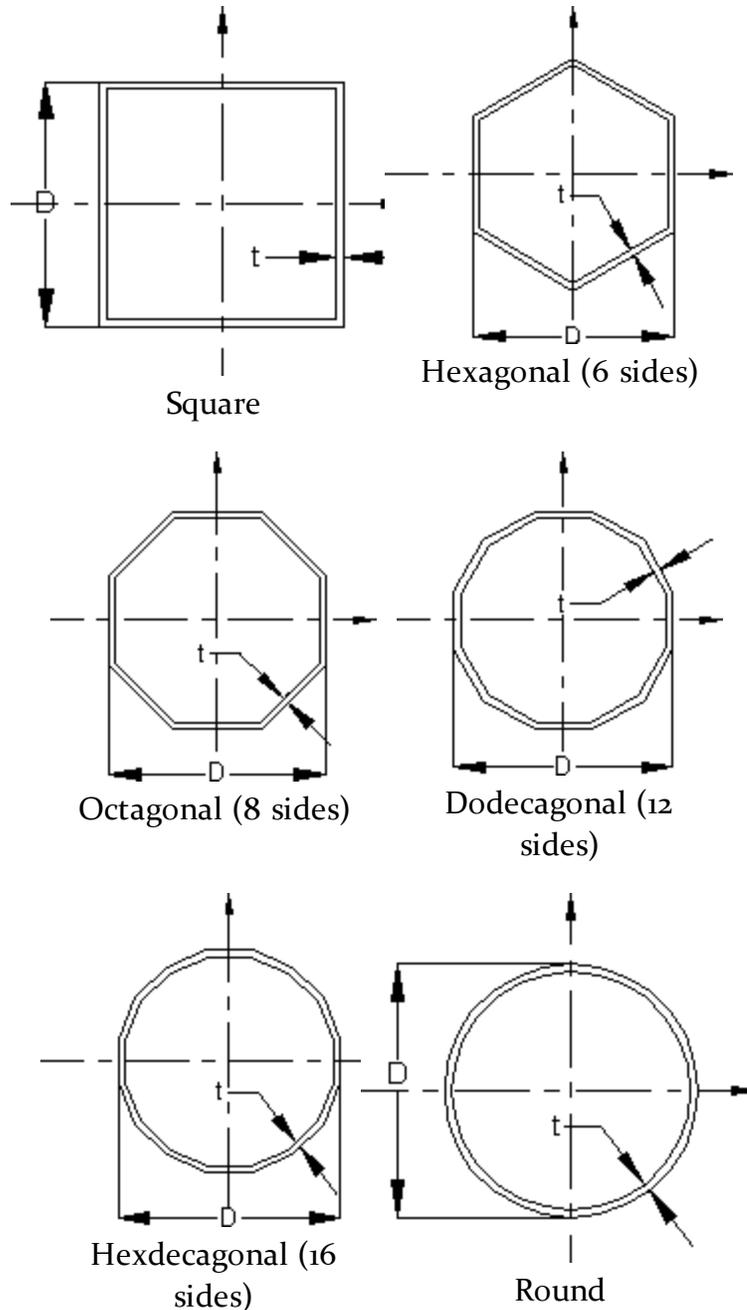
d_2 = Depth of section at end of member.

Section 5 Commands and Input Instructions

5.20 Member Property Specification

t = Thickness of section (constant throughout the member length).

Figure 5-12: Prismatic tapered tube shapes



Note: Section properties are calculated using the rules applicable for thin-walled sections.

Example

```

UNIT ...
MEMBER PROPERTY
1 PRIS ROUND STA 10 END 8 THI 0.375
2 PRIS HDC      STA 15 END 10 THI 0.375
3 PRIS DOD      STA 12 END 12 THI 0.375

```

5.20.3 Tapered Member Specification

The following commands are used to specify section properties for tapered I-shapes.

General Format

argument-list = f_1 f_2 f_3 f_4 f_5 (f_6 f_7)

Where:

f_1 = Depth of section at start node.

f_2 = Thickness of web.

f_3 = Depth of section at end node.

f_4 = Width of top flange.

f_5 = Thickness of top flange.

f_6 = Width of bottom flange. Defaults to f_4 if left out.

f_7 = Thickness of bottom flange. Defaults to f_5 if left out

Example

```

MEMBER PROPERTY
1 TO 5 TAPERED 13.98 0.285 13.98 6.745 .455 6.745
.455

```

Section 5 Commands and Input Instructions

5.20 Member Property Specification

Notes

- All dimensions (f_1, f_2, \dots, f_7) should be in current units.
- f_1 (Depth of section at start node) should always be greater than f_3 (Depth of section at end node). You must provide the member incidences accordingly.

5.20.4 Property Specification from User Provided Table

The following commands are used to specify section properties from a previously created **USER-PROVIDED STEEL TABLE**.

General Format

member-list U**TABLE** I_1 **section-name**

Where:

U**TABLE** stands for User-Provided **TABLE**

i_1 = table number as specified previously (1 to 20)

section-name = section name as specified in the table.

See "Examples of Member Property Specification" on page 351 for an example.

5.20.5 Assign Profile Specification

The **ASSIGN** command may be used to instruct the program to assign a suitable steel section to a frame member based on the profile-spec shown below.

General Format

profile-spec = { B**EA**M | C**OLU**MN | C**HAN**NEL | A**NG**LE (DOUBLE) }

See "Examples of Member Property Specification" on page 351 for an example.

Note: Sections are always chosen from the relevant built-in steel table. To find out the details of the sections that are chosen, the command **PRINT MEMBER PROPERTIES** should be provided after specification of all member properties. These commands may not work with certain tables such as cold-formed steel, Timber, Aluminum, and even some of the standard steel tables.

5.20.6 Examples of Member Property Specification

This section illustrates the various options available for **MEMBER PROPERTY** specification.

```

UNIT ...
MEMBER PROPERTIES
1 TO 5 TABLE ST W8X31
9 10 TABLE LD L40304 SP 0.25
12 TO 15 PRISMATIC AX 10.0 IZ 1520.0
17 18 TA ST PIPE OD 2.5 ID 1.75
20 TO 25 TA ST TUBE DT 12. WT 8. TH 0.5
27 29 32 TO 40 42 PR AX 5. IZ 400. IY 33. -
IX 0.2 YD 9. ZD 3.
43 TO 47 UPT 1 W10X49
50 51 UPT 2 L40404
52 TO 55 ASSIGN COLUMN
56 TA TC W12X26 WP 4.0 TH 0.3
57 TA CM W14X34 CT 5.0 FC 3.0

```

This example shows each type of member property input. Members 1 to 5 are wide flanges selected from the AISC tables; 9 and 10 are double angles selected from the AISC tables; 12 to 15 are prismatic members with no shear deformation; 17 and 18 are pipe sections; 20 to 25 are tube sections; 27, 29, 32 to 40, and 42 are prismatic members with shear deformation; 43 to 47 are wide flanges selected from the user input table number 1; 50 and 51 are single angles from the user input table number 2; 52 through 55 are designated as COLUMN members using the ASSIGN specification. The program will assign a suitable I-section from the steel table for each member.

Member 56 is a wideflange W₁₂X₂₆ with a 4.0 unit wide cover plate of 0.3 units of thickness at the top. Member 57 is a composite section with a concrete slab of 5.0 units of thickness at the top of a wide flange W₁₄X₃₄. The compressive strength of the concrete in the slab is 3.0 force/length².

5.20.7 Composite Decks

A composite deck generation facility is available in the program.

Section 5 Commands and Input Instructions

5.20 Member Property Specification

General Format

The command syntax for defining the deck within the STAAD input file is as shown below.

```
START DECK DEFINITION  
_DECKdeck-name  
PERIPHERYmember-list  
DIRECTIONd1 d2 d3  
COMPOSITEmember-list  
OUTERmember-list  
VENDOR name  
FCf1  
CTf2  
CDf3  
RBHf4  
RBWf5  
PLTf6  
PLWf7  
DIAf8  
HGTf9  
DR1f10  
SHRf11  
CMPf12  
CW ;f13MEMB cw-member-list  
END DECK DEFINITION
```

Where:

deck-name = an alphanumeric name specified by the user to identify the deck. The deck-name line must start with '_DEC'. The deck-name is the second word and is limited to 23 characters. This name must not be the same as any group name.

member-list = the list of members belonging to the deck. TO, BY, ALL, and BEAM are permitted. ALL means all members in structure; BEAM means all beams.

d_1 = x component of the direction of the deck.

d_2 = y component of the direction of the deck.

d_3 = z component of the direction of the deck.

The following parameters may be in any order or omitted. They only apply to the composite members listed above. Do not enter a member list for these parameters.

f_1 = compressive strength of the concrete for all composite members listed above for this composite deck.

f_2 = concrete thickness.

f_3 = concrete density.

f_4 = height of rib of form steel deck. This is the distance from the top of the I beam to the bottom of the concrete deck.

f_5 = width of rib of form steel deck.

f_6 = thickness of cover plate welded to bottom flange of composite beam.

f_7 = width of cover plate welded to bottom flange of composite beam.

f_8 = diameter of shear connectors.

f_9 = height of shear connectors after welding.

f_{10} = ratio of moment due to dead load applied before concrete hardens to the total moment.

f_{11} = temporary shoring during construction.

0 = no shoring

1 = with shoring

f_{12} = composite action with connectors.

0 = no composite action in design

1 = composite action

2 = ignore positive moments during design

The following parameter may be specified by member list. They only apply to the composite members listed above.

Section 5 Commands and Input Instructions

5.20 Member Property Specification

f_{13} = concrete width for each composite member listed. cw-member-list = the list of composite members in this deck that have this width. Enter as many CW lines as necessary to define the width of all composite members of this deck.

This Deck definition data should be entered after the member properties have been entered.

Notes

1. The DECK definition must start with the START DECK DEFINITION command and end with the END command.
2. More than one DECK may be specified between the START and END.
3. The same member number may be included in up to 4 deck/groups. Multiple definitions are useful for output but can be ambiguous for input data such as constants, section property, release, etc.
4. If two or more consecutively entered decks have the same name, then they will be merged. If not consecutive, the second entry of the same name will be ignored.
5. The _deck-name must be unique within the Deck definitions and the Group definitions.
6. PER, DIR, OUT are data created by the GUI. Do not edit this data.
7. This Deck definition data should be entered after the member properties have been entered.

Example

```
START DECK DEFINITION
_DECK DEC-1
PERIPHERY 4 1640 18 38 56 50 49
DIRECTION 0.000000.000000 -1.000000
COMPOSITE 41 74 38
OUTER 7 8 3130
VENDOR USSTEEL
DIA 0.700
```

```

HGT 2.75
CT 11.0
FC 3.1
RBW 2.6
RBH 0.1
CMP 1.0
SHR 1
CD 0.0000870
CW 123.000000 MEMB41
CW 123.000000 MEMB7
CW 61.500000 MEMB4
CW 61.500000 MEMB38
END DECK DEFINITION

```

See "Composite Beams and Composite Decks" on page 45 for additional details.

5.20.8 Curved Member Specification

The following commands are used to specify that a member is curved. The curve must be a segment of a circle and the internal angle subtended by the arc must be less than 180 degrees. Any non-tapered cross-section is permitted.

General Format

MEMBER CURVED

member-list RADIUS r GAMMA g PRESS p

Where:

r = radius in length units

g = The angle in degrees used to define the plane of the circle. The angle is defined using the same set of rules used to define the orientation (beta angle) of a straight member connected between the two nodes.

p = Pressure/Flexibility parameter for pipe bends. See notes.

Section 5 Commands and Input Instructions

5.20 Member Property Specification

Notes

1. The radius should be in current units.
2. Certain attributes like releases, **TENSION/COMPRESSION** flags, and several member load types are currently not available. Section forces too are currently not available.
3. The design of curved members is not supported.

Pressure/Flexibility Parameter

This applies only to pipe bend (elbow) members (OD and ID entered). These members will flex more due to ovalization depending on internal pressure. The ASME Boiler and Pressure Vessel Code, Section III, NB-3687.2, 1971, for Class I components is used to calculate the flexibility reduction factor.

- Set $p = 0$ or omit for this flexibility increase calculation to occur with internal pressure equal to zero.
- Set $p > 0$ to specify internal pressure to use in this flexibility calculation. Pressure reduces the flexibility increase.
- Set $p = -9999$ to ignore this additional flexibility calculation and use only beam theory.
- Set $p =$ flexibility reduction factor (-FLEXF below); which must be a negative number less than -1.0 .

ASME Pipe Elbow flexibility factors theory [ASME Section NB-3687.3]

This section only applies if $(\text{Bend Radius}/\text{Mean Radius}) \geq 1.70$ or if $(\text{Arclength}) > (2 \times \text{Mean Radius})$

$$\text{Flexibility Factor, FLEXF} = (1.65 \times (\text{Mean Radius})^2) / [t \cdot (\text{Bend Radius})] \times 1 / [1 + (\text{Press})(\text{Mean Radius})(\text{FACT.})]$$

Where:

$$\text{FACT.} = 6 \cdot (\text{MR}/t)^{4/3} \cdot (\text{BR}/\text{MR})^{1/2} / (\text{Et})$$

MR = Mean Radius of elbow wall

BR = Bend Radius

Press = Internal Pressure

t = elbow wall thickness

E = Modulus of Elasticity

If the Flexibility Factor computed is less than 1.0, then STAAD will use 1.0. The Flexibility Factor directly multiplies or contributes to most non-shear terms in the elbow flexibility matrix.

Notes

1. The input for defining the curved member involves 2 steps. The first is the member incidence, which is the same as that for a straight line member. The second is the command described above, which indicates that the segment between the 2 nodes of the member is curved, and not a straight line.
2. Any non-tapered cross section property currently available in STAAD can be assigned to these members.
3. Currently, two load types are permitted on curved members. One is the **SELFWEIGHT** load type, described in Section 5.32.9 of the STAAD.Pro Technical Reference manual. The other is the uniformly (**UNI**) distributed load type of the **MEMBER LOAD** options explained in Section 5.32.2 of the same manual. The uniformly distributed load has to be applied over the full span of the member. Other member loads such as **LINEAR**, **TRAP**, **CONCENTRATED** force or moment, **UNIFORM** moment, etc. are not currently supported. These options are expected to become available in future versions of the program.
4. Some of the other member load types such as **PRESTRESS**, **TEMPERATURE**, **STRAIN** loads, etc. are also not currently supported. These options too are expected to become available in future versions of the program.
5. The results of the analysis currently consist of the nodal displacements of the ends of the curved member, and the member end forces. The nodal displacements are in the global coordinate system. The member end forces are in the local coordinate system, with each end of the member having its own unique local axis system. Results at intermediate sections, such as sectional displacements, and sectional forces will be available in future versions of the program.

Gamma Angle

The plane of the circle defines the plane formed by the straight line joining the two ends of the arc, and the local Y axis of an imaginary straight member

Section 5 Commands and Input Instructions

5.20 Member Property Specification

between those two points. The positive value of the **GAMMA** angle is obtained using the same sense as the positive value of the beta angle of that imaginary straight line member whose local Y axis points towards the vertex of the arc.

Several diagrams intended to show the **GAMMA** angle for various segments lying in the three global planes are shown.

Figure 5-13: Gamma angle for various configurations of the circular arc lying in the global XY plane

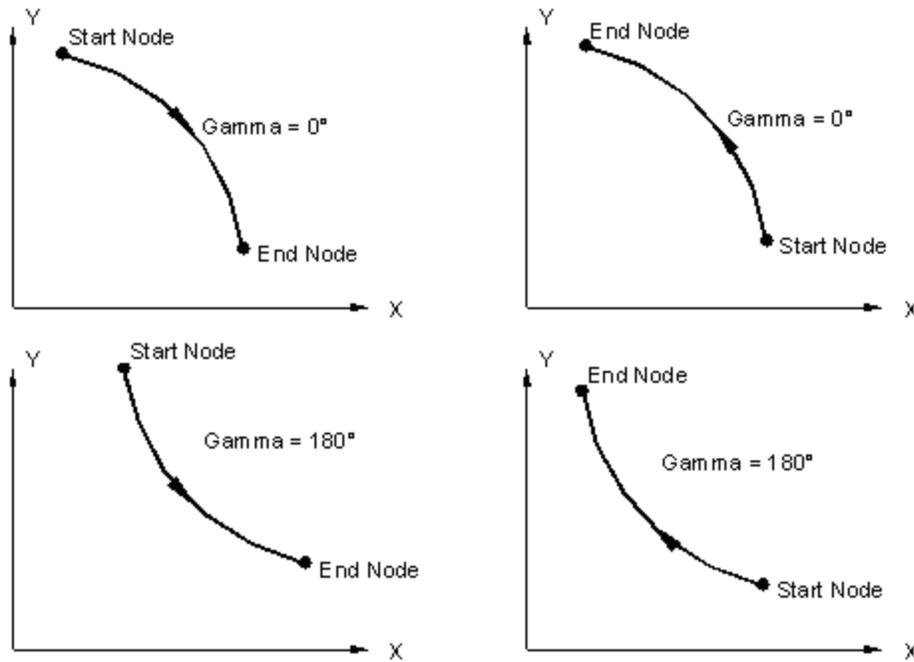


Figure 5-14: Gamma angle for various configurations of the circular arc lying in the global XY plane

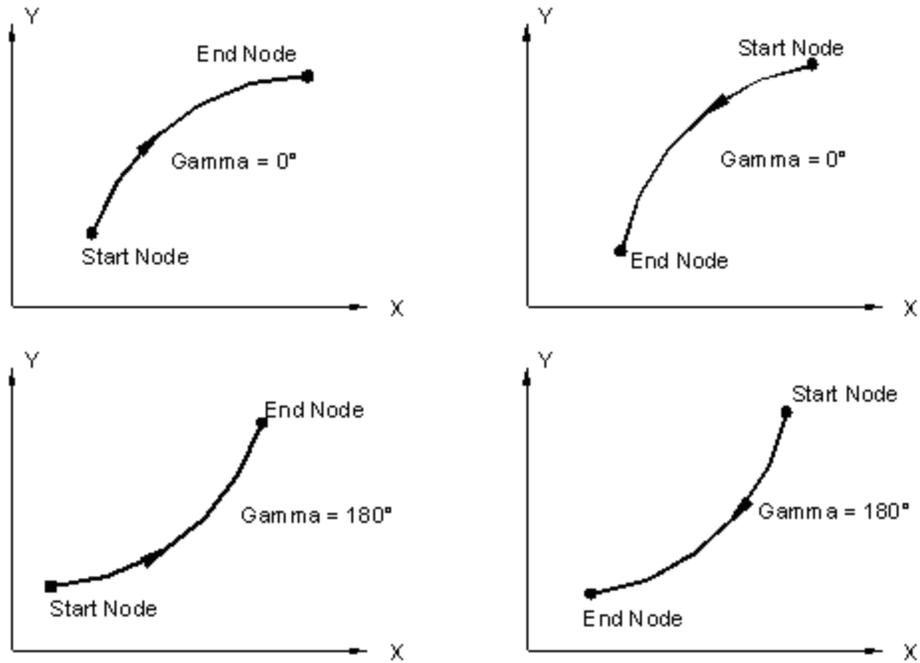


Figure 5-15: Gamma angle for various configurations of the circular arc lying in the global YZ plane

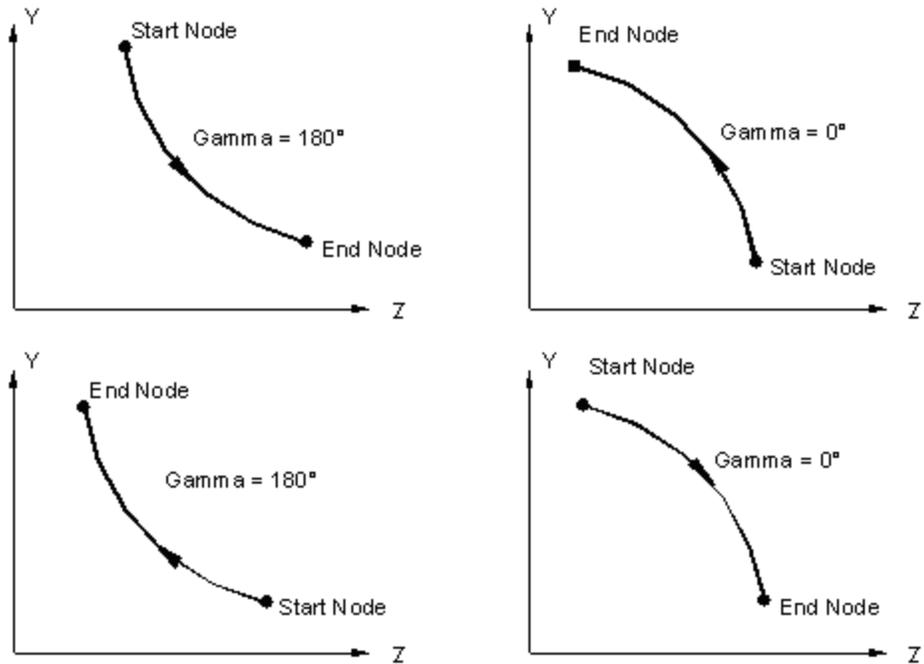


Figure 5-16: Gamma angle for various configurations of the circular arc lying in the global YZ plane

Section 5 Commands and Input Instructions

5.20 Member Property Specification

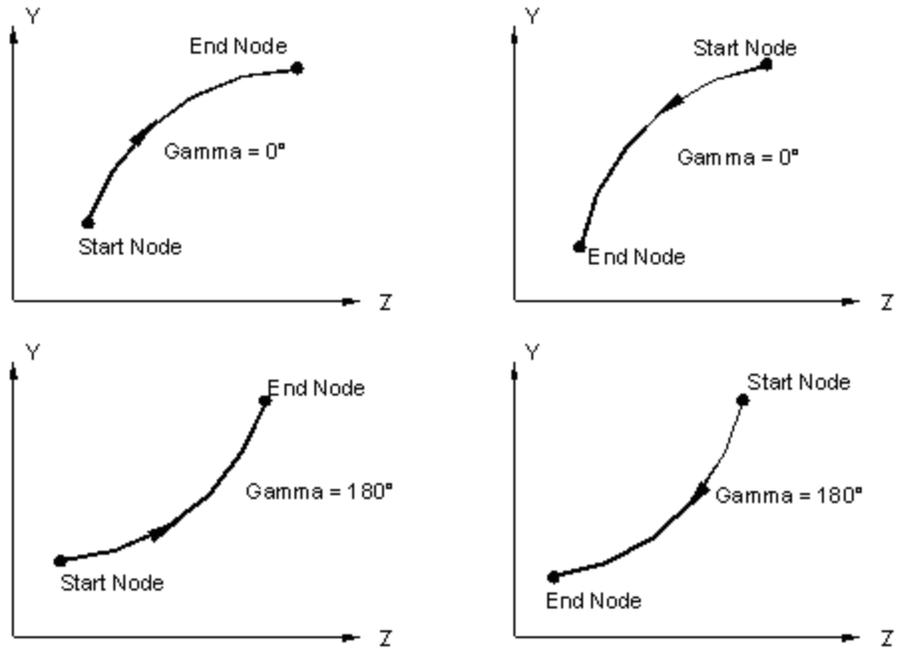


Figure 5-17: Gamma angle for various configurations of the circular arc lying in the global XZ plane

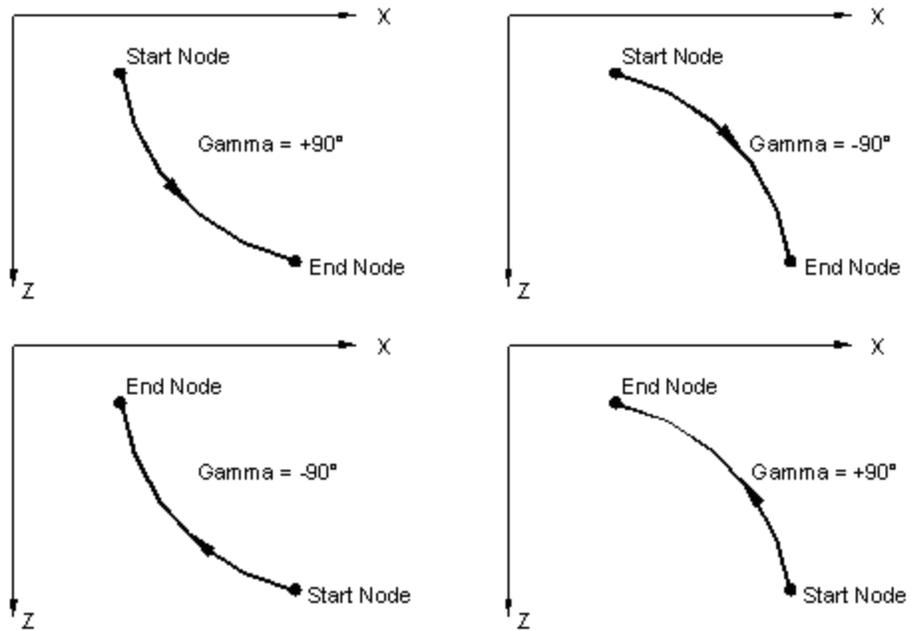
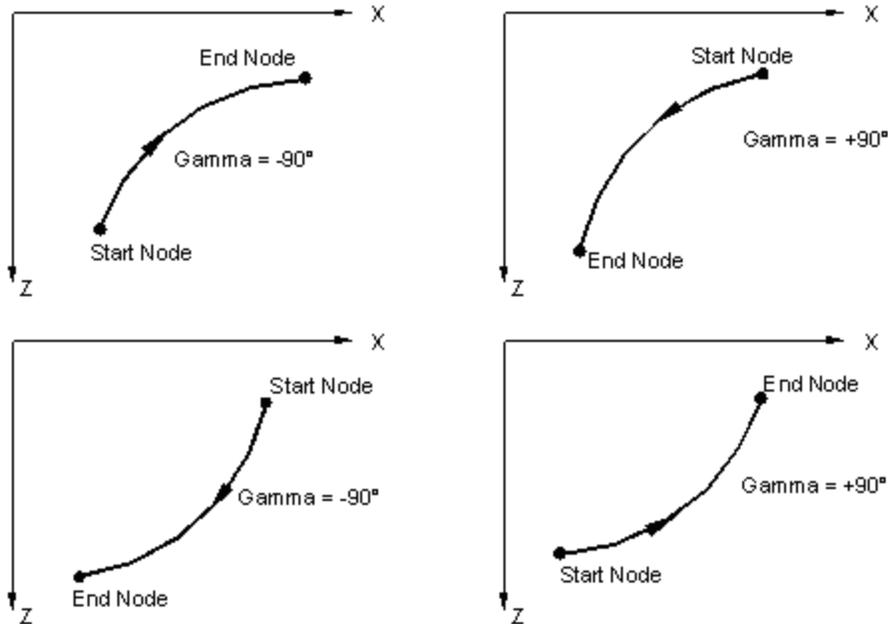


Figure 5-18: Gamma angle for various configurations of the circular arc lying in the global XZ plane



Member local axis system

The local axis directions for curved members are dependent on the point of interest along the curve. The general rules for local axis, as laid out in Section 1.5.2 of this manual are applicable. The figure shown later for member end forces indicates the directions of axes at the start and end nodes.

Rotation of local axis

There is a limited facility available to change the orientation of a curved member cross section. The cross-section may be at the default position where the strong axis (local y) is normal to the plane of the curve and the weak axis is in that plane.

The **BETA ANGLE** and **REFERENCE POINT** options, explained in section 1.5.3 and 5.26.2 of this manual, are not available for curved members.

Sign conventions

The displacements of the nodes of the curved member are along the global axis system just as in the case of straight members.

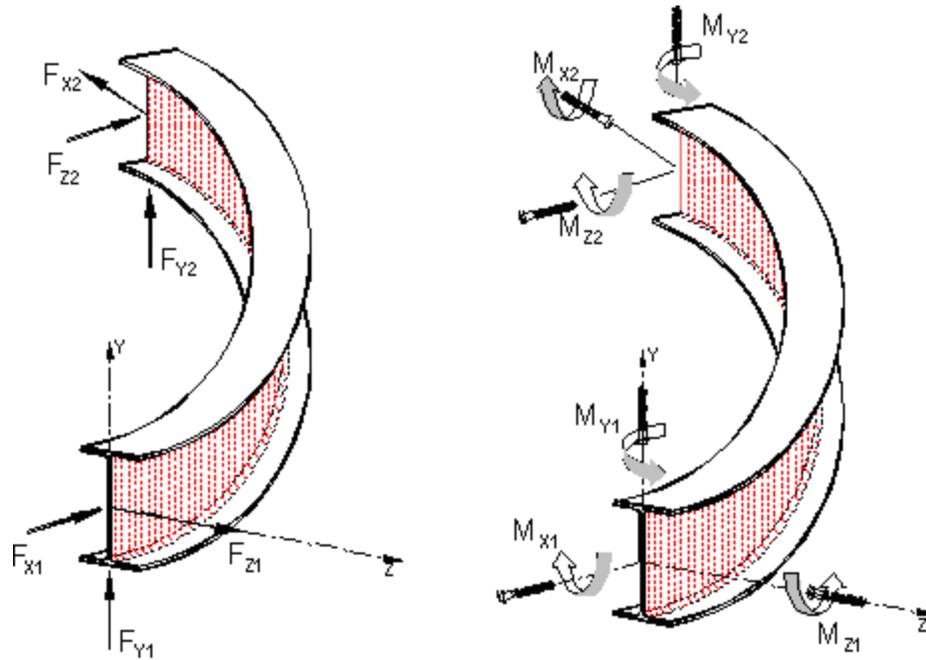
The member end forces for curved members are quite similar to that for straight members. The only distinguishing item is that they are normal and tangential to the local axis at the corresponding ends. For example, F_X at the start is

Section 5 Commands and Input Instructions

5.20 Member Property Specification

tangential to the curve at the start node, and F_X at the end is tangential to the curve at the end node. Similarly, F_Z is along the radial direction at the two ends. Member releases, offsets, tension/compression, truss and cable may not be specified for curved beams.

Figure 5-19: Sign conventions for member end actions; Global Y is vertical



Example

```
STAAD SPACE
UNIT KIP FEET
JOINT COORD CYL REVERSE
1 150 0 0 13 150 0 90
REPEAT 1 30 0 0
REPEAT ALL 1 0 15 0
MEMBER INCIDENCES
1 1 27 26
101 27 28 112
113 40 41 124
201 27 40 213
START GROUP DEFINITION
```

```
MEMBER
_COLUMN 1 TO 26
_CIRCUMFERENTIAL 101 TO 124
_RADIAL 201 TO 213
END GROUP DEFINITION
MEMBER PROPERTIES
_COLUMN PRIS YD 3.0
_CIRCUMFERENTIAL PRIS YD 3.0
_RADIAL PRIS YD 3.0
CONSTANT
E CONCRETE ALL
DENSITY CONCRETE ALL
POISSON CONCRETE ALL
MEMBER CURVED
101 TO 112 RADIUS 150 GAMMA 90.0
113 TO 124 RADIUS 180 GAMMA 90.0
SUPPORTS
1 TO 26 PINNED
LOAD 1
SELF Y -1.0
PERFORM ANALYSIS PRINT STAT CHECK
PRINT MEMBER FORCE LIST 101 113
FINISH
```

5.20.9 Applying Fireproofing on members

STAAD.Pro now includes a method to automatically consider the weight of fireproofing material applied to structural steel.

Two types of fireproofing configurations are currently supported. They are:

Section 5 Commands and Input Instructions

5.20 Member Property Specification

Block Fireproofing (BFP)

The next figure shows this configuration. The fire-protection material forms a rectangular block around the steel section.

The area of fireproofing material (A_{fp}) at any section along the member length is calculated in the following manner.

For Wide Flanges (I-shaped sections), Channels and Tees,

$$A_{fp} = [(B_f + 2T) * (D + 2T)] - A_{steel}$$

For single angles,

$$A_{fp} = [(B_f + 2T) * (D + 2T)] - A_{steel}$$

Where:

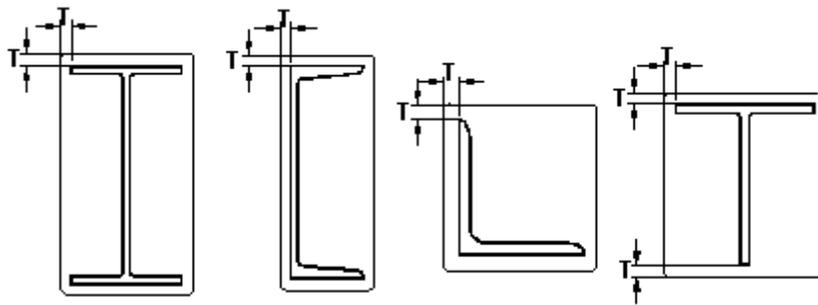
B_f is the flange width

D the overall depth of the steel section

T is the thickness of the fireproofing material beyond the outer edges of the cross section as shown in the next figure.

A_{steel} = Area of the steel section

Figure 5-20: Block fireproofing (BFP) on various shapes



Contour Fireproofing (CFP)

In this configuration, the fire-protection material forms a coating around the steel section as shown in the next figure. The area of fireproofing material (A_{fp}) for this case is calculated in the following manner.

For Wide Flanges (I-shaped sections)

$$A_{fp} = [(B_f + 2T) * (T_f + 2T)] * 2 + [(D - 2T - 2T_f) * (T_w + 2T)] - A_{steel}$$

For single angles,

$$A_{fp} = [(L_1 + 2T) * (2T + T_a) + (L_2 - T_a) * (2T + T_a)] - A_{steel}$$

For Tees,

$$A_{fp} = [(B_f + 2T) * (T_f + 2T)] + [(D - T_w) * (T_w + 2T)] - A_{steel}$$

Where:

B_f is the flange width

D the overall depth of the steel section

T is the thickness of the fireproofing material beyond the outer edges of the cross section as shown in the next figure.

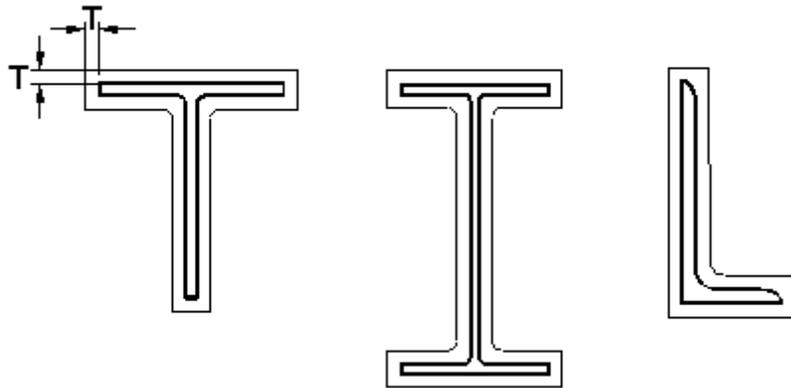
T_f is the thickness of the flange for the I shape and Tee

T_a is the thickness of the leg of the angle

T_w is the thickness of the web for the I shape and Tee

A_{steel} = Area of the steel section

Figure 5-21: Contour fireproofing (CFP) on various shapes



The number of input items required to apply this attribute is four – a) the type of fireproofing b) the thickness T shown in the above figures c) the density of the fireproofing material d) the members on which it is to be applied.

For each such member, A_{fp} is calculated and multiplied by the density of the fireproofing material to obtain the weight per unit length of the member. This is added to the weight per unit length of the steel section itself and the total is used in calculating selfweight. Hence, **SELFWEIGHT** must be one of the load components of load cases if the weight of the fireproofing material should be considered as part of those load cases.

General Format

MEMBER FIREPROOFING

Member-list FIRE { BFP | CFP } THICKNESS f_1 DENSITY f_2

Section 5 Commands and Input Instructions

5.20 Member Property Specification

Where:

f_1 = thickness (T in the figures above) in length units

f_2 = density of fireproofing material in (force / length³) units

In the actual load case itself, nothing besides the **SELFWEIGHT** command is necessary to instruct the program to include the weight of the fireproofing material in the selfweight calculation.

Notes

1. STAAD calculates the fire proofing weight only for the following sections:
For block fireproofing - I-shaped sections like those from the built-in tables (American W,S,M,HP, British UC and UB, etc.), tapered I shaped sections, single channels, angles and Tees.
For CFP-contour fireproofing - the sections are I-beam straight or tapered, angle, Tee.
I-shaped sections like those from the built-in tables (American W,S,M,HP, British UC and UB, etc.), tapered I shaped sections, angles and Tees..
2. Fire proofing weight is not calculated for the following section types: Pipe, tube, composite I beams with slab on top, double channel, double angle, HSS, I-beam with cover plates, prismatic, solid circle or rectangle, castellated, cold formed sections, wood, aluminum, tapered poles, etc.

Example Problem

```
STAAD SPACE
UNIT KIP FEET
JOINT COORDINATES
1 0. 0. ; 2 0. 15. ; 3 20. 15. ; 4 20. 0.
MEMBER INCIDENCE
1 1 2 ; 2 2 3 ; 3 3 4
MEMBER PROPERTY AMERICAN
1 3 TABLE ST W12X26
2 TABLE ST W14X34
CONSTANTS
E STEEL ALL
```

```
POISSON STEEL ALL
DENSITY STEEL ALL
SUPPORT
1 FIXED ; 4 PINNED
UNIT POUND INCH
MEMBER FIREPROOFING
1 3 FIRE BFP THICK 2.0 DENSITY 40
2 FIRE CFP THICK 1.5 DENSITY 40
UNIT KIP FT
LOADING 1 DEADWEIGHT OF STEEL + FIREPROOFING
SELF Y -1.0
LOAD 2 LIVE
MEMBER LOAD
2 UNI GY -0.8
LOAD COMBINATION 3
1 0.75 2 0.75
PERFORM ANALYSIS
PRINT MEMBER FORCES
PRINT SUPPORT REACTIONS
FINISH
```

5.20.10 Member Property Reduction Factors

Concrete design specifications recommend the use of cracked section properties for the analysis and design of concrete sections. Though the methodology to handle cracked section properties is Nonlinear in nature (i.e., the section capacities should be checked and modified depending upon the section forces the section is handling). The model should then be re-analyzed with modified reduced section properties and redesigned. This iteration should be continued until the forces in all sections designed are below the allowable limit of ultimate strength.

In STAAD.Pro, you can specify a set of reduction factors to be applied on the calculated section properties such as Area, Moments of Inertia, and Torsional Constant. If a you want to adopt this approach to account for cracking of

Section 5 Commands and Input Instructions

5.21 Element/Surface Property Specification

concrete sections, refer to Section 10.11.1 of ACI 318 for a set of values to use for these reduction factors depending upon the nature of forces and moments the member is subjected to.

Similarly, the specifications in the AISC 13th edition manual suggest reducing the stiffness of the member during the analysis. The **REDUCEDEI** parameter may also be used when the **PERFORM DIRECT ANALYSIS** command is used. See "Direct Analysis" on page 686 for additional information.

General Format

The format of the command is:

MEMBER CRACKED

<Member List> REDUCTION *{ RAX | RIX | RIY | RIZ } factor

The reduction factor should be a fraction of unity.

Also, this is a multiplication factor on the property value. It does not signify the amount by which the property is reduced, but, it is simply a value by which the unreduced property is multiplied by. Thus, the calculated (or the user specified value) of the property will be multiplied by the reduction factor to arrive at the value used in the analysis.

For example, a factor of 0.45 defined for **RAX** will mean that if the cross sectional area of the gross section is 0.8 ft², the value used in the analysis will be $0.8 \times 0.45 = 0.36$ ft².

Multiple factors can be assigned on the same line.

The reduction factor is considered only for analysis but not for design.

Example

```
MEMBER CRACKED
```

```
1 REDUCTION RAX 0.35 RIX 0.40 RIY 0.45 RIZ 0.45
```

5.21 Element/Surface Property Specification

Individual plate elements, and the Surface element need to have their thickness specified before the analysis can be performed. The commands for specifying this information are explained in this section. No similar properties are required for solid elements. However, constants such as modulus of elasticity, Poisson's Ratio, etc. are required.

5.21.1 Element Property Specification

This set of commands may be used to specify properties of plate finite elements. Elements of uniform or linearly varying thickness may be modeled using this command. The value of the thickness must be provided in current units.

Unlike members and plate/shell elements, no properties are required for solid elements. However, constants such as modulus of elasticity and Poisson's ratio are to be specified.

General Format

ELEMENT PROPERTY

element-list THICKNESS f_1 (f_2 , f_3 , f_4)

Where:

f_1 = Thickness of the element.

$f_2 \dots f_4$ = Thicknesses at other nodes of the element, if different from f_1 .

Example

```
UNIT . . .
ELEMENT PROPERTY
1 TO 8 14 16 THI 0.25
```

5.21.2 Surface Property Specification

This set of commands may be used to specify properties of surface entities.

General Format

SURFACE PROPERTY

surface-list THICKNESS t

Where:

t = Thickness of the surface element, in current units.

Example

Section 5 Commands and Input Instructions

5.22 Member/Element Releases

SURFACE PROPERTY

1 TO 3 THI 18

The attributes associated with the surface element, and the sections of this manual where the information may be obtained, are listed below:

Attributes	Related Sections
Surfaces Incidences	5.13.3
Openings in surface	5.13.3
Local Coordinates system for surfaces	1.6.3
Specifying sections for stress/force output	5.13.3
Property for surfaces	5.21.2
Material constants	5.26.3
Surface loading	5.32.3.4
Stress/Force output printing	5.42
Shear Wall Design	3.8.2, 5.55

5.22 Member/Element Releases

STAAD allows specification of releases of degrees of freedom for frame members and plate elements.

5.22.1 Member Release Specification

This set of commands may be used to fully release specific degrees of freedom at the ends of frame members. They may also be used to describe a mode of attachment where the member end is connected to the joint for specific degrees of freedom through the means of springs.

General Format

MEMBER RELEASES

```
member-list {START | END | BOTH } { *{ FX | FY | FZ | MX |
MY | MZ } | *{KFX f1 | KFY f2 | KFZ f3 | KMX f4 | KMY f5
| KMZ f6 } | {MP f7 | *{ MPX f8 | MPY f9 | MPZ f10 }} }
```

Where:

FX ... MZ = represent force-x through moment-z degrees of freedom in the member local axes

KFX ... KMZ f₁ ... f₆ = spring constants for these degrees of freedom, in current units.

f₇ = release factor for all three moments

f₈, f₉, f₁₀ = release factors for each moment separately. The moment related stiffness co-efficient will be multiplied by a factor of $(1 - f_n)$ at the specified end. Release factors must be in the range of 0.001 through 0.999.

Note: If FX through MZ is used, it signifies a full release for that degree of freedom and if KFX through KMZ is used, it signifies a spring attachment.

Notes

- Member releases are a means for describing a type of end condition for members when the default condition, namely, fully moment and force resistant, is not applicable. Examples are bolted or riveted connections. Partial moment releases are a way of specifying bending and torsional moment capacity of connections as being a fraction of the full bending and torsional strength.
- It is important to note that the factor f_1 indicates a reduction in the stiffness corresponding to the rotational degrees of freedom MX, MY, and MZ. In other words, you should not expect the moment on the member to reduce by a factor of f_1 . It may be necessary to perform a few trials in order to arrive at the right value of f_1 that results in the desired reduction in moment.

Section 5 Commands and Input Instructions

5.22 Member/Element Releases

- c. The **START** and **END** are based on the **MEMBER INCIDENCE** specification. The **BOTH** specification will apply the releases at both ends.
- d. At any end of the member—for any particular **DOF**—full, partial, and spring release cannot be applied simultaneously. Only one out of the three is permitted.
- e. If **MY** (or **MZ**) is fully released at both ends, then **VZ** (or **VY**) cannot be transmitted through the member. The final shears in the member will be entirely due to loads applied directly to the member.

Example

```
MEMBER RELEASE
1 3 TO 9 11 12 START KFX 1000.0 MY MZ
1 10 11 13 TO 18 END MZ KMX 200.0
```

In this example, for members 1, 3 to 9, 11 and 12, the moments about the local Y and Z axes are released at their start joints (as specified in **MEMBER INCIDENCES**). Further, these members are attached to their **START** joint along their local x axis through a spring whose stiffness is 1000.0 units of force/length. For members 1, 10, 11 and 13 to 18, the moment about the local Z axis is released at their end joint. Also, the members are attached to their **END** joint about their local x axis through a moment-spring whose stiffness is 200.0 units of force-length/Degree. Members 1 and 11 are released at both start and end joints, though not necessarily in the same degrees of freedom.

Partial Moment Release

Moments at the end of a member may be released partially using the **MP** option (to provide the same partial release in all . This facility may be used to model partial fixity of connections. The following format may be used to provide a partial moment release. This facility is provided under the **MEMBER RELEASE** option and is in addition to the existing **RELEASE** capabilities.

Example

```
MEMBER RELEASE
15 TO 25 START MP 0.25
```

The above **RELEASE** command will apply a factor of 0.75 on the moment related stiffness coefficients at start node of members 15 to 25.

5.22.2 Element Release Specification

This set of commands may be used to release specified degrees of freedoms at the corners of plate finite elements.

General Format

ELEMENT RELEASE

```
element-list { J1 | J2 | J3 | J4 } *{ FX | FY | FZ | MX | MY
| MZ }
```

Where:

J₁, J₂, J₃ and J₄ = signify joints in the order of the specification of the element incidence. For example, if the incidences of the element were defined as 35 42 76 63, J₁ represents 35, J₂ represents 42, J₃ represents 76, and J₄ represents 63.

FX through MZ = represents forces/moments to be released per local axis system.

Note: Element releases at multiple joints cannot be specified in a single line. Those must be specified separately as shown below.

Examples

Example of Correct Usage

```
ELEMENT RELEASE
10 TO 50 J1 MX MY
10 TO 50 J2 MX MY
10 TO 50 J3 MY
10 TO 50 J4 MY
```

Example of Incorrect Usage

```
ELEMENT RELEASE
10 TO 50 J1 J2 MX MY
```

10 TO 50 J3 J4 MY

Notes

- a. All releases are in the local axis system. See Figure 1.13 for the various degrees of freedom. Fx and Fy have the same sense as Sx and Sy in Figure 1.13. Fz has the same sense as SQx or SQy. Generally, do not over release. The element must still behave as a plate after the releases.
- b. Selfweight is applied at each of the nodes as if there were no releases.
- c. Thermal stresses will include the fixed-end thermal pre-stress as if there were no release.
- d. May not be used with the Element Plane Stress or Element Ignore Inplane Rotation commands on the same element.
- e. Note that the usual definitions of local Mx and My are reversed here. See Figure 1.13 for the definitions of Mx and My. Releasing Fz, Mx, My will release all bending capability. Releasing Fx, Fy, Mz will release all in-plane stiffness.

5.22.3 Element Ignore Stiffness

Structural units like glass panels or corrugated sheet roofs are subjected to loads like wind pressures or snow loads. While these units are designed to carry those loads and transmit those loads to the rest of the structure, they are not designed to provide any additional stiffness to the structure. One way to handle the situation is to not input the unit as part of the structural model, and apply the load using load generation techniques like **AREA LOAD** or **FLOOR LOAD**.

STAAD provides another way of handling such units. This is through the help of the **ELEMENT IGNORE STIFFNESS** command. To use this feature, the glass panel or roof unit must be defined using plate elements. The **IGNORE STIFFNESS** command enables one to consider the unit just for the purpose of application of **ELEMENT LOAD** commands, while its stiffness will not be considered during the assembly of the stiffness matrix. In other words, it is like an **INACTIVE** member which is active for **ELEMENT LOAD** command application but **INACTIVE** for stiffness. Like the **INACTIVE** command, the plates listed here will become active at the next **CHANGE** command. To keep them inactive, re-enter this data after each **CHANGE**.

The **SELFWEIGHT**, **ELEMENT WEIGHT**, **TEMPERATURE**, and mass of the plates listed here will still be ignored.

General Format

```
IGNORE STIFFNESS {ELEMENT} element-list
```

Example

```
IGNORE STIFFNESS ELEMENT 78 TO 80
```

5.23 Axial Member Specifications

A member can have only *one* of the following specifications:

- Truss
- Tension-only
- Compression-only
- Cable

If multiple specifications are applied to the same member, only the last entered will be used (Warnings will be printed).

Note: **MEMBER TRUSS**, **MEMBER TENSION**, **MEMBER COMPRESSION**, and **MEMBER CABLE** are axial-only for stiffness. **MEMBER CABLE** are special truss members that may also be specified as tension-only.

5.23.1 Member Truss Specification

This command may be used to model a specified set of members as TRUSS members.

This specification may be used to specify TRUSS type members in a PLANE, SPACE or FLOOR structure. The TRUSS members are capable of carrying only axial forces. Typically, bracing members in a PLANE or SPACE frame will be of this nature.

General Format

```
MEMBER TRUSS  
member-list TENSION f1
```

Where:

Section 5 Commands and Input Instructions

5.23 Axial Member Specifications

f_1 = optional Initial Tension in truss member, in current units.

Note: For Nonlinear CABLE ANALYSIS Only: The Tension parameter is ignored except in Nonlinear Cable Analysis. For that analysis type, a truss with pretension is considered to be nonlinear (large displacement). In this analysis type, trusses without preload are assumed to be linear members that carry both tension and compression regardless of this command.

This command is superfluous when a **TRUSS** type structure has already been specified using the command **STAAD TRUSS**.

Example

```
MEMB TRUSS
1 TO 8 10 12 14 15
```

Notes

- The **TRUSS** member has only one degree of freedom—the axial deformation. Note also that Member Releases are not allowed. Selfweight and transverse loads may induce shear/moment distributions in the member.
- Member loads are lumped at each end, whereas a frame member with moment releases only retains the axial component of the applied member load.

5.23.2 Member Cable Specification

This command may be used to model a specified set of members as **CABLE** members.

Cable members, in addition to elastic axial deformation, are also capable of accommodating the stiffness effect of initial tension and tension due to static loads. When used in a nonlinear cable analysis, cable members are capable of accommodating large displacements. See "1.11 Cable Members" on page 47 for a theoretical discussion of cable members.

General Format

```
MEMBER CABLE
```

member-list { TENSION f_1 | LENGTH f_2 }

Where:

f_1 = Initial Tension in cable member (in current units)

f_2 = Unstressed cable length (in current units)

Notes

1. The tension specified in the cable member is applied on the structure as an external load as well as is used to modify the stiffness of the member. The tension value must be positive to be treated as a cable otherwise it is a truss (See "1.11 Cable Members" on page 47). If the **TENSION** parameter or the value is omitted, a minimum tension will be used.
2. This is a truss member but not a tension-only member unless you also include this member in a **MEMBER TENSION** input (See "5.23.3 Member Tension/Compression Specification" on page 377). Note also that Member Releases are *not* allowed.
3. The tension is a preload and will not be the final tension in the cable after the deformation due to this preload.
4. The tension is used to determine the unstressed length. That length will be shorter than the distance between the joints by the distance that the tension will stretch the cable.

Example

```
MEMB CABLE
20 TO 25 TENSION 15.5
```

5.23.3 Member Tension/Compression Specification

This command may be used to designate certain members as Tension-only or Compression-only members.

Tension-only members are truss/cable members that are capable of carrying tensile forces only. Thus, they are automatically inactivated for load cases that create compression in them.

General Format

MEMBER TENSION
member-list

Section 5 Commands and Input Instructions

5.23 Axial Member Specifications

MEMBER COMPRESSION

member-list

MEMBER TENSION 0

(no list required)

Linear Tension/ Compression Analysis

Compression-only members are truss members that are capable of carrying compressive forces only. Thus, they are automatically inactivated for load cases that create tension in them. Member Releases are not allowed on members with this attribute.

The procedure for analysis of Tension-only or Compression-only members requires iterations for every load case and therefore may be quite involved. you may also consider using the **INACTIVE** specification (instead of Tension/Compression) if the solution time becomes unacceptably high.

If a **CHANGE** command is used (because of a change in the list of tension members, cable tension, supports, etc.), then the **SET NL** command must be used to convey to STAAD that multiple analyses and multiple structural conditions are involved.

Note: For Nonlinear cable analysis, this command is unnecessary and ignored. Cables are automatically assumed to be partially to fully tension only (except that there should always be selfweight) without this command. In this analysis type, trusses without preload are assumed to be linear members that carry both tension and compression regardless of this command.

- a. Loads that have been defined on members declared as **MEMBER TENSION** or **MEMBER COMPRESSION** will be active even when the member becomes **INACTIVE** during the process of analysis. This applies to **SELFWEIGHT**, **MEMBER LOADS**, **PRESTRESS & POSTSTRESS LOADS**, **TEMPERATURE LOAD**, etc.
- b. A member declared as a **TENSION** only member or a **COMPRESSION** only member will carry axial forces only. It will not carry moments or shear forces. In other words, it is a truss member.
- c. Do not use Load Combination to combine these cases. Tension/Compression cases are Nonlinear and should not be linearly

combined as in Load Combination. Use a primary load case with the Repeat Load command.

Example

```
MEMBER TENSION
12 17 19 TO 37 65
MEMBER COMPRESSION
5 13 46 TO 53 87
```

Member Tension 0

This command switches off ALL tension/compression only specifications for load cases which are specified subsequent to this command, usually entered after a CHANGE command. There is no list associated with this command. Hence, for any further primary load cases, the tension/compression only attributed is disabled for ALL members.

Example

The following is the general sequence of commands in the input file if the **MEMBER TENSION** or **MEMBER COMPRESSION** command is used. This example is for the **MEMBER TENSION** command. Similar rules are applicable for the **MEMBER COMPRESSION** command. The dots indicate other input data items.

```
STAAD ...
SET NL ...
UNITS ...
JOINT COORDINATES
...
MEMBER INCIDENCES
...
ELEMENT INCIDENCES
...
CONSTANTS
```

Section 5 Commands and Input Instructions

5.23 Axial Member Specifications

```
...  
MEMBER PROPERTY  
...  
SUPPORTS  
...  
MEMBER TENSION  
...  
LOAD 1  
...  
LOAD 2  
...  
LOAD 3  
...  
LOAD 4  
...  
LOAD 5  
REPEAT LOAD  
...  
PERFORM ANALYSIS  
CHANGE  
LOAD LIST ALL  
PRINT ...  
PRINT ...  
PARAMETER  
...  
CHECK CODE ...  
FINISH
```

Notes

- a. See "5.5 Set Command Specification" on page 284 for explanation of the **SET NL** command. The number that follows this command is an upper bound on the total number of primary load cases in the file.
- b. STAAD performs up to 10 iterations automatically, stopping if converged. If not converged, a warning message will be in the output. Enter a **SET ITERLIM i** command ($i > 10$) before the first load case to increase the default number of iterations. Since convergence may not be possible using this procedure, do not set the limit too high.
- c. The principle used in the analysis is the following.
 - The program reads the list of members declared as MEMBER TENSION and/or COMPRESSION.
 - The analysis is performed for the entire structure and the member forces are computed.
 - For the members declared as MEMBER TENSION / COMPRESSION, the program checks the axial force to determine whether it is tensile or compressive. If the member cannot take that load, the member is "switched off" from the structure.
 - The analysis is performed again without the switched off members.
 - Up to 10 iterations of the above steps are made for each load case, unless a higher value is set using the command ITERLIM.
 - This method does not always converge and may become unstable. Check the output for instability messages. Do not use results if the last iteration was unstable.
- d. A revised MEMBER TENSION / COMPRESSION command and its accompanying list of members may be provided after a CHANGE command. If entered, the new MEMBER TENSION/COMPRESSION commands replace all prior such commands. If these commands are not entered after a CHANGE, then the previous commands will still be applicable.
- e. The MEMBER TENSION command should not be used if the following load cases are present : Response Spectrum load case, Time History Load case, or Moving Load case. If used, the MEMBER TENSION /COMPRESSION will be ignored in all load cases.
- f. If UBC Load cases are included, then follow each UBC load case with an Analysis command, then a Change command.

5.24 Element Plane Stress and Ignore Inplane Rotation Specification

These commands allow the user to model the following conditions on plate elements

- a. PLANE STRESS condition
- b. In-plane rotation stiffness reformulated to be rigid or to be zero.

General Format

```
ELEMENT { PLANE STRESS | RIG ID ( INPLANE ROTATION )  
| IGNORE ( INPLANE ROTATION ) }  
element-list
```

The **PLANE STRESS** specification allows the user to model selected elements for PLANE STRESS action only [No bending or transverse shear stiffness].

The **RIGID INPLANE ROTATION** command causes the program to connect the corner Mz "in-plane rotation" action to the other corner Mz rotations rigidly. The STAAD plate element formulation normally produces a very soft Mz stiffness that improves the inplane shear deformation. However when the plate Mz is connected to a beam bending moment as the only load path for that moment, then the RIGID INPLANE option may be used to have that element carry the Mz moment to the other joints rigidly to avoid the instability at the beam end. Usually only the elements connected to beams in this manner would have this specification.

The **IGNORE INPLANE ROTATION** command causes the program to ignore "in-plane rotation" actions. The STAAD plate element formulation normally includes this important action automatically. However, it may be noted that some element formulations ignore this action by default. The user may utilize this option to compare STAAD results with solutions from these programs.

These options are exclusive of each other and also exclusive of element releases. No single element may have more than one of these options.

Example

```
ELEMENT PLANE STRESS  
1 TO 10 15 20 25 35  
ELEMENT IGNORE
```

30 50 TO 55

5.25 Member Offset Specification

This command may be used to rigidly offset a frame member end from a joint to model the offset conditions existing at the ends of frame members.

General Format

MEMBER OFFSETS

member-list { START | END } (LOCAL) f_1 , f_2 , f_3

Where:

f_1 , f_2 , and f_3 = correspond to the distance, measured in LOCALized or Global coordinate system, from the joint (START or END as specified) to the centroid of the starting or ending point of the members listed.

LOCAL = optional parameter, if not entered then f_1 , f_2 , f_3 are assumed to be in global. LOCAL means that the distances f_1 , f_2 , f_3 are in the same member coordinate system that would result if the member were not offset and BETA = o.o.

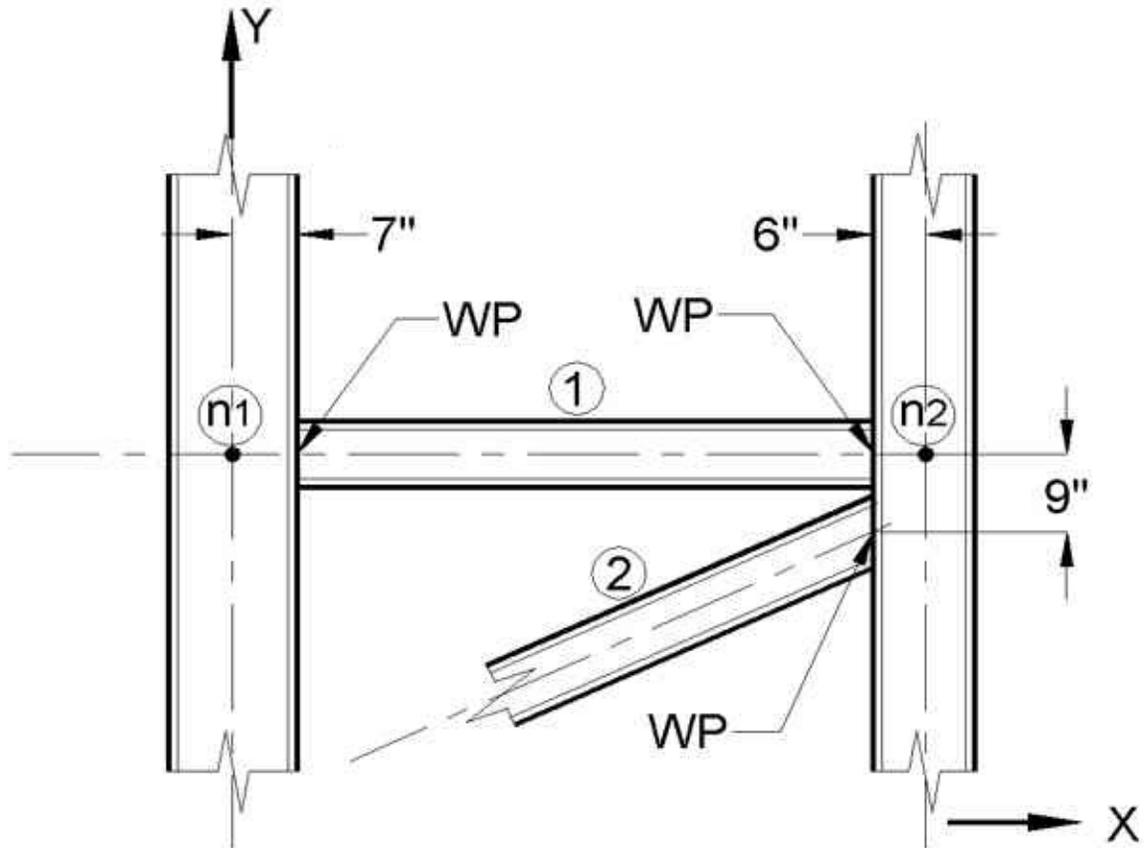
Description

The **MEMBER OFFSET** command can be used for any member whose starting or ending point is not concurrent with the given incident joint. This command enables the user to account for the secondary forces, which are induced due to the eccentricity of the member. Member offsets can be specified in any direction, including the direction that may coincide with the member x-axis.

Figure 5-22: Example of working point (WP) represented by member end offsets

Section 5 Commands and Input Instructions

5.25 Member Offset Specification



In the figure above, WP refers to the location of the centroid of the starting or ending point of the member.

Example

MEMBER OFFSET

1 START 7

1 END -6

2 END -6 -9

Notes

- If a MEMBER LOAD (see MEMBER LOAD specification) is applied on a member for which MEMBER OFFSETS have been specified, the location of the load is measured not from the coordinates of the starting joint. Instead, it is measured from the offset location of the starting joint.
- START and END is based on the user's specification of MEMBER INCIDENCE for the particular member.

5.26 Specifying and Assigning Material Constants

Material constants are attributes like Modulus of Elasticity and Density which are required for operations like generating the stiffness matrix, computing selfweight, and for steel and concrete design.

In STAAD, there are two ways in which this data may be specified :

- a. A two-step process that involves:
 1. Creating the material data by defining **MATERIAL** tags specified under the heading **DEFINE MATERIAL** (See "5.26.1 Define Material" on page 386)
 2. Assigning them to individual members, plates and solids under the heading **CONSTANTS** (See "Specifying Constants for Members and Elements" on page 388)

This will create commands as shown below :

```

DEFINE MATERIAL PART 1
... NAME
...
...
END MATERIAL

CONSTANTS PART 2
MATERIAL NAME ...
  
```

- b. Assign material attributes explicitly by specifying the individual constants (See "Specifying Constants for Members and Elements" on page 388).

```

CONSTANTS
E ...
POISSON .
  
```

See "Surface Constants Specification" on page 394 for an explanation for the commands required to assign material data to Surface elements.

Section 5 Commands and Input Instructions

5.26 Specifying and Assigning Material Constants

5.26.1 Define Material

This command may be used to specify the material properties by material name. You will then assign the members and elements to this material name in the **CONSTANTS** command (See "Specifying Constants for Members and Elements" on page 388 for details and an example).

Note: **ISOTROPIC** materials can be assigned to all element types, **2DORTHOTROPIC** materials should only be assigned to plate elements.

General Format

DEFINE MATERIAL

then

ISOTROPIC name

E f_1

G f_3

POISSON f_6

DENSITY f_7

ALPHA f_8

DAMPING f_{10}

or

2DORTHOTROPIC name

E f_1 (f_2)

G f_3 (f_4) (f_5)

POISSON f_6

DENSITY f_7

ALPHA f_8 (f_9)

DAMPING f_{10}

Repeat **ISOTROPIC** or **2DORTHOTROPIC** name and values for as many materials as desired then:

END MATERIAL (DEFINITION)

Where:

name = material name (name of up to 36 characters).

f_1, f_2 = specifies Young's Modulus (E). (f_2 in local Y for 2D orthotropic materials)

f_3, f_4, f_5 = specifies Shear Modulus (G). For plates, the following are G values in local directions: f_3 is the G for in-plane shear; f_4 is the G for transverse shear in the local Y-Z direction; f_5 is the G for transverse shear in the local Z-X direction. (Enter only f_5 for beams when Poisson is not in the range of 0.01 to 0.499.)

f_6 = specifies Poisson's Ratio. If G is not entered, then this value is used for calculating the Shear Modulus ($G = 0.5 \times E / (1 + \text{POISSON})$). This value must be in the range of 0.01 to 0.499. Poisson's ratio must be entered for orthotropic plates or when Poisson cannot be computed from G.

f_7 = specifies weight density.

f_8, f_9 = Co-efficient of thermal expansion. (f_9 in local Y for 2D orthotropic materials)

f_{10} = Damping ratio to be used in computing the modal damping by the composite damping method. Damping must be in the range of 0.001 to 0.990.

Note: Any material property which you do not explicitly specify is assumed to be the default value.

f_1 defaults to 0.0 but a positive value must be entered or an error will ensue

f_2 defaults to f_1

f_3 defaults to $0.5 \times E / (1 + \text{POISSON})$

f_4 defaults to f_3

f_5 defaults to f_4

f_6 defaults to a sliding scale value based on E (that is: 0.30 when E is near that of steel, 0.33 when E is near that of aluminum, 0.17 when E is near that of concrete) .

f_7 defaults to 0.0

f_8 defaults to 0.0

Section 5 Commands and Input Instructions

5.26 Specifying and Assigning Material Constants

f_9 defaults to f_8
 f_{10} defaults to 0.0

Hint: If one or more of the material properties is not explicitly specified, the results may be unpredictable or even incorrect with respect to the intended behavior. Therefore, it is best practice to always specify each material property for each defined material.

5.26.2 Specifying Constants for Members and Elements

This command may be used to specify the material properties (Modulus of Elasticity, Poisson's ratio, Density, Coefficient of linear expansion, and material damping) of the members and elements. In addition, this command may also be used to specify the member orientation (BETA angle or reference point/vector).

General Format

CONSTANTS

MATERIAL *name* { **MEMBER** *member/element-list* | **(ALL)** }

Where:

name = material name as specified in the **DEFINE MATERIAL** command (See "Define Material" on page 386).

or

{ **E** f_1 | **G** f_2 | **POISSON** f_3 | **DENSITY** f_4 | **BETA** { f_5 | **ANGLE** | **RANGLE** } | **ALPHA** f_6 | **CDAMP** f_7 } { **MEMBER** *memb/elem-list* | **BEAM** | **PLATE** | **SOLID** | **(ALL)** }
{ **REF** f_8 , f_9 , f_{10} | **REFJT** f_{11} | **REFVECTOR** f_{12} f_{13} f_{14} } **MEMBER** *memb/elem-list*

Where:

memb/elem-list = MEM, BEA, PLA, SOL, ALL. Only MEM may be followed by a list. If none are specified, the default is to use **ALL**, which means all members and elements; BEA means all members; PLA, all plates; SOL, all solids.

f_1 = Specifies Young's Modulus (E). This value must be provided before the POISSON for each member/element in the Constants list.

f_2 = specifies Shear Modulus (G). Enter only for beams when Poisson would not be 0.01 to 0.499.

f_3 = specifies Poisson's Ratio. This value is used for calculating the Shear Modulus ($G = 0.5 \times E / (1 + \text{POISSON})$).

f_4 = specifies weight density.

f_5 = Specifies member rotation angle in degrees (See "Relationship Between Global & Local Coordinates" on page 15).

f_6 = Co-efficient of thermal expansion.

f_7 = Damping ratio to be used in computing the modal damping.

The following values are used in various methods to define the BETA angle based on geometry:

f_8, f_9, f_{10} = Global X, Y, and Z coordinates for the reference point

f_{11} = use location of joint f_{11} for the reference point, from which the **BETA** angle will be calculated by STAAD.

f_{12}, f_{13}, f_{14} = Establishes a Reference Vector along which the local y-axis is aligned. From the start node of the member, move by a distance of f_{12} along the beam's local X axis, f_{13} along the local Y axis, and f_{14} along the local Z axis to define the end node of the reference vector. (The BETA angle is thus the angle between the local y-axis and the reference vector)

Using BETA ANGLE and RANGLE

Single angle sections are oriented according to their principal axes by default. If it is necessary to orient them such that their legs are parallel to the global axes, the **BETA** specification must be used. STAAD offers the following additional specifications for this purpose:

- **BETA ANGLE**
- **BETA RANGLE**

Both of the above options will result in an orientation with the legs parallel to the global axis. The **ANGLE** option rotates the section through the angle ($90^\circ - \alpha$) (where α = angle between the principal axis system and the geometric axis

Section 5 Commands and Input Instructions

5.26 Specifying and Assigning Material Constants

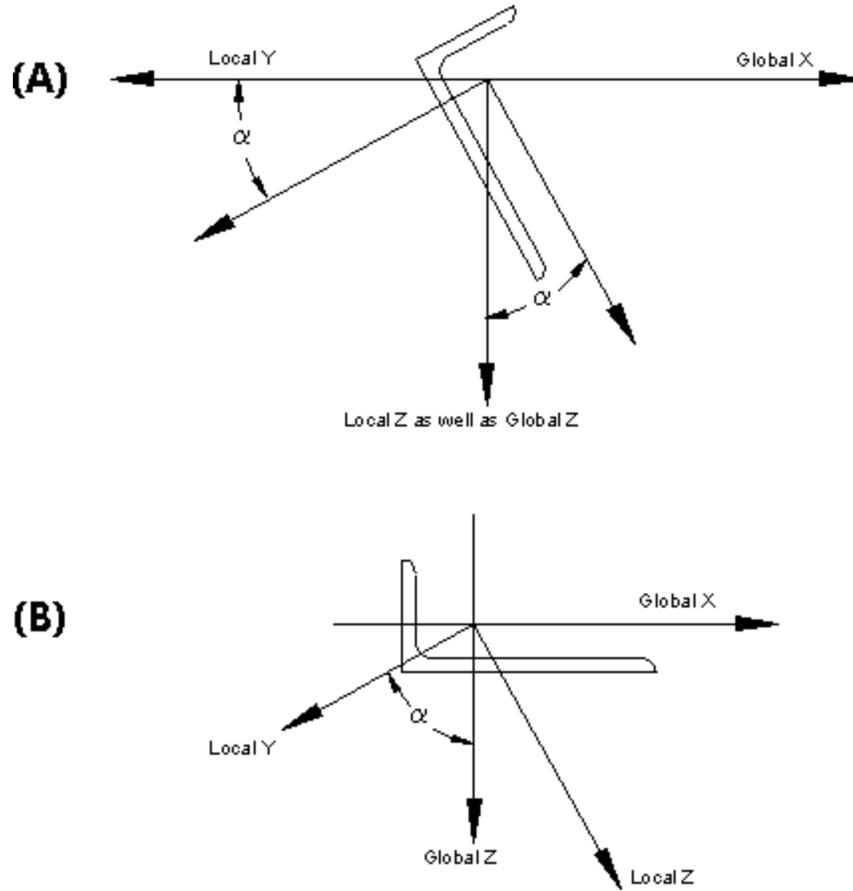
system of the angle). The **RANGLE** option rotates the section through an angle equal to $(180^\circ - \alpha)$. For unequal angles, the right option must be used based on the required orientation.

Table 5-5: Effect of **BETA ANGLE** and **BETA RANGLE** commands

BETA value =	Zero (0)	ANGLE	RANGLE
"BT" Angle			
"RA" Angle			

Note: In the figures in the preceding table, the local x-axis goes into the page/screen. Global Y-axis is vertical.

Figure 5-23: Orientation of a column (vertical member) corresponding to A) $BETA = 0$ and B) $BETA = ANGLE$



Note: In the preceding figures, the local x-axis and Global Y-axis come out of the page/screen (i.e., the local x-axis is parallel to the Global Y-axis).

Built-In Material Constants

For E, G, POISSON, DENSITY, ALPHA and CDAMP, built-in material names can be entered instead of a value for f . The built-in names are **STEEL**, **CONCRETE**, & **ALUMINUM**. Appropriate values will be automatically assigned for the built-in names.

Table 5-6: Constants (in Kip, inch, Fahrenheit units)

Constant	Steel	Concrete	Aluminum	Units
E (US)	29,000	3,150	10,000	Kip/in ²

Section 5 Commands and Input Instructions

5.26 Specifying and Assigning Material Constants

Con- stant	Steel	Con- crete	Alu- minum	Unit- s
Poisson's	0.30	.17	.33	
Density	.0002- 83	.000086- 8	.000098	Kip/i- n ³
Alpha	6.5E-6	5.5E-6	12.8E-6	L/L/° F
CDAMP	.03	.05	.03	Rati- o
E (nonUS)	29, 732.73- 6			Kip/i- n ²

Table 5-7: Constants (in MKS, Celsius units)

Con- stant	Ste- el	Con- crete	Alu- minum	Unit- s
E (US)	199, 947, 960	21,718, 455	68,947,573	kN/- m ²
Poisson's	0.30	.17	.33	
Density	76.81- 9 541	23.561612	26.601820	kN/- m ³
Alpha	12.0- E-6	10.0E-6	23.0E-6	L/L/° C
CDAMP	.03	.05	.03	Rati- o
E (nonUS)	205, 000, 000			kN/- m ²

Note: E (US) is used if US codes were installed or if Member Properties American is specified for an analysis; otherwise E (nonUS) is used.

Example 1

```

DEFINE MATERIAL
ISOTROPIC CFSTEEL
E 28000.
POISSON 0.25
DENSITY 0.3E-3
ALPHA 11.7E-6
DAMP 0.075
END MATERIAL
CONSTANTS
MATERIAL CFSTEEL MEMB 1 TO 5
CONSTANTS
E 2.1E5 ALL
BETA 45.0 MEMB 5 7 TO 18
DENSITY STEEL MEMB 14 TO 29
BETA 90 MEMB X

```

Example 2

The **REFVECTOR** command is used as in the following example:

```
REFVECTOR 0 2 1 MEMBER 27 TO 32
```

This command will set **BETA** as 90° for all members parallel to the X-axis and instructs the program to do the following procedure:

1. Establish the beam's local X,Y and Z axis corresponding to Beta = 0
2. Set the start node of the reference vector to be the same as the start node of the member.

Section 5 Commands and Input Instructions

5.26 Specifying and Assigning Material Constants

3. From the start node of the reference vector, move by a distance of o along the beam's local X axis, 2 along the local Y axis, and 1 along the local Z axis. This establishes the end node of the reference vector.
4. At the end of step 3, the start node as well as the end node of the reference vector are known. That is the now the final direction of the member's local Y axis.

Since the local Y axis corresponding to Beta o is known, and the local-Y axis corresponding to the beam's final position has been established in step 4, Beta angle is calculated as the angle between these two vectors.

In this example, the angle is $\text{Tan}^{-1}(1/2) = 26.5651$ degrees

Notes

- a. The value for E must always be given first in the Constants list for each member/element.
- b. All numerical values must be provided in the current units.
- c. It is not necessary or possible to specify the units of temperature or ALPHA. The user must ensure that the value provided for ALPHA is consistent in terms of units with the value provided for temperature (see Section 5.32.6).
- d. If G is not specified, but Poisson is specified, G is calculated from $E/[2(1 + \text{Poisson})]$.
- e. If neither G nor Poisson is specified, Poisson is assumed based on E, and G is then calculated.
- f. If G and Poisson are both specified, the input value of G is used, G is not calculated in this situation.
- g. If G and Poisson are both required in the analysis, such as for the stiffness matrix of plate elements, and G is specified, but Poisson is not, then, Poisson is calculated from $[(E/2G) - 1]$.
- h. To obtain a report of the values of these terms used in the analysis, specify the command **PRINT MATERIAL PROPERTIES**.

5.26.3 Surface Constants Specification

Explained below is the command syntax for specifying constants for surface entities.

The attributes associated with the surface element, and the sections of this manual where the information may be obtained, are listed below:

Attributes	Related Sections
Surfaces Incidences	5.13.3
Openings in surface	5.13.3
Local Coordinates system for surfaces	1.6.3
Specifying sections for stress/force output	5.13.3
Property for surfaces	5.21.2
Material constants	5.26.3
Surface loading	5.32.3.4
Stress/Force output printing	5.42
Shear Wall Design	3.8.2, 5.55

General Format

SURFACE CONSTANTS

{ E f_1 | G f_2 | POISSON f_3 | DENSITY f_4 | ALPHA f_5 } { LIST
surface-list | ALL }

Where:

f_1 = specifies Young's Modulus (E)

f_2 = specifies Modulus of Rigidity (G)

f_3 = specifies Poisson's Ratio

f_4 = specifies weight density

f_5 = Co-efficient of thermal expansion

Section 5 Commands and Input Instructions

5.26 Specifying and Assigning Material Constants

In lieu of numerical values, built-in material names may be used for the above specification of constants. The built-in names are STEEL, CONCRETE and ALUMINUM. See "Specifying Constants for Members and Elements" on page 388 for the values used in the built-in materials.

Example

```
SURFACE CONSTANTS
E 3150 1 TO 4
POISSON 0.17 ALL
DENSITY 8.68E-005 1 TO 4
ALPHA 5.5E-006 1 TO 4
```

Notes

- If G is not specified, but Poisson is specified, G is calculated from $E / [2 (1 + \text{Poisson})]$.
- If neither G nor Poisson is specified, Poisson is assumed based on E, and G is then calculated.
- If G and Poisson are both specified, the input value of G is used, G is not calculated in this situation.
- If G and Poisson are both required in the analysis, such as for the stiffness matrix of plate elements, and G is specified, but Poisson is not, then, Poisson is calculated from $[(E/2G) - 1]$.
- To obtain a report of the values of these terms used in the analysis, specify the command PRINT MATERIAL PROPERTIES.

5.26.4 Modal Damping Information

To define unique modal damping ratios for every mode. STAAD.Pro allows you to specify modal damping either directly or by using Rayleigh damping (the algebraic combination of mass-proportional damping and stiffness-proportional damping).

If all modes have the same damping, then enter damping with the Define Time History Load or with the Dynamic Loading commands.

Damping may be entered here

- by specifying that **STAAD EVALUATE** each mode's damping based on the frequency of the mode and the minimum and maximum damping entered here. The formula used to evaluate the damping is given below.
- by specifying that **STAAD CALCULATE** each mode's damping based on the frequency of the mode and the mass factor, ALPHA, and the STIFFNESS factor, BETA. The formula used to calculate the damping is given below.
- explicitly for some or all modes (**EXPLICIT**).

The damping entered will be used in Time History load cases; and in Response Spectrum load cases that use the **CQC** or **ASCE4** methods and/or Spectra vs. Period curves versus damping.

General Format

DEFINE DAMPING INFORMATION

```
{ EVALUATE dmin (dmax) | CALC ALPHA c1 BETA c2 (MAX c3 MIN
c4) | EXPLICIT d1 (d2 d3 d4 ... dn ) }
```

END

Where:

dmin, dmax = the minimum and maximum damping ratios respectively to be used in the EVALUATE damping formula below.

c1, c2 = The mass-proportional damping coefficient (α) and the stiffness-proportional damping coefficient (β), respectively. Used in the CALCULATE damping formula below.

c3, c4 = specified minimum and maximum damping ratios, respectively, to be used in the CALCULATE damping formula below.

d1, d2, ... dn . = the damping ratios for each mode.

Note: Damping ratios must be in the range 0.0 through 1.0.

Evaluate Damping

The formula used for **EVALUATE** (to evaluate the damping per modal frequency) is:

Damping for the first 2 modes is set to dmin from input.

Section 5 Commands and Input Instructions

5.26 Specifying and Assigning Material Constants

Damping for modes $i = 3$ to N given d_{min} and the first two frequencies ω_1 and ω_2 and the i^{th} modal frequency ω_i .

$$A_1 = d_{min} / (\omega_1 + \omega_2)$$

$$A_0 = A_1 * \omega_1 * \omega_2$$

$$D(i) = (A_0 / \omega_i) + (A_1 * \omega_i)$$

If the resulting damping is greater than the d_{max} value of maximum damping, then d_{max} will be used.

Example:

```
DEFINE DAMPING INFORMATION
EVALUATE 0.02 0.12
END
```

for $d_{min} = .02$, $d_{max} = .12$ and the ω_i given below:

Mode	ω_i	Damping Ratio
1	3	0.0200
2	4	0.0200
3	6	0.0228568
N	100	0.1200 (calculated as .28605 then reset to maximum entered)

Calculate Damping

The formula used to calculate the damping for modes $i = 1$ to N per modal frequency based on mass and/or stiffness proportional damping (for **CALCULATE**) is:

$$D(i) = (\alpha / 2\omega_i) + (\omega_i \beta / 2)$$

If the resulting damping is greater than MAX , then MAX will be used ($MAX=1$ by default). If the resulting damping is less than MIN , then MIN will be used ($MIN=1.E-9$ by default). This is the same damping as $D = (\alpha M + \beta K)$.

Example:

```

DEFINE DAMPING INFORMATION
CALC ALPHA 1.13097 BETA 0.0013926
END

```

To get 4% damping ratio at 4 Hz and 6% damping ratio at 12 Hz

Mode	Hz	Rad/sec	Damp Ratio
1	4.0	25.133	0.04
3	12.0	75.398	0.06

$$D(i) = (\alpha / 2\omega_i) + (\omega_i \beta / 2)$$

$$0.04 = \alpha / 50.266 + 12.567 \beta$$

$$0.06 = \alpha / 150.796 + 37.699 \beta$$

$$\alpha = 1.13097$$

$$\beta = 0.0013926$$

However they are determined, the α and β terms are entered in the CALC data above. For this example calculate the damping ratio at other frequencies to see the variation in damping versus frequency.

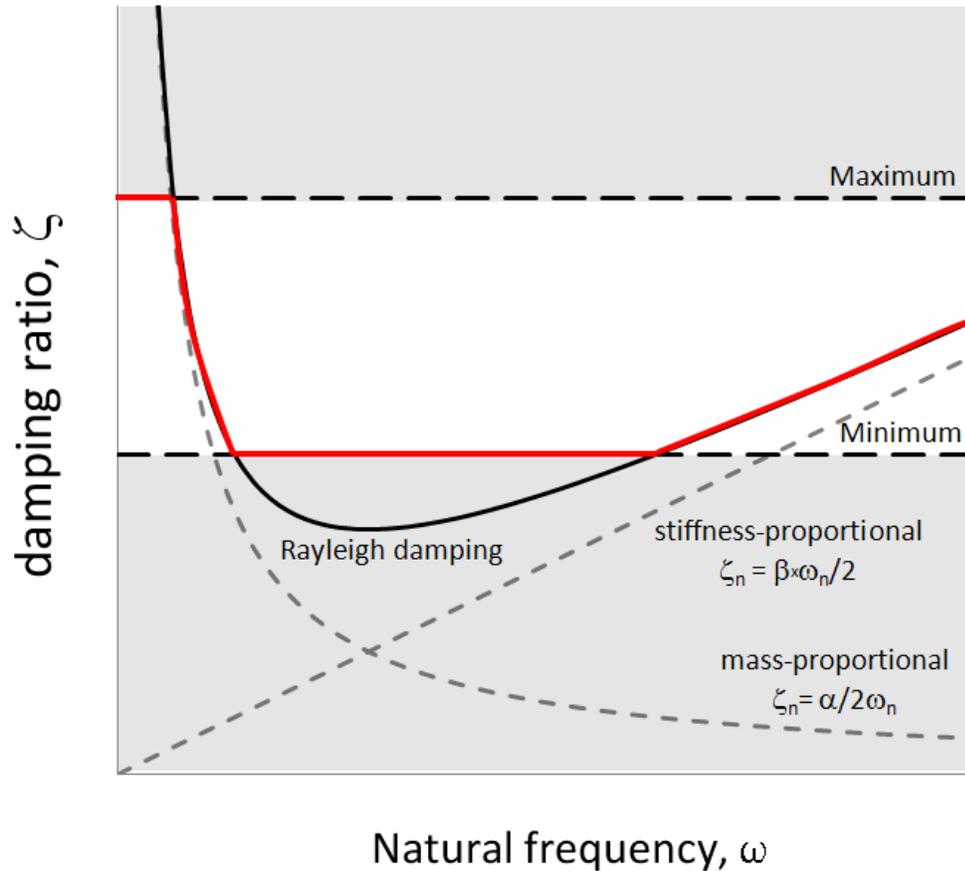
Mode	Hz	Rad/sec	Damp Ratio
1	4.0	25.133	0.040
3	12.0	75.398	0.060
	2	12.0664	0.05375
	8	50.2655	0.04650
	20	120.664	0.09200
	4.5	28.274	0.03969

The damping, due to β times stiffness, increases linearly with frequency; and the damping, due to α times mass, decreases parabolically. The combination of the two is hyperbolic.

Figure 5-24: Graph of Damping Ratio, $D(i)$, versus Natural Frequency, ω with Max. and Min. values applied.

Section 5 Commands and Input Instructions

5.26 Specifying and Assigning Material Constants



Explicit Damping

With the **EXPLICIT** option, you must provide unique modal damping values for some or all modes. Each value can be preceded by a repetition factor (rf*damp) without spaces.

Example

```
DEFINE DAMPING INFORMATION
EXPLICIT 0.03 7*0.05 0.04 -
0.012
END
```

In the above example, mode 1 damping is .03, modes 2 to 8 are .05, mode 9 is .04, mode 10 (and higher, if present) are 0.012.

If there are fewer entries than modes, then the last damping entered will apply to the remaining modes. This input may be continued to 10 more input lines with

word **EXPLICIT** only on line 1; end all but last line with a space then a hyphen. There may be additional sets of **EXPLICIT** lines before the **END**.

5.26.5 Composite Damping for Springs

This command may be used to designate certain support springs as contributing to the computation of modal damping by the composite damping method. The Response Spectrum or Time History dynamic response analyses must select composite damping for this data to have any effect on results.

General Format

SPRING DAMPING

```
joint-list *{ KFX f1 | KFY f2 | KFZ f3 }
```

Where:

f_1, f_2, f_3 = damping ratios (0.001 to 0.990) in the local X, Y, and Z directions, respectively.

Note: At least one of **KFX**, **KFY**, or **KFZ** must be entered and each one entered must have a spring damp value following it.

Description

If this Spring Damping command is entered, then all springs in the structure are included in the composite damping calculation, otherwise no spring is considered in that calculation.

This input command does not create a spring. Rather, if a support spring exists at the joint in the specified direction, then it will be assigned the damping ratio. See "5.27 Support Specifications" on page 405 to define springs.

Note: This is not a discrete damper definition.

5.26.6 Member Imperfection Information

To define camber and drift specifications for selected members. Drift is usually for columns and camber for beams.

General Format

DEFINE IMPERFECTION

Section 5 Commands and Input Instructions

5.26 Specifying and Assigning Material Constants

```
CAMBER { Y | Z } (f1) RESPECT (f2) *{ XR f4 f5 | YR f4 f5  
| ZR f4 f5 | MEM memb-list | LIST memb-list | ALL }  
DRIFT { Y | Z } (f3) *{ XR f4 f5 | YR f4 f5 | ZR f4 f5 |  
MEM memb-list | LIST memb-list | ALL }
```

Where:

f₁ = Camber value. Default = 300.

f₂ = Respect value. Default = 1.6 .

f₃ = Drift value. Default = 200.

f₄, f₅ = global coordinate values to specify X, Y, or Z range for member selection.

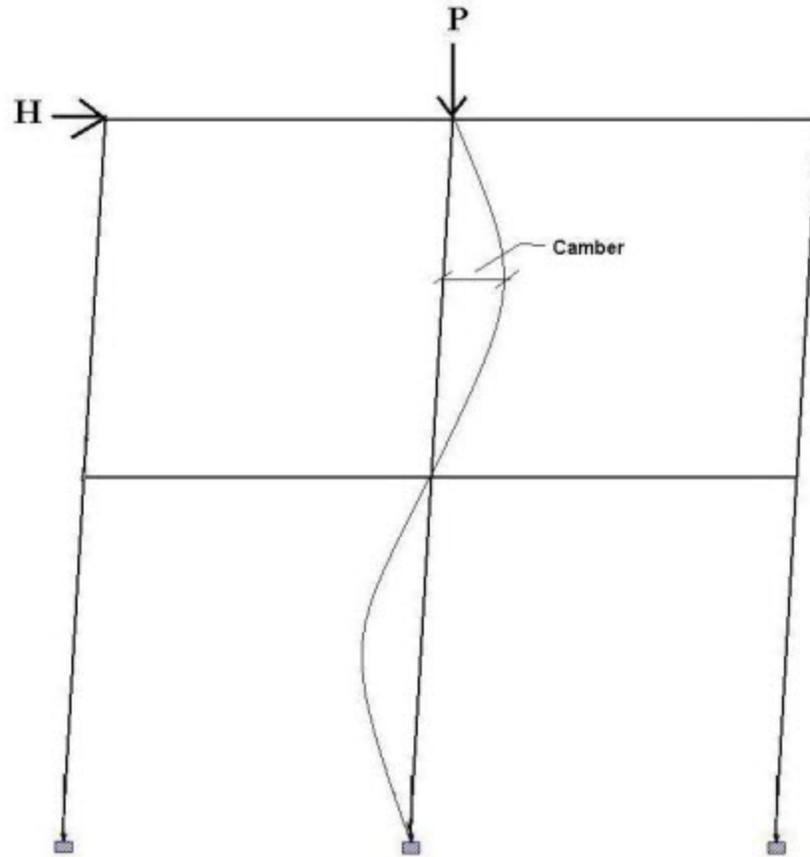
Imperfections will be simulated by loads. These loads will be generated for the specified members if there is an Imperfection Analysis specified and if the specified members are active, in compression, and are not truss or tension/compression only members.

Please refer to sections 1.18.2.4 and 5.37 for more details.

Notes

Camber is the maximum offset of the neutral axes in the defined direction from a vector that passes through the ends of the beam (i.e., the local X axis) defined as the ratio of offset/member length.

Figure 5-25: Camber definition

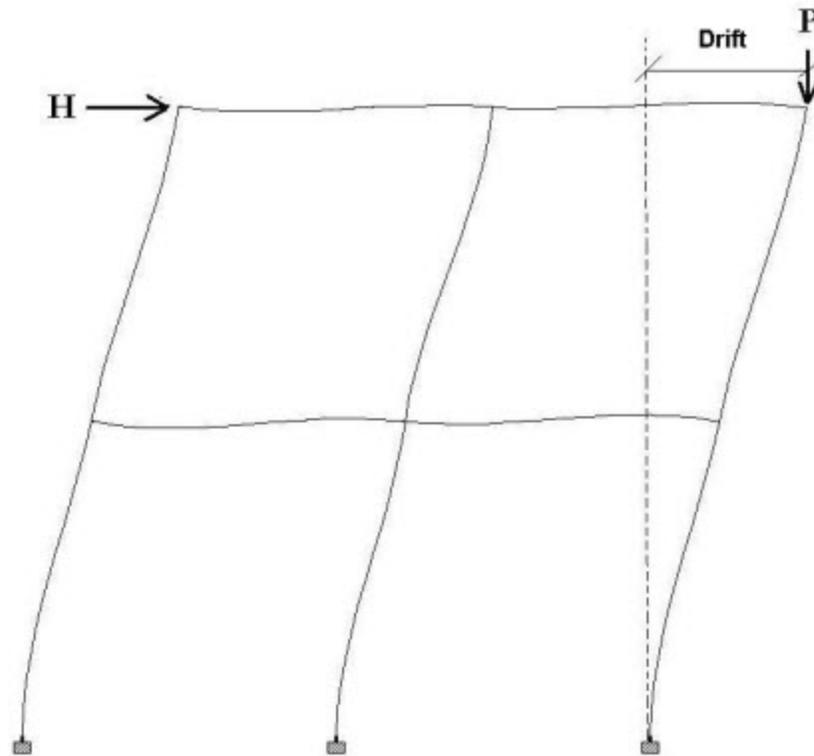


Drift is the offset of the end a member from its specified location defined as a ratio of offset/member length.

Figure 5-26: Drift definition

Section 5 Commands and Input Instructions

5.26 Specifying and Assigning Material Constants



RESPECT is a non dimensional constant used to skip the camber imperfection calculation if the compressive load is small or EI is great or length is short. A combination of these terms is calculated and called **EPSILON**. If **EPSILON** is less than **RESPECT**, then the imperfection calculation is skipped for that local direction, for that case, for that member. The imperfection calculation is also skipped for any member that is in tension.

$$\text{EPSILON}_y = \text{Length} * \text{SQRT}[(\text{abs}(\text{axial load})) / \text{EI}_z]$$

$$\text{EPSILON}_z = \text{Length} * \text{SQRT}[(\text{abs}(\text{axial load})) / \text{EI}_y]$$

Example

```
SUPPORTS
1 FIXED
2 FIXED BUT FX
DEFINE IMPERFECTION
CAMBER Y 300 RESPECT 0.4 ALL
LOAD 1 GRAVITY + LATERAL
```

```

MEMBER LOAD
1 UNI GY -1
JOINT LOAD
2 FX -10
PERFORM IMPERFECTION ANALYSIS PRINT STATICS CHECK

```

5.27 Support Specifications

STAAD support specifications may be either parallel or inclined to the global axes. See "Global Support Specification" on page 405 for specification of supports parallel to the global axes. See "5.27.2 Inclined Support Specification" on page 407 for specification of inclined supports.

5.27.1 Global Support Specification

This set of commands may be used to specify the SUPPORT conditions for supports parallel to the global axes.

For SURFACE elements, if nodes located along a straight line are all supported identically, as in the case of the base of a wall, support generation can be performed for assigning restraints to those nodes. See the "GENERATE" option in the command syntax below. You only need to provide the starting and ending nodes of the range, and the type of restraint.

General Format

SUPPORTS

```
{ joint-list | ni TO nj GENERATE } { PINNED | FIXED ( BUT
release-spec (spring-spec) ) | ENFORCED ( BUT release-spec)
}
```

```
release-spec = { FX | FY | FZ | MX | MY | MZ }
```

```
spring-spec = *{KFX f1 | KFY f2 | KFZ f3 | KMX f4 | KMY f5
| KMZ f6 }
```

Where:

n_i, n_j = Start and end node numbers, respectively, for generating supports along a SURFACE element edge.

$f_1 \dots f_6$ = Spring constants corresponding to support spring directions $X, Y,$ and Z and rotations about $X, Y,$ and $Z,$ respectively.

Description of Pinned and Fixed

A **PINNED** support is a support that has translational, but no rotational restraints. In other words, the support has no moment carrying capacity. A **FIXED** support has both translational and rotational restraints. A **FIXED BUT** support can be released in the global directions as described in release-spec (FX for force-X through MZ for moment-Z). Also, a **FIXED BUT** support can have spring constants as described in spring-spec (translational spring in global X-axis as KFX through rotational spring in global Z-axis as KMZ). Corresponding spring constants are f1 through f6. The rotational spring constants are always per degree of rotation. All six releases may be provided as may be required when using the **CHANGE** command. If both release specifications and spring specifications are to be supplied for the same support joint, release specifications must come first.

1. See "Change Specification" on page 709 for information on specification of **SUPPORTS** along with the **CHANGE** command specifications.
2. Spring constants must be provided in the current units.
3. All spring DOF must be entered after the last non-spring DOF is specified, if both are on the same line.
4. If there are two entries for the same joint, then:
 - a. any direction that is pinned/fixed on either will be fixed in that direction.
 - b. any direction released on one and is a spring on the other will use the spring.
 - c. any direction that is pinned/fixed on one and a spring on the other will use the spring.
 - d. any direction that is a spring on two or more entries will have the spring constants added.

Example 1

```
SUPPORTS
1 TO 4 7 PINNED
5 6 FIXED BUT FX MZ
8 9 FIXED BUT MZ KFX 50.0 KFY 75.
18 21 FIXED
```

```
27 FIXED BUT KFY 125.0
```

In this example, joints 1 to 4 and joint 7 are pinned. No moments are carried by those supports. Joints 5 and 6 are fixed for all DOF except in force-X and moment-Z. Joints 8 and 9 are fixed for all DOF except moment-Z and have springs in the global X and Y directions with corresponding spring constants of 50 and 75 units respectively. Joints 18 and 21 are fixed for all translational and rotational degrees of freedom. At joint 27, all the DOF are fixed except the FY DOF where it has a spring with 125 units spring constant.

Description of Enforced

Enforced Support defines which translational and rotational directions, at a joint, may have a support displacement imposed. The support displacements are defined for each load case in section 5.32.8. If no support displacement is entered, then zero displacement will be imposed, as if that direction was FIXED. The enforced displacement directions will be fixed for dynamic load cases.

If there are two entries for the same joint, then any direction that is enforced on either will be enforced in that direction, overriding any other support specification for that joint-direction.

Currently the support generation command can only be used in conjunction with the Surface element support specifications.

Example 2

```
SUPPORTS
3 TO 7 GENERATE PIN
```

The above command will generate pinned supports for all joints located between nodes No. 3 and 7 along a straight line. This may include joints explicitly defined by the user or joints generated by the program for internal use only (e.g., as a result of SET DIVISION and SURFACE INCIDENCES commands).

5.27.2 Inclined Support Specification

These commands may be used to specify supports that are inclined with respect to the global axes.

General Format

SUPPORT

Section 5 Commands and Input Instructions

5.27 Support Specifications

```
joint-list INCLINED {  $f_1$   $f_2$   $f_3$  | REF  $f_4$   $f_5$   $f_6$  | REFJT  $f_7$  } {  
PINNED | FIXED ( BUT release-spec (spring-spec) ) | ENFORCED  
( BUT release-spec ) }
```

Where:

f_1, f_2, f_3 = x, y, z global distances from the joint to the reference point
 f_4, f_5, f_6 = x, y, z global coordinates of the reference point
 f_7 = a joint number whose x, y, z global coordinates is the reference point.

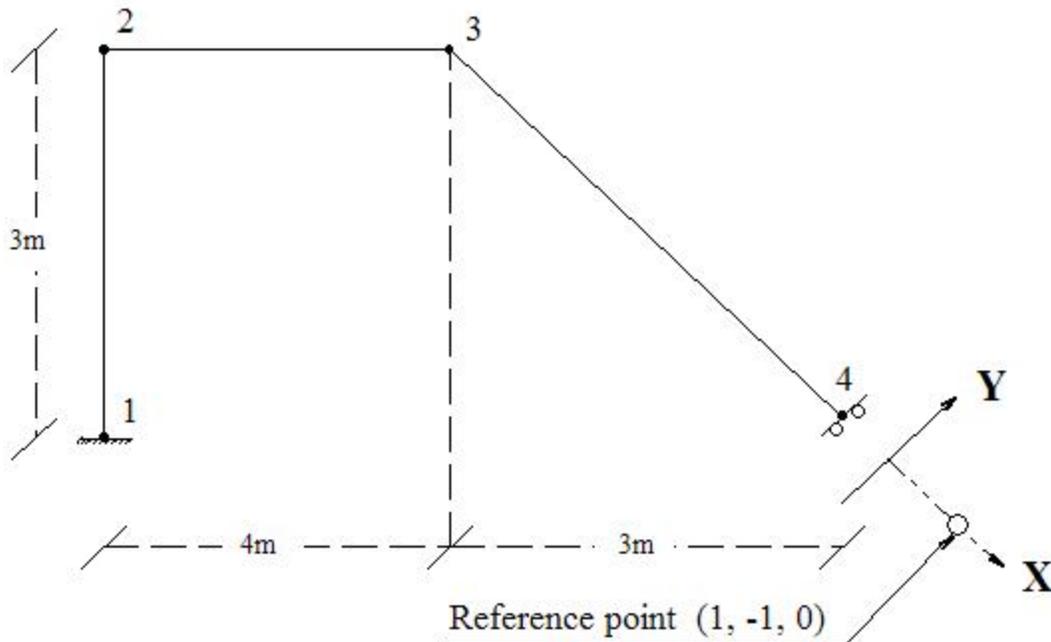
A vector from the joint location to the reference point location serves to define a local coordinate system (same as member with **BETA** = 0). The inclined support directions are in this local "Inclined Support Axis System" (see more below).

Note: The release-spec and spring-spec are the same as in the Global Supports (See "5.27.1 Global Support Specification" on page 405). However, FX through MZ and KFX through KMZ refer to forces/moments and spring constants in the "Inclined Support Axis System" (see below).

Inclined Support Axis System

The **INCLINED SUPPORT** specification is based on the "Inclined Support axis system". The local x-axis of this system is defined by assuming the inclined support joint as the origin and joining it with a "reference point" with coordinates of f_1, f_2 and f_3 (see figure 5.18) measured from the inclined support joint in the global coordinate system.

Figure 5-27: Reference point for defining an inclined support angle



The Y and Z axes of the inclined support axis system have the same orientation as the local Y and Z axes of an imaginary member whose **BETA** angle is zero and whose incidences are defined from the inclined support joint to the reference point. See "Relationship Between Global & Local Coordinates" on page 15 of this manual for more information on these concepts.

Note: Inclined support directions are assumed to be same as global when computing some dynamic and UBC intermediate results (e.g., global participation factors). If masses and/or forces in the free directions at inclined supports are a relatively small portion of the overall forces, then the effect should be very small.

Example

SUPPORT

```
4 INCLINED 1.0 -1.0 0.0 FIXED BUT FY MX MY MZ
```

5.27.3 Automatic Spring Support Generator for Foundations

STAAD has a facility for automatic generation of spring supports to model footings and foundation mats. This command is specified under the SUPPORT command.

General Format

SUPPORT

```
{ joint-list ELASTIC FOOTING f1 (f2) | joint-list ELASTIC
MAT | plate-list PLATE MAT } DIR { X | XONLY | Y | YONLY |
Z | ZONLY } SUBGRADE f3
(PRINT) ( {COMP | MULTI } )
```

or

```
plate-list PLATE MAT DIR ALL SUBGRADE f4 (f5 f6)
(PRINT) ( {COMP | MULTI } )
```

Where:

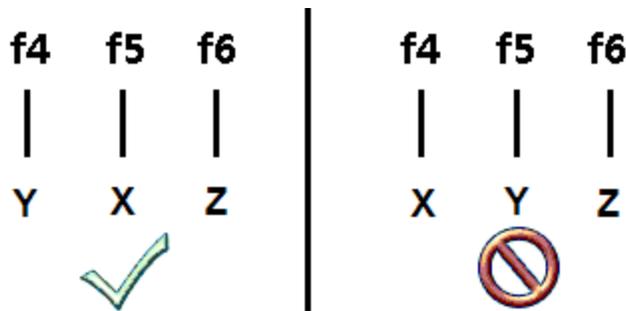
f₁, f₂ = Length and width of the **ELASTIC** footing. If f₂ is not given, the footing is assumed to be a square with sides f₁

X,Y,Z = Global direction in which soil springs are to be generated

f₃ = Soil sub-grade modulus in current force/area/length units

f₄, f₅, f₆ = Soil sub-grade modulus for use with **ALL** option in current force/area/length units in Y, X, Z directions respectively. f₄, f₅ default to f₃ if omitted. Note the order in which the Y value is specified first, followed by X and then Z.

Figure 5-28: Correct and Incorrect order for specifying subgrade modulus values



Warning: Do not use this command with **SET Z UP**.

Description

If you want to specify the influence area of a joint yourself and have STAAD simply multiply the area you specified by the sub-grade modulus, use the

ELASTIC FOOTING option. Situations where this may be appropriate are such as when a spread footing is located beneath a joint where you want to specify a spring support. A value for f_1 (and f_2 if its a non-square footing) is required for the FOOTING option.

If you want to have STAAD calculate the influence area for the joint (instead of you specifying an area yourself) and use that area along with the sub-grade modulus to determine the spring stiffness value, use the **ELASTIC MAT** option. Situations where this may be appropriate are such as when a slab is on soil and carries the weight of the structure above. You may have modeled the entire slab as finite elements and wish to generate spring supports at the nodes of the elements.

The **PLATE MAT** option is similar to the Elastic Mat except for the method used to compute the influence area for the joints. If your mat consists of plate elements and all of the influence areas are incorporated in the plate areas, then this option is preferable. Enter a list of plates or YRANGE f_1 f_2 at the beginning of the command, the joint influence areas are then calculated using the same principles as joint forces would be from uniform pressure on these plates. This method overcomes a major limitation of the Delaunay triangle method used in the ELASTIC MAT option, which is that the contour formed by the nodes of the mat must form a convex hull.

The **PLATE MAT DIR ALL** option is similar to the Plate Mat except that the spring supports are in 3 directions. If the compression only option is also specified, then the compression direction will be assumed to be in the Y direction. If the Y spring at a joint goes slack (lift off), then the X and Z spring stiffnesses for that joint will also be set to zero. Otherwise the X and Z springs act in both directions. The influence area for the X and Z springs is the same as used for the Y spring. Three values of subgrade reaction may be entered, the first is for the Y direction, the second for X and the third for Z.

The keyword **DIR** is followed by one of the alphabets X, Y or Z (or XONLY, YONLY, or ZONLY) which indicate the direction of resistance of the spring supports. If X or Y or Z is selected then a spring support is generated in that direction plus 3 other directions receive a fixed support, e.g., if Y is selected, then FY is supported by a spring; FX and FZ and MY are fixed supports; and MX and MZ are free. If XONLY, YONLY, or ZONLY are selected then only a spring support in that direction is generated.

The keyword **SUBGRADE** is followed by the value of the subgrade reaction. The value should be provided in the current unit system signified by the most recent UNIT statement prior to the SUPPORT command.

The **PRINT** option prints the influence area of each joint.

Section 5 Commands and Input Instructions

5.27 Support Specifications

Use the **COMP** option generated will be compression only springs

Use the **MULTI** option to generate multilinear springs. Add the associated multilinear curve input after each MAT command (with the multi option) to describe the displacement-spring constant curve. See section 5.27.4 for additional information on this input format. The actual spring constant used will be the subgrade modulus (f_3 entered above) times the influence area (computed by STAAD) times the s_i values entered in the curve (so the curve stiffness values will likely be between 0.0 and 1.0).

```
SPRINGS d1 s1 d2 s2 ... dn sn
```

Example

```
SUPPORTS
1 TO 126 ELASTIC MAT DIREC Y SUBG 200.
1 TO 100 PLATE MAT DIREC Y SUBG 200.
YR -.01 .01 PLA MAT DIR Y SUBG 200 MUL
SPRINGS -0.51 40.0 -0.50 50.0 0.5 65.0
```

The first command above instructs STAAD to internally generate supports for all nodes 1 through 126 with elastic springs. STAAD first calculates the influence area perpendicular to the global Y axis of each node and then multiplies the corresponding influence area by the soil subgrade modulus of 200.0 to calculate the spring constant to be applied to the node. In the 2nd example, the nodes of plates 1 to 100 are assigned spring supports, generated using a subgrade modulus of 200 units.

Notes

- A closed surface is generated by the program based on the joint-list that accompanies the ELASTIC MAT command. The area within this closed surface is determined and the share of this area for each node in the list is then calculated.

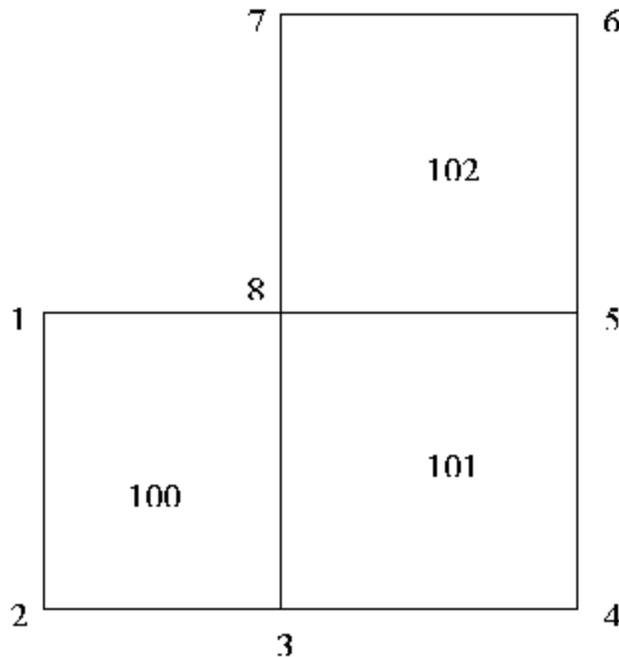
Hence, while specifying the joint-list, one should make sure that these joints make up a closed surface. Without a proper closed surface, the area calculated for the region may be indeterminate and the spring constant values may be erroneous. Consequently, the list should have at a minimum, 3 nodes.

- b. The internal angle formed by 2 adjacent segments connecting 3 consecutive nodes in the list should be less than 180 degrees. In other words, the region should have the shape of a convex polygon. The example below explains the method that may be used to get around a situation where a convex polygon is not available.
- c. For the model comprised of plate elements 100 to 102 in the figure below, one wishes to generate the spring supports at nodes 1 to 8. However, a single ELASTIC MAT command will not suffice because the internal angle between the edges 1-8 and 8-7 at node 8 is 270 degrees, which violates the requirements of a convex polygon.

So, you should break it up into two commands:

```
1 2 3 8 ELASTIC MAT DIREC Y SUBG 200.
3 4 5 6 7 8 ELASTIC MAT DIREC Y SUBG 200.
```

Figure 5-29: Example for elastic mat generation for a convex polygonal shape



Joints 3 and 8 will hence get the contribution from both of the above commands.

The command works only when the plane of the closed region is parallel to one of the global planes X-Y, Y-Z or X-Z. For regions that are inclined to

Section 5 Commands and Input Instructions

5.27 Support Specifications

one of the global planes, the spring constant will have to be evaluated manually and specified using the FIXED BUT type of spring support.

5.27.4 Multilinear Spring Support Specification

When soil is modeled as spring supports, the varying resistance it offers to external loads can be modeled using this facility, such as when its behavior in tension differs from its behavior in compression.

General Format

MULTILINEAR SPRINGS

joint-list SPRINGS d_1 s_1 d_2 s_2 ... d_n s_n

Where:

d_i , s_i pairs represent displacement and spring constant pairs (s_i is zero or positive), starting from the maximum negative displacement to the maximum positive displacement.

The first pair defines the spring constant from negative infinity displacement up to the displacement defined in the second pair. The second pair define the spring constant when the support displaces in the range from the displacement defined in the second pair, up to the displacement defined in the third pair. This continues for each displacement and spring constant pair until the last pair which defines the spring constant for displacements greater than the displacement in the last pair to positive infinity.

Each load case in a multi-linear analysis must be separated by a **CHANGE** command and have its own **PERFORM ANALYSIS** command. There may not be any **PDELTA**, **NONLIN**, dynamics, **CABLE**, or **TENSION/COMPRESSION** analysis included. The multi-linear spring command will initiate an iterative analysis and convergence check cycle. The cycles will continue until the root mean square (RMS) of the effective spring rates used remain virtually the same for two consecutive cycles.

Example

```
UNIT ...  
SUPPORT  
1 PINNED; 2 4 FIXED BUT KFY 40.0
```

MULTILINEAR SPRINGS

```
2 4 SPRINGS -1 40.0 -0.50 50.0 0.5 65.0
```

Load-Displacement characteristics of soil can be represented by a multi-linear curve. Amplitude of this curve will represent the spring characteristic of the soil at different displacement values. A typical spring characteristic of soil may be represented as the step curve as shown in the figure below. In the above example, the multi-linear spring command specifies soil spring at joints 2 and 4. (Note that the amplitude of the step curve does not change after the first point.)

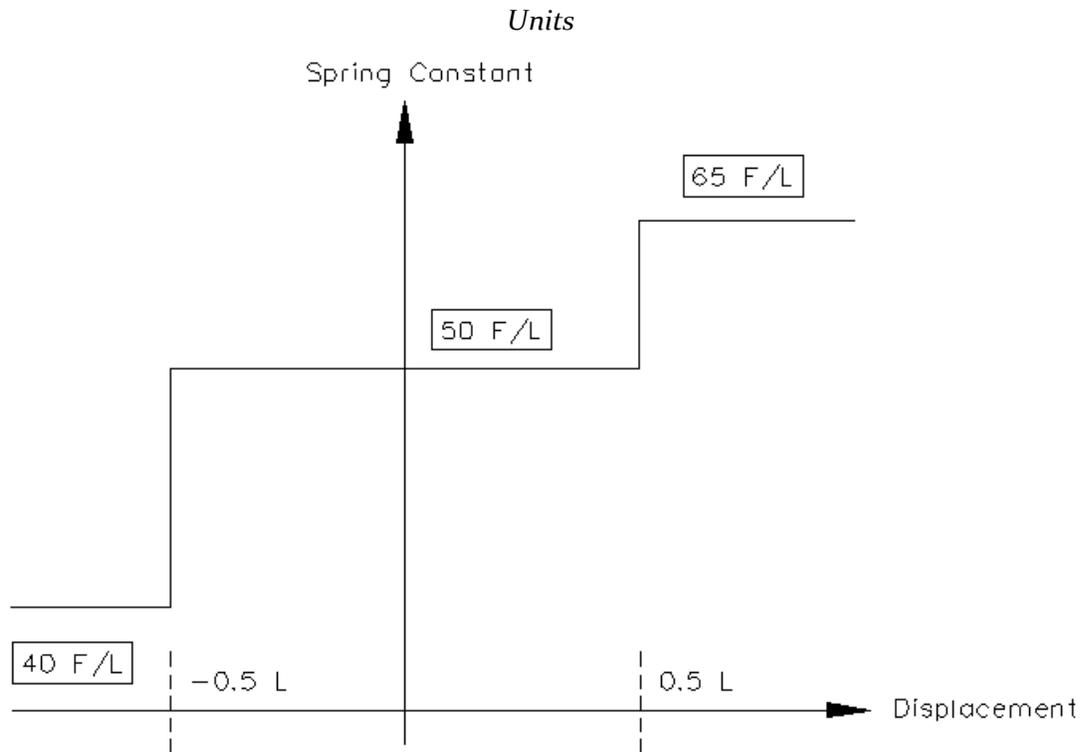
Notes

- a. SUPPORT springs must have previously been entered for each spring entered here. For the first cycle, the spring value used will be the support spring value (not the zero displacement value here). Use a realistic and stable value.
- b. All directions that have been defined with an initial spring stiffness in the SUPPORT command will become multi-linear with this one curve.
- c. This command can be continued to up to 11 lines by ending all but last with a hyphen. The semi-colons and the X RANGE, Y RANGE, Z RANGE list items may not be used.
- d. This command needs a minimum of two displacement and spring constant pairs.

Figure 5-30: Spring constant is always positive or zero. F = Force Units, L = Length

Section 5 Commands and Input Instructions

5.27 Support Specifications



5.27.5 Spring Tension/Compression Specification

This command may be used to designate certain support springs as Tension-only or Compression-only springs.

General Format

SPRING TENSION

joint-list (spring-spec)

SPRING COMPRESSION

joint-list (spring-spec)

spring-spec = *{ KFX | KFY | KFZ | ALL }

Description

Tension-only springs are capable of carrying tensile forces only. Thus, they are automatically inactivated for load cases that create compression in them.

Compression-only springs are capable of carrying compressive forces only. Thus, they are automatically inactivated for load cases that create tension in them.

If no spring spec is entered or ALL is entered then all translational springs at that joint will be tension (or compression) only. This input command does not create a spring, only that if a support spring exists at the joint in the specified direction then it will also be tension (or compression) only. See "Support Specifications" on page 405 to define springs.

For compression only springs the **ALL** parameter has special meaning. The compression only spring is in the Y direction; the X and Z direction springs are bi-directional. However when the Y direction spring goes slack, the X and Z springs at the same joint are made inactive as well.

Note: The procedure for analysis of Tension-only or Compression-only springs requires iterations for every load case and therefore may be quite involved.

Since this command does not specify whether the spring is in the positive or negative direction from the joint, it is assumed in STAAD to be in the negative direction. For negative displacement the spring is in compression and for positive the spring is in tension.

If a **CHANGE** command is used (because of a change in list of tension springs, supports, etc.), then the **SET NL** command must be used to convey to STAAD that multiple analyses and multiple structural conditions are involved.

1. See "5.5 Set Command Specification" on page 284 for explanation of the **SET NL** command. The number that follows this command is an upper bound on the total number of primary load cases in the file.
2. STAAD performs up to 10 iterations automatically, stopping if converged. If not converged, a warning message will be in the output. Enter a **SET ITERLIM i** command (with **i** > 10) before the first load case to increase the default number of iterations. Since convergence may not be possible using this procedure, do not set the limit too high. If not converged, a message will be in the output.
3. The principle used in the analysis is the following.
 - The program reads the list of springs declared as **SPRING TENSION** and/or **COMPRESSION**.
 - The analysis is performed for the entire structure and the spring forces are computed.
 - For the springs declared as **SPRING TENSION / COMPRESSION**, the program checks the axial force to determine whether it is tensile or

Section 5 Commands and Input Instructions

5.27 Support Specifications

compressive. Positive displacement is **TENSION**. If the spring cannot take that load, the spring is "switched off" from the structure.

- The analysis is performed again without the switched off springs.
 - Up to **ITERLIM** iterations of the above steps are made for each load case.
 - This method does not always converge and may become unstable. Check the output for instability messages. Do not use results if the last iteration was unstable. You may need to include some support in each global direction that is not tension (or compression) only to be stable on every iteration.
4. A revised **SPRING TENSION / COMPRESSION** command and its accompanying list of joints may be provided after a **CHANGE** command. If entered, the new **SPRING** commands replace all prior **SPRING** commands. If not entered after a **CHANGE**, then the previous spring definitions are used.
 5. The **SPRING TENSION** command should not be used if the following load cases are present: Response Spectrum load case, Time History Load case, Moving Load case. If used, the **SPRING TENSION / COMPRESSION** will be ignored in all load cases.
 6. If the **SPRING TENSION / COMPRESSION** command is used in a model with UBC, IBC or other such seismic load cases, each such load case must be followed by an **ANALYSIS** and **CHANGE** command.

Notes

- a. A spring declared as tension-only or compression-only will carry axial forces only. It will not carry moments.
- b. The **SPRING TENSION / COMPRESSION** commands should not be specified if the **INACTIVE MEMBER** command is specified.
- c. Do not use Load Combination to combine these cases. Tension/Compression cases are non-linear and should not be linearly combined as in Load Combination. Use a primary load case with the Repeat Load command.

Example

```
SPRING TENSION
```

```
12 17 19 TO 37 65
SPRING COMPRESSION
5 13 46 TO 53 87 KFY
```

The following is the general sequence of commands in the input file if the **SPRING TENSION** or **COMPRESSION** command is used. This example is for the **SPRING TENSION** command. Similar rules are applicable for the **SPRING COMPRESSION** command. The dots indicate other input data items.

```
STAAD ...
SET NL ...
UNITS ...
JOINT COORDINATES
...
MEMBER INCIDENCES
...
ELEMENT INCIDENCES
...
CONSTANTS
...
MEMBER PROPERTY
...
ELEMENT PROPERTY
...
SUPPORTS
...
SPRING TENSION
...
LOAD 1
...
LOAD 2
...
LOAD 3
```

Section 5 Commands and Input Instructions

5.28 Rigid Diaphragm Modeling

```
REPEAT LOAD
...
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT ...
PRINT ...
PARAMETER
...
CHECK CODE ...
FINISH
```

5.28 Rigid Diaphragm Modeling

STAAD.Pro has two methods for defining rigid floor diaphragms. Both are functionally equivalent but the Master-Slave joint requires that an analytical node be located and specified as the master node. For most models, the simpler Floor Diaphragm option is preferred.

5.28.1 Master/Slave Specification

This set of commands may be used to model specialized linkages (displacement tying, rigid links) through the specification of MASTER and SLAVE joints. Please read the notes for restrictions.

General Format

```
SLAVE *{ XY | YZ | ZX | RIGID | FX | FY | FZ | MX | MY | MZ
} MASTER j JOINT joint-spec
```

Where:

```
joint-spec = { joint-list or *{ XRANGE | YRANGE | ZRANGE }
f1, f2 }
```

Description

The master/slave option provided in STAAD allows the user to model specialized linkages (displacement tying, rigid links) in the system. For example, SLAVE FY ...

connects the two joints such that the Y displacement at the slave will be the sum of Y displacement at the master plus the rigid rotation,

$$R \sin \theta$$

Notice that instead of providing a joint list for the slaved joints, a range of coordinate values (in global system) may be used. All joints whose coordinates are within the range are assumed to be slaved joints. For convenience, the coordinate range specified for slaved joints in this entry may include the master joint for this entry. However, master and slave joints of other entries must not be included in the coordinate range. All 2 or 3 ranges can be entered to form a “tube” or “box” for selecting joints in the tube or box region.

Fx, Fy etc. are the directions in which they are slaved (any combination may be entered).

If two or more entries have the same master, the slave lists will be merged. Please ensure that the same direction specs are used.

The direction specifiers (XY, YZ, ZX) are combinations of the basic directions, XY is the same as entering FX, FY, MZ; etc. Any combination of direction specifiers may be entered. An example of the use of this format is: a rigid diaphragm floor that still retains bending flexibility entered as SLA ZX

If RIGID or if all directions are provided, the joints are assumed to be rigidly connected as if SLA DIA RIG were entered, even if DIA is omitted.

Restrictions

- Solid elements may not be connected to slave joints.
- Master joints may not be master or slaves in another entry.
- Slave joints may not be master or slaves in another entry.
- Slave directions at joints may not be supported directions or have displacements imposed.
- Master and/or slave joints may not be inclined supports.
- The master / slave specification is only intended for linear static and dynamic analysis.
- Multilinear springs are not permitted.

Example

Fully Rigid and Rigid Floor Diaphragm

Section 5 Commands and Input Instructions

5.28 Rigid Diaphragm Modeling

```
SLAVE RIGID MASTER 22 JOINT 10 TO 45
SLAVE RIGID MASTER 70 JOINT YRANGE 25.5 27.5
SLA ZX MAS 80 JOINT YR 34.5 35.5
```

In STAAD.Pro 2007 Build 1001, the internal processing of any Master Slave command was enhanced to allow an automatic bandwidth reduction to take place.

The analysis engine now performs a bandwidth reduction on files that include Master/Slave commands which must occur in the input file after the definition of supports. In previous versions of STAAD.Pro, for the bandwidth reduction to take place, the data of Master/Slave would need to be repeated before the support definitions. This requirement is now no longer required.

5.28.2 Floor Diaphragm

This command is used to create rigid floor diaphragms without the need to specify a master joint at each. When specified, this command directs the engine to perform the following:

- a. calculate the center of mass for each rigid diaphragm (where master joint is to be located) considering the mass model of the structure. The mass must be modeled using mass reference load. See "Mass Modeling Using Reference Loads" on page 533
- b. create, internally, an analytical node at the center of mass location to be included during analysis (unless a master node is specified) if an existing analytical node exists at this point, then the existing joint is used in lieu of creating a new joint.
- c. search all nodes available within a diaphragm and add them as slave nodes; with the master node located at the center of mass for the diaphragm (or at the specified master node)

Note: This feature requires STAAD.Pro V8i (SELECTseries 3) (release 20.07.08) or higher.

General Format

FLOOR DIAPHRAGM

DIAPHRAGM i_1 **TYPE RIGID** *diaphragm-spec*

After all diaphragms are defined, then:

(CHECK STORY DRIFT [1893 | ASCE7])

Where:

diaphragm-spec = HEIGHT f_1 (MASTER i_2) (JOINT *joint-list*)

or

diaphragm-spec = YRANGE f_2 f_3 (MASTER i_2) (JOINT XRANGE f_4 f_5 ZRANGE f_6 f_7)

or

diaphragm-spec = (MASTER i_2) JOINT *joint-list*

i_1 = Diaphragm identification number.

f_1 = Global coordinate value, in Y direction, to specify floor level.

f_2, f_3 = Global coordinate values to specify a Y range. The diaphragm is considered to be located at that floor height.

f_4, f_5 = Global coordinate values to specify an X range. The diaphragm is considered to be located between this X range. If full floor is to be considered as only one diaphragm there is no need to define X range.

f_6, f_7 = Global coordinate values to specify Z range. The diaphragm is considered to be located between this Z range. If full floor is to be considered as only one diaphragm there is no need to define Z range.

i_2 = User specified master joint number at the specified floor level. If not defined, the program will automatically calculate this joint as the diaphragm center of mass.

Instead of providing height or Y-range, joint lists can be provided to indicate the number of joints present at a particular floor level which will be connected to a master joint (either specified or calculated by the program).

Soft Story Checks

The option check for story drift can be performed per the IS 1893:2002 or ASCE 7-95 codes.

Note: This feature requires STAAD.Pro V8i (SELECTseries 4) (release 20.07.09) or higher.

A soft story building is a multi-story building where one or more floors are soft due to structural design. These floors can be dangerous in earthquakes, because they cannot cope with the lateral forces caused by the swaying of the building

Section 5 Commands and Input Instructions

5.28 Rigid Diaphragm Modeling

during a quake. As a result, the soft story may fail, causing what is known as a soft story collapse.

Soft story buildings are characterized by having a story which has a lot of open space. Parking garages, for example, are often soft stories, as are large retail spaces or floors with a lot of windows. While the unobstructed space of the soft story might be aesthetically or commercially desirable, it also means that there are less opportunities to install shear walls, specialized walls which are designed to distribute lateral forces so that a building can cope with the swaying characteristic of an earthquake.

If a building has a floor which is 70% less stiff than the floor above it, it is considered a soft story building. This soft story creates a major weak point in an earthquake, and since soft stories are mostly associated with retail spaces and parking garages, they are often on the lower stories of a building, which means that when they collapse, they can take the whole building down with them, causing serious structural damage which may render the structure totally unusable.

Notes

- a. Full diaphragm definition should be provided in one line. Second diaphragm definition should be provided in second line. However, if there is joint-list, the list can extend to the second line with a continuation sign ("-").

```
DIA F1 TYP RIG YR F2 F3 JOI XR F4 F5 ZR F6 F7
DIA F11 TYP RIG YR F21 F31 JOI XR F41 F51 ZR F61 F71
DIA F12 TYP RIG JOI 35 45 51 TO 57 59 TO 83 -
90 TO 110
```

where f_1 , f_{11} and f_{12} are three rigid diaphragms located at floor height ranging between f_2 and f_3 , f_{21} and f_{31} and the joints lying in the plane as indicated by their global Y coordinates respectively.

- b. Diaphragms should be specified in ascending order (i.e., diaphragms at first floor level should be specified first before specifying that on 2nd floor level and so on).
- c. If user defined master joint is specified in one diaphragm, user defined master joints should be specified for all diaphragms. Combination of user

defined master joint for one diaphragm and program calculated master joint for another diaphragm is not supported.

- d. Before specifying floor diaphragm mass model (in terms of reference load) must be specified.
- e. Floor diaphragm can be specified only once in an input file.
- f. Floor diaphragm cannot be specified along with **FLOOR HEIGHT** command.
- g. Floor diaphragm cannot be specified along with **MASTER-SLAVE** command.
- h. Floor diaphragm cannot be specified with **SET Z UP** command.
- i. Sloped diaphragms are not supported.
- j. Base level (or ground floor level or support level) is taken as the minimum of Y coordinates defined. Different base level can be specified using following syntax.

FLOOR DIAPHRAGM

DIA f1 TYPE RIG YR f2 f3 JOI XR f4 f5 ZR f6 f7

DIA f11 TYPE RIG YR f21 f31 JOI XR f41 f51 ZR f61 f71

...

BASE b₁

Where:

b₁ = base/ground floor level of the structure

- k. The maximum number of diaphragms allowed by the program (default value) is 150. If more than 150 diaphragms need to be specified, then **SET RIGID DIAPHRAGM n** must be specified before specifying joint incidence, where n = total number of diaphragms in the structure.

Example

```
*****
**
UNIT FEET KIP
DEFINE REF LOAD
LOAD R1 LOADTYPE MASS
* MASS MODEL
```

Section 5 Commands and Input Instructions

5.28 Rigid Diaphragm Modeling

```
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
JOINT LOAD
17 TO 48 FX 2.5 FY 2.5 FZ 2.5
49 TO 64 FX 1.25 FY 1.25 FZ 1.25
*
LOAD R2
SELFWEIGHT Y -1
JOINT LOAD
17 TO 48 FY -2.5
49 TO 64 FY -1.25
END DEF REF LOAD
*****
*****
FLOOR DIAPHRAGM
DIAPHRAGM 1 TYPE RIGID YR 4.1 4.3 JOI XR -0.1 21.3 ZR -
0.1 31.9
DIAPHRAGM 2 TYPE RIGID YR 8.3 8.5 JOI XR -0.1 21.3 ZR -
0.1 31.9
DIAPHRAGM 3 TYPE RIGID YR 12.5 12.7 JOI XR -0.1 21.3 ZR -
0.1 31.9
DIAPHRAGM 4 TYPE RIGID YR 16.7 16.9 JOI XR -0.1 21.3 ZR -
0.1 31.9
DIAPHRAGM 5 TYPE RIGID YR 20.9 21.1 JOI XR -0.1 21.3 ZR -
0.1 31.9
DIAPHRAGM 6 TYPE RIGID YR 25.1 25.3 JOI XR -0.1 21.3 ZR -
0.1 31.9
DIAPHRAGM 7 TYPE RIGID YR 29.3 29.5 JOI XR -0.1 21.3 ZR -
0.1 31.9
DIAPHRAGM 8 TYPE RIGID YR 33.5 33.7 JOI XR -0.1 21.3 ZR -
0.1 31.9
```

```
DIAPHRAGM 9 TYPE RIGID YR 37.7 37.9 JOI XR -0.1 21.3 ZR -
0.1 31.9
BASE 0.5
*****
**
```

5.29 Draw Specifications

This command has been discontinued in STAAD.Pro. Please use the Graphical User Interface for screen and hard copy graphics.

5.30 Miscellaneous Settings for Dynamic Analysis

When dynamic analysis such as frequency and mode shape calculation, response spectrum analysis and time history analysis is performed, it involves eigenvalue extraction and usage of a certain number of modes during the analysis process. These operations are built around certain default values. This section explains the commands required to override those defaults.

5.30.1 Cut-Off Frequency, Mode Shapes, or Time

These commands are used in conjunction with dynamic analysis. They may be used to specify the highest frequency or the number of mode shapes that need to be considered.

General Format

CUT (OFF) { FREQUENCY f_1 | MODE SHAPE i_1 | TIME t_1 }

Where:

f_1 = Highest frequency (cycle/sec) to be considered for dynamic analysis.

i_1 = Number of mode shapes to be considered for dynamic analysis. If the cut off frequency command is not provided, the cut off frequency will default to 108 cps. If the cut off mode shape command is not provided, the first six modes will be calculated. These commands should be provided prior to the loading specifications.

t_1 = Ending time for a time history analysis. If zero (default), the time history will end when the last forcing function ends.

Section 5 Commands and Input Instructions

5.30 Miscellaneous Settings for Dynamic Analysis

A maximum of i_1 mode shapes will be computed regard-less of f_1 . If during convergence testing, the 0 through f_1 frequencies are converged, then the modal calculation will be completed before i_1 mode shapes are calculated.

If the **CUT OFF FREQ f_1** and **CUT OFF MODE i_1** commands are both entered, then after completing each iteration step in the Subspace iteration, convergence testing is performed. If every frequency from 0.0 to f_1 meets the convergence tolerance, then the Subspace iteration is done. Similarly, if every mode from 0 to i_1 meets the convergence tolerance, then the Subspace iteration is done. If the cut off frequency f_1 results in fewer modes than i_1 , then only those frequencies up to the cut off are used. If the cut off frequency would result in more modes than i_1 , then only the first i_1 modes are used. That is, the modes cut off takes precedence over the frequency cut off.

5.30.2 Mode Selection

This command allows specification of a reduced set of active dynamic modes. All modes selected by this command remain selected until a new **MODE SELECT** is specified.

This command is used to limit the modes used in dynamic analysis to the modes listed in this command and deactivate all other modes that were calculated but not listed in this command. If this command is not entered, then all modes calculated are used in the dynamic analysis.

General Format

```
MODE SELECT mode-list
```

Example

```
CUT OFF MODES 10  
MODE SELECT 1 TO 3
```

In this example, 10 modes will be calculated but only modes 1 and 3 will be used in dynamic analysis.

Notes

- Do not enter this command within the loads data (from the first Load command in an analysis set down to the associated Analysis command).

- b. The advantage of this command is that you may find the amount of structural response generated from a specific mode or a set of modes. For example, if 50 modes are extracted, but the effect of just the 40th to the 50th mode in a response spectrum analysis is to be determined, you may set the active modes to be 40 through 50. The results will then be devoid of any contribution from modes 1 through 39.

5.31 Definition of Load Systems

This section describes the specifications necessary for defining various load systems, for automatic generation of Moving loads, UBC Seismic loads and Wind loads. In addition, this section also describes the specification of Time History load for Time History analysis.

STAAD has built-in algorithms to generate moving loads, lateral seismic loads, and wind loads on a structure. Use of the load generation facility consists of two parts:

1. Definition of the load system(s).
2. Generation of primary load cases using previously defined load system(s).

Definition of the load system(s) must be provided before any primary load case is specified. This section describes the specification of load system(s). Information on how to generate primary load cases using the defined load system(s) is available in Section 5.32.12.

UBC loads do not fully consider the effects of forces at inclined support directions or at slave joint directions. Applying forces at these locations may introduce errors that are generally small.

5.31.1 Definition of Moving Load System

This set of commands may be used to define the moving load system. Enter this command only once with up to 100 TYPE commands.

The MOVING LOAD system may be defined in two possible ways: directly within the input file or using an external file.

General Format

```
DEFINE MOVING LOAD (FILE file-name)
TYPE j { LOAD f1, f2, ..., fn ( DISTANCE d1, d2, ..., dn-1 (WIDTH
w) ) | Load-name (f) }
```

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

(DISTANCE d_1, d_2, \dots, d_{n-1} (WIDTH w)) < *optionally as 2nd set* >

The FILE option should be used only in the second case when the data is to be read from an external file. The filename should be limited to 24 characters.

Note: Moving Loads can be generated for frame members only. They will not be generated for finite elements.

Note: All loads and distances are in current unit system.

Define Moving Load Within the Input File

Use the first TYPE specification. Input Data must be all in one line (as shown above) or two sets of lines. If two sets, then the second set must begin with DIS as shown above. If two sets, then Load and Dist lines may end each line but last of each set with a hyphen (See example).

TYPE j LOAD f_1, f_2, \dots, f_n
DISTANCE d_1, d_2, \dots, d_{n-1} (WIDTH w)

Where:

j = moving load system type number (integer limit of 200 types)

n = number of loads (e.g., axles), 2 to 200.

f_i = value of conc. i^{th} load

d_i = distance between the $(i+1)^{\text{th}}$ load and the i^{th} load in the direction of movement

w = spacing between loads perpendicular to the direction of movement. If left out, one dimensional loading is assumed. (e.g., the width of vehicle). This parameter will double the total load since the f_i is applied to each wheel.

Note: For a single moving load use: TYPE j LOAD $f1$ DIST 0

Define Moving Load Using an External File

Use the second TYPE specification.

TYPE j *Load-name* (f)

Where:

j = moving load system type no. (integer).

load-name = the name of the moving load system (maximum of 24 characters).

f = Optional multiplying factor to scale up or down the value of the loads. (default = 1.0)

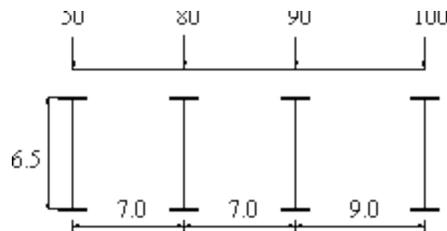
Following is a typical file containing the data.

```

CS200
50.      NAME OF LOAD SYSTEM (LOAD-NAME, MUST START IN COLUMN 1)
80. 90. 100.
7. 7. 9.      LOADS (ALL ON ONE 79 CHAR INPUT LINE)
6.5          DISTANCE BETWEEN LOADS (ONE LINE)
                                                    WIDTH

```

Figure 5-31: Graphical representation of the previous load system



Several load systems may be repeated within the same file.

The STAAD moving load generator assumes:

1. All positive loads are acting in the negative global vertical (Y or Z) direction. The user is advised to set up the structure model accordingly.
2. Resultant direction of movement is determined from the X and Z (or Y if Z is up) increments of movements as provided by the user.

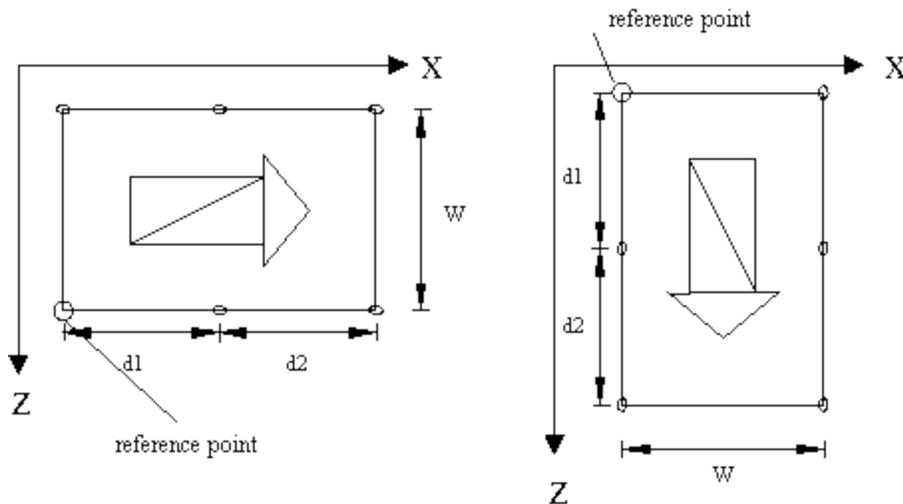
Reference Load

The first specified concentrated load in the moving load system is designated as the reference load. While generating subsequent primary load cases, the initial position of the load system and the direction of movement are defined with respect to the reference load location. Also, when selecting the reference load location with a positive value of Width specified, then the following two views define the reference load location.

Figure 5-32: Movement parallel to global X axis; Movement parallel to global Z axis

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems



Notice that in the left view, the reference point is on the positive Z wheel track side; whereas in the right view, the reference point is on the least positive X wheel track side.

Specifying Standard AASHTO Loadings

Truck loads specified in AASHTO specifications are also built in to STAAD.

TYPE *i* { HS20 | HS15 | H20 | H15 } (*f*) (*vs*)

Where:

i = moving load system type no. (integer).

f = optional multiplying factor (default = 1.0)

vs = variable spacing as defined by AASHTO, for HS series trucks (default = 14 ft.)

Example 1

```
DEFINE MOVING LOAD
TYPE 1 LOAD 10.0 20.0 -
15.0 10.0
DISANCE 5.0 7.5 -
6.5 WIDTH 6.0
TYPE 2 HS20 0.80 22.0
```

Example 2

When data is provided through an external file called MOVLOAD

Data in input file

```
UNIT ...
DEFINE MOVING LOAD FILE MOVLOAD
TYPE 1 AXLTYP1
TYPE 2 AXLTYP2 1.25
```

Data in external file MOVLOAD

```
AXLTYP1
10 20 15
5.0 7.5
6.0
AXLTYP2
20 20
10
7.5
```

5.31.2 Definitions for Static Force Procedures for Seismic Analysis

STAAD offers facilities for determining the lateral loads acting on structures due to seismic forces, using the rules available in several national codes and widely accepted publications. The codes and publications allow for so called equivalent static force methods to be used in place of more complex methods like response spectrum and time history analysis. Among the several such codes supported by STAAD are UBC, IBC, IS 1893, AIJ, etc.

Once the lateral loads are generated, the program can then analyze the structure for those loads using the applicable rules explained in the code documents.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

5.31.2.1 UBC 1997 Load Definition

This feature enables one to generate horizontal seismic loads per the UBC 97 specifications using a static equivalent approach. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the **SET Z UP** command in Section 5.5). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Methodology

The design base shear is computed in accordance with Section 1630.2.1 of the UBC 1997 code. The primary equation, namely, 30-4 of UBC 1997, as shown below, is checked.

$$V = C_v I / (RT) \cdot W$$

In addition, the following equations are checked :

Equation 30-5 - The total design base shear shall not exceed

$$V = 2.5 \cdot C_a I / R \cdot W$$

Equation 30-6 - The total design base shear shall not be less than

$$V = 0.11 \cdot C_a I W$$

Equation 30-7 - In addition, for Seismic Zone 4, the total base shear shall also not be less than

$$V = 0.8 \cdot Z N_v I / R \cdot W$$

For an explanation of the terms used in the above equations, please refer to the UBC 1997 code.

There are two stages of command specification for generating lateral loads. This is the first stage and is activated through the **DEFINE UBC LOAD** command.

Procedure Used by the Program

Steps to calculate base shear are as follows:

1. Time Period of the structure is calculated based on clause 1630.2.2.1 (Method A) and 1630.2.2.2 (Method B).

2. The user may override the period that the program calculates using Method B by specifying a value for PX or PZ (Items f9 and f10) depending on the direction of the UBC load. The specified value will be used in place of the one calculated using Method B.
3. The governing Time Period of the structure is then chosen between the above-mentioned two periods on the basis of the guidance provided in clause 1630.2.2.2.
4. From Table 16-Q and 16-R, Ca and Cv coefficients are calculated.
5. The Design Base Shear is calculated based on clause 1630.2.1 and distributed at each floor using the rules of clause 1630.5.
6. If the ACCIDENTAL option is specified, the program calculates the additional torsional moment. The lever arm for calculating the torsional moment is obtained as 5% of the building dimension at each floor level perpendicular to the direction of the UBC load (clause 1630.6). At each joint where a weight is located, the lateral seismic force acting at that joint is multiplied by this lever arm to obtain the torsional moment at that joint.
7. If the value of Ct is not specified, the program scans the Modulus of Elasticity (E) values of all members and plates to determine if the structure is made of steel, concrete or any other material. If the average E is smaller than 2000 ksi, Ct is set to 0.02. If the average E is between 2000 & 10000 ksi, Ct is set to 0.03. If the average E is greater than 10,000 ksi, Ct is set to 0.035. If the building material cannot be determined, Ct is set to 0.035. Ct is in units of seconds/feet^{3/4} or in units of seconds/meter^{3/4}. Ct < 0.42 if the units are in feet, and Ct > 0.42 if the units are in meter.
8. Due to the abstractness of the expression "Height above foundation", in STAAD, height "h" is measured above supports. If supports are staggered all over the vertical elevations of the structure, it is not possible to calculate "h" if one doesn't have a clear elevation level from where to measure "h". Also, the code deals with distributing the forces only on regions above the foundation. If there are lumped weights below the foundation, it is not clear as to how one should determine the lateral forces for those regions.

General Format

DEFINE UBC (ACCIDENTAL) LOAD

ZONE f1 *ubc-spec*

SELFWEIGHT

JOINT WEIGHT

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

joint-list WEIGHT *w*

...

Note: See "UBC 1994 or 1985 Load Definition" on page 438 for complete weight input definition.

Where:

ubc-spec = {I f2, RWX f3, RWZ f4, STYP f5, NA f6, NV f7, (CT f8), (PX f9), (PZ f10)}

f1 = Seismic zone coefficient. Instead of using an integer value like 1, 2, 3 or 4, use the fractional value like 0.075, 0.15, 0.2, 0.3, 0.4, etc.

f2 = Importance factor

f3 = Numerical coefficient R for lateral load in X direction

f4 = Numerical coefficient R for lateral load in Z direction

f5 = Soil Profile type

f6 = Near source factor Na

f7 = Near source factor Nv

f8 = Optional CT value to calculate time period based on Method A (see Note 7)

f9 = Optional Period of structure (in sec) in X-direction to be used in Method B

f10 = Optional Period of structure (in sec) in Z-direction to be used in Method B

The Soil Profile Type parameter **STYP** can take on values from one to f. These are related to the values shown in Table 16-J of the UBC 1997 code in the following manner :

Table 5-8: Soil profile types (**STYP**) for UBC 1997

STAAD Value	UBC 199-7 code value
1	S _A
2	S _B
3	S _C

STAAD Value	UBC 199-7 code value
4	S_D
5	S_E

Note: The soil profile type S_F is not supported.

The seismic zone factor (ZONE) in conjunction with the soil profile type (STYP), Near source factor (Na), and the Near source factor (Nv), is used to determine the values of seismic coefficients C_a and C_v from Tables 16-Q and 16-R of the UBC 1997 code.

If the **ACCIDENTAL** option is specified, the accidental torsion will be calculated per the UBC specifications. The value of the accidental torsion is based on the "center of mass" for each level. The "center of mass" is calculated from the SELFWEIGHT, JOINT WEIGHTs and MEMBER WEIGHTs you have specified.

Example 1

```

DEFINE UBC LOAD
ZONE 0.38 I 1.0 STYP 2 RWX 5.6 RWZ 5.6 NA 1.3 NV 1.6 CT
0.037
SELFWEIGHT
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
MEMBER WEIGHT
12 17 24 UNI 25.7
FLOOR WEIGHT
YRA 9 11 FOOR 200 XRA -1 21 ZR -1 41
ELEMENT WEIGHT
234 TO 432 PR 150

```

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Example 2

The following example shows the commands required to enable the program to generate the lateral loads. See "Generation of Loads" on page 651 for this information.

```
LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
UBC LOAD X 0.75
LOAD 2 ( SEISMIC LOAD IN Z DIRECTION )
UBC LOAD Z 0.75
```

The UBC / IBC input can be provided in two or more lines using the continuation mark (hyphen) as shown in the following example :

```
DEFINE UBC ACCIDENTAL LOAD
ZONE 3.000 -
I 1.00 RWX 1.100 -
RWZ 1.200 STYP 5.000 NA 1.40 NV 1.50 CT -
1.300 PX 2.100 PZ 2.200
```

5.31.2.2 UBC 1994 or 1985 Load Definition

This set of commands may be used to define the parameters for generation of UBC-type equivalent static lateral loads for seismic analysis. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

Philosophy

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the **SET Z UP** command in Section 5.5). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Total lateral seismic force or base shear is automatically calculated by STAAD using the appropriate UBC equations (All symbols and notations are per UBC).

UBC 1994: Equation 1

$$V = ZIC W$$

Rw

UBC 1984: Equation 2

$$V = ZIK \cdot C_S \cdot W$$

Base shear V may be calculated by STAAD using either the 1994 procedure (equation 1) or the 1985 procedure (equation 2). The user should use the appropriate "ubc-spec" (see General Format below) to instruct the program accordingly.

Procedure Used by the Program

STAAD utilizes the following procedure to generate the lateral seismic loads.

1. You must specify seismic zone co-efficient and desired *ubc-spec* (1985 or 1994) following the **DEFINE UBC LOAD** command.
2. Program calculates the structure period T .
3. Program calculates C from appropriate UBC equation(s) utilizing T .
4. Program calculates V from appropriate equation(s). W is obtained from **SELFWEIGHT**, **JOINT WEIGHT**, **MEMBER WEIGHT**, **ELEMENT WEIGHT**, **FLOOR WEIGHT**, and **ONEWAY WEIGHT** commands specified following the **DEFINE UBC LOAD** command. The weight data must be in the order shown.

Note: If both mass table data (**SELFWEIGHT**, **JOINT WEIGHT**, **MEMBER WEIGHT**, etc. options) and a **REFERENCE LOAD** are specified, these will be added algebraically for a combined mass.

5. The total lateral seismic load (base shear) is then distributed by the program among different levels of the structure per UBC procedures.

General Format

```

DEFINE UBC (ACCIDENTAL) LOAD
ZONE  $f_1$  ubc-spec
SELFWEIGHT ( $f_{11}$ )
JOINT WEIGHT
joint-list WEIGHT  $w$ 
MEMBER WEIGHT
mem-list { UNI  $v_1$   $v_2$   $v_3$  | CON  $v_4$   $v_5$  }
ELEMENT WEIGHT

```

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

```

plate-list PRESS  $p_1$ 
FLOOR WEIGHT
floor-weight-spec
ONEWAY WEIGHT
oneway-weight-spec
REFERENCE LOAD { X | Y | Z }
 $Ri_1 f_{11}$ 

```

Where:

ubc-spec for UBC 1994 = *{ I f_2 RWX f_3 RWZ f_4 S f_5 (CT f_8) (PX f_9) (PZ f_{10}) }

ubc-spec for UBC 1985 = *{ K f_6 I f_2 (TS f_7) }

floor-weight-spec See "Floor Load Specification" on page 562 for floor weight specification.

oneway-weight-spec See "One-way Load Specification" on page 556 for One-way load specification.

Note: The weight definitions must be in the order specified above. That is, selfweight, joint weight, member weight, element weight, and then floor weights. If one or more is not present, it can be skipped as long as the general order is preserved.

Where:

Parameter	Default Value	Description
f_1	-	The seismic zone coefficient (0.2, 0.3 etc.). Instead of using an integer value like 1, 2, 3 or 4, use the fractional value like 0.075, 0.15, 0.2, 0.3, 0.4, etc.
f_2	-	The importance factor. Used for both 1985 and 1994 specifications.
f_3	-	(UBC 1994 spec only) Numerical coefficient R_w for lateral load in X-direction.

Parameter	Default Value	Description
f_4	-	(UBC 1994 spec only) Numerical coefficient R_w for lateral load in Z-directions
f_5	-	(UBC 1994 spec only) Site co-efficient for soil characteristics
f_6	-	(UBC 1985 spec only) Horizontal force factor
f_7	0.5	(UBC 1985 spec only) Site characteristic period (Referred to as T_s in the UBC code).
f_8	0.035	(UBC 1994 spec only) Value of the term C_t which appears in the equation of the period of the structure per Method A. See note 7 in section 5.31.2.1.
f_9	-	(UBC 1994 spec only) Period of structure (in seconds) in the X- direction.
f_{10}	-	(UBC 1994 spec only) Period of structure (in seconds) in the Z direction (or Y if SET Z UP is used).
w	-	The joint weight associated with list
v_1, v_2, v_3	-	Used when specifying a uniformly distributed load with a value of v_1 starting at a distance of v_2 from the start of the member and ending at a distance of v_3 from the start of the member. If v_2 and v_3 are omitted, the load is assumed to cover the entire length of the member.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Parameter	Default Value	Description
v_4, v_5	-	Used when specifying a concentrated force with a value of v_4 applied at a distance of v_5 from the start of the member. If v_5 is omitted, the load is assumed to act at the center of the member.
p_1	-	The weight per unit area for the plates selected. Assumed to be uniform over the entire plate. Element Weight is used if plate elements are part of the model, and uniform pressures on the plates are to be considered in weight calculation.
Ri_1	-	Identification number of a previously defined reference load case. See "Defining Reference Load Types" on page 531
f_{11}	1.0	Magnification factor (required for reference loads).

Floor Weight is used if the pressure is on a region bounded by beams, but the entity which constitutes the region, such as a slab, is not defined as part of the structural model. It is used in the same sort of situation in which you would use **FLOOR LOADS** (See "Floor Load Specification" on page 562 for details). Similarly, you can use the Oneway Weight command to specify a load path direction for the pressure on a region.

Notes

- If the option **ACCIDENTAL** is used, the accidental torsion will be calculated per UBC specifications. The value of the accidental torsion is based on the center of mass for each level. The center of mass is calculated from the **SELFWEIGHT, JOINT WEIGHT, MEMBER WEIGHT, ELEMENT WEIGHT, FLOOR WEIGHT, and ONEWAY WEIGHT** commands you have specified.

- b. In *ubc-spec* for 1985 code, specification of TS is optional. If TS is specified, resonance co-efficient S is determined from the building period T and user provided TS using UBC equations. If TS is not specified, the default value of 0.5 is assumed.
- c. By providing either PX or PZ or both, you may override the period calculated by STAAD for Method B of the UBC Code. The user defined value will then be used instead of the one recommended by UBC per equation 28.5 of UBC 94. If you do not define PX or PZ, the period for Method B will be calculated by the program per equation 28.5.
- d. Some of the items in the output for the UBC analysis are explained below.

CALC / USED PERIOD

The CALC PERIOD is the period calculated using the Rayleigh method (Method B as per UBC code). For UBC in the x-direction, the USED PERIOD is PX. For the UBC in the z-direction, the USED PERIOD is PZ. If PX and PZ are not provided, then the used period is the same as the calculated period for that direction. The used period is the one substituted into the critical equation of the UBC code to calculate the value of C.

- e. In the analysis for UBC loads, all the supports of the structure have to be at the same level and have to be at the lowest elevation level of the structure.

Example

```

DEFINE UBC LOAD
ZONE 0.2 I 1.0 RWX 9 RWZ 9 S 1.5 CT 0.032
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 2.5
49 TO 64 WEIGHT 1.25
LOAD 1
UBC LOAD X 0.75
SELFWEIGHT Y -1.0
JOINT LOADS
17 TO 48 FY -2.5
FLOOR WEIGHT

```

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

```
_SLAB1 FLOAD 0.045  
ONEWAY LOAD  
_ROOF ONE 0.035 GY
```

5.31.2.3 Colombian Seismic Load

The purpose of this command is to define and generate static equivalent seismic loads as per Colombian specifications using a static equivalent approach similar to those outlined by UBC. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the **SET Z UP** command in Section 5.5). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Methodology

Seismic zone coefficient and parameter values are supplied by the user through the DEFINE COLOMBIAN LOAD command.

Program calculates the natural period of building T utilizing clause 1628.2.2 of UBC 1994.

Design spectral coefficient, S_a , is calculated utilizing T as:

$$S_a = \begin{cases} A_a I (1.0 + 5.0T) & \text{when } 0 \leq T \leq 0.3 \text{ sec} \\ 2.5A_a I & \text{when } 0.3 < T \leq 0.48 \text{ sec} \\ 1.2A_a S \frac{I}{T} & \text{when } 0.48 < T < 2.4 \text{ sec} \\ A_a I / 2 & \text{when } 2.4 \text{ sec} < T \end{cases}$$

Where:

A_a = Seismic Risk factor

S = Soil Site Coefficient

I = Coefficient of Importance

Base Shear, V_s is calculated as

$$V_s = W * S_a$$

W = Total weight on the structure

The total lateral seismic load, V_s , is then distributed by the program among different levels as:

$$F_x = C_{vx} * V_s$$

Where:

$$C_{vx} = \frac{W_x h_x K}{\sum_{i=1}^n (W_x h_x K)}$$

W_x = Weight at the particular level

h_x = Height of that particular level

1.0 when $T \leq 0.5$ sec

$K = 0.75 + 0.5T$ when $0.5 < T \leq 2.5$ sec

2.0 when 2.5 sec $< T$

General Format

DEFINE COLUMBIAN (ACCIDENTAL) LOAD

ZONE f_1 *col-spec*

SELFWEIGHT

JOINT WEIGHT

***joint-list* WEIGHT w**

...

Note: See "UBC 1994 or 1985 Load Definition" on page 438 for complete weight input definition.

Where:

$$\text{col-spec} = (\underline{I} f_2 \underline{S} f_3)$$

f_1 = Seismic Risk factor

f_2 = Soil Site Coefficient

f_3 = Coefficient of Importance

General format to provide Colombian Seismic load in any load case:

LOAD i

COLMBIAN LOAD {X | Y | Z} (f4) (ACC f5)

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Where:

i = load case number

f_4 = factor to multiply horizontal seismic load

f_5 = multiplying factor for Accidental Torsion, to be used to multiply the accidental torsion load (default = 1.0). May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

If the **ACCIDENTAL** option is specified, the accidental torsion will be calculated per the specifications. The value of the accidental torsion is based on the "center of mass" for each level. The "center of mass" is calculated from the SELFWEIGHT, JOINT WEIGHTs and MEMBER WEIGHTs you have specified.

Example

```
DEFINE COLOMBIAN LOAD
ZONE 0.38 1.0 S 1.5
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
COLOMBIAN LOAD X
```

5.31.2.4 Japanese Seismic Load

The purpose of this command is to define and generate static equivalent seismic loads as per Japanese specifications using a static equivalent approach similar to those outlined by UBC. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s). The implementation is as per Article 88 in the 'Building Codes Enforcement Ordinance 2006'.

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the **SET Z UP** command in Section 5.5). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Methodology

Seismic zone coefficient and parameter values are supplied by the user through the **DEFINE AIJ LOAD** command.

Program calculates the natural period of building *T* utilizing the following equation:

$$T = h(0.02 + 0.01\alpha)$$

Where:

h = height of building (m)

α = Alpha, ratio of steel height for overall height

Design spectral coefficient, R_t , is calculated utilizing T and T_c as follows:

$$R_t = \begin{cases} 1.0 & \text{when } T < T_c \\ 1 - 0.2(T/T_c - 1)^2 & \text{when } T_c \leq T \leq 2T_c \\ 1.6T_c/T & \text{when } 2T_c < T \end{cases}$$

T_c = Period defined by ground support specification for R_t .

α_i is calculated from the weight values provided in the Define AIJ Load command.

$$\alpha_i = W_i / W$$

Where:

W_i = sum of weight from a top to floor i .

W = all weight

The seismic coefficient of floor, C_i , is calculated using:

$$C_i = Z R_t A_i C_o$$

Where:

Z = zone factor

C_o = normal coefficient of shear force

$$A_i = 1 + \frac{2T(1/\sqrt{a_i} - a_i)}{1 + 3T}$$

Seismic load shear force, Q_i , of each floor is calculated by C_i and W_i :

$$Q_i = C_i W_i$$

Where:

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

W_i = sum of weight from a top to i floor

Load value of each floor P_i is calculated by seismic load shear force Q_i .

$$P_i = Q_i - Q_{i+1}$$

The total lateral seismic load is distributed by the program among different levels.

General Format

DEFINE AIJ (ACCIDENTAL) LOAD

ZONE f_1 *AIJ-spec*

SELFWEIGHT

JOINT WEIGHT

***joint-list* WEIGHT w**

...

Note: See "UBC 1994 or 1985 Load Definition" on page 438 for complete weight input definition.

Where:

***AIJ-spec* = { CO f_2 TC f_3 ALPHA f_4 }**

f_1 = Zone factor (0.7, 0.8, 0.9 or 1.0)

f_2 = Normal coefficient of shear force (0.2 or 1.0)

f_3 = Period defined by ground support specification (0.4, 0.6 or 0.8 sec)

f_4 = Ratio of steel height to overall building height, which are used in the calculation of R_t .

General format to provide Japanese Seismic load in a primary load case:

LOAD i

AIJ LOAD {X | Y | Z} (f_5) (ACC f_6)

Where:

i = load case number

f_5 = optional factor to multiply horizontal seismic load.

f_6 = multiplying factor for Accidental Torsion, to be used to multiply the AIJ accidental torsion load (default = 1.0). May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

Choose horizontal directions only.

If the **ACCIDENTAL** option is specified, the accidental torsion will be calculated per the AIJ specifications. The value of the accidental torsion is based on the "center of mass" for each level. The "center of mass" is calculated from the SELFWEIGHT, JOINT WEIGHTs and MEMBER WEIGHTs you have specified

Example

```

DEFINE AIJ LOAD
ZONE 0.8 CO 0.2 TC 0.6 ALPHA 1.0
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
LOAD 1 ( SEISMIC LOAD IN X)
AIJ LOAD X

```

5.31.2.5 Indian IS:1893 (Part 1) 2002 Code - Lateral Seismic Load

This feature enables one to generate seismic loads per the IS:1893 specifications using a static equivalent approach.

The seismic load generator can be used to generate lateral loads in the X and Z directions only. Y is the direction of gravity loads. This facility has not been developed for cases where the Z axis is set to be the vertical direction (See the **SET Z UP** command in Section 5.5).

Methodology

The design base shear is computed by STAAD in accordance with the IS: 1893 (Part 1)-2002 equation 7.5.3.

$$V = A_h W$$

Where:

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

$$A_h = Z/2 \cdot I/R \cdot S_a/g$$

Note: All symbols and notations in the above equation are as per IS: 1893(Part 1)-2002.

STAAD utilizes the following procedure to generate the lateral seismic loads:

1. You provide seismic zone coefficient and desired "1893(Part 1)-2002 specs" through the **DEFINE 1893 LOAD** command.
2. Program calculates the structure period (T).
3. Program calculates S_a/g utilizing T.
4. Program calculates V from the above equation. W is obtained from mass table data entered via **SELFWEIGHT**, **JOINT WEIGHT(s)**, **MEMBER WEIGHT(S)**, and/or **REFERENCE LOAD** you provide through the **DEFINE 1893 LOAD** command.
5. The total lateral seismic load (base shear) is then distributed by the program among different levels of the structure per the IS: 1893 procedures.

See "Generation of Loads" on page 651 for additional information.

General Format

```
DEFINE 1893 (ACCIDENTAL) LOAD  
ZONE f1 1893-spec  
SELFWEIGHT  
JOINT WEIGHT  
joint-list WEIGHT w  
MEMBER WEIGHT  
mem-list { UNI v1 v2 v3 | CON v4 v5 }  
REFERENCE LOAD { X | Y | Z }  
Ri1 f11  
(CHECK SOFT STORY)
```

Where:

```
1893-spec = RF f2 I f3 SS f4 (ST f5) DM f6 (PX f7) (PZ f8)  
( { DT f9 | GL f10 } )
```

Note: If both mass table data (**SELFWEIGHT**, **JOINT WEIGHT**, and **MEMBER WEIGHT** options) and a **REFERENCE LOAD** are specified, these will be added algebraically for a combined mass.

Where:

f_1 = Seismic zone coefficient. Refer to Table 2 of IS:1893 (Part 1)-2002.

f_2 = Response reduction factor. Refer Table 7 of IS: 1893 (Part 1) -2002.

f_3 = Importance factor depending upon the functional use of the structures, characterized by hazardous consequences of its failure, post-earthquake functional needs, historical value, or economic importance. Refer Table 6 of IS: 1893(Part 1)-2002.

f_4 = Rock or soil sites factor (=1 for hard soil, 2 for medium soil, 3 for soft soil). Depending on type of soil, average response acceleration coefficient S_a/g is calculated corresponding to 5% damping. Refer Clause 6.4.5 of IS: 1893 (Part 1) -2002.

f_5 = Optional value for type of structure (= 1 for RC frame building, 2 for Steel frame building, 3 for all other buildings). If this parameter is mentioned the program will calculate natural period as per Clause 7.6 of IS:1893(Part 1)-2002.

f_6 = Damping ratio to obtain multiplying factor for calculating S_a/g for different damping. If no damping is specified 5% damping (default value 0.05) will be considered corresponding to which multiplying factor is 1.0. Refer Table 3 of IS:1893(Part 1)-2002.

f_7 = Optional period of structure (in sec) in X direction. If this is defined this value will be used to calculate S_a/g for generation of seismic load along X direction.

f_8 = Optional period of structure (in sec) in Z direction. If this is defined this value will be used to calculate S_a/g for generation of seismic load along Z direction.

f_9 = Depth of foundation below ground level. It should be defined in current unit. If the depth of foundation is 30 m or more, the value of A_h is taken as half the value obtained. If the foundation is placed between the ground level and 30 m depth, this value is linearly interpolated between A_h and $0.5A_h$.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

f_{10} = Y coordinate of ground level (or global Z coordinate for **SET Z UP**). A reduced lateral force is applied to levels below this height, per Clause 6.4.4.

f_{11} = Magnification factor (required for reference loads). Default is 1.0.

R_i = Identification number of a previously defined reference load case. See "Defining Reference Load Types" on page 531

w = joint weight associated with joint list

v_1, v_2, v_3 = Used when specifying a uniformly distributed load with a value of v_1 starting at a distance of v_2 from the start of the member and ending at a distance of v_3 from the start of the member. If v_2 and v_3 are omitted, the load is assumed to cover the entire length of the member.

v_4, v_5 = Used when specifying a concentrated force with a value of v_4 applied at a distance of v_5 from the start of the member. If v_5 is omitted, the load is assumed to act at the center of the member.

Notes

- If the **ACCIDENTAL** option is specified, the accidental torsion will be calculated per the IS 1893 specifications. The value of the accidental torsion is based on the center of mass for each level. The center of mass is calculated from the **SELFWEIGHT**, **JOINT WEIGHT**, and **MEMBER WEIGHT** commands you have specified.
- By default STAAD calculates natural periods of the structure in both X and Z directions respectively which are used in calculation for base shear. If, however, PX and PZ are mentioned the program will consider these values for calculation of average response acceleration coefficient. If instead of PX and PZ values ST is mentioned the program will calculate natural period depending upon the empirical expression given in IS: 1893 (Part 1)-2002.
- Either the **DT** or **GL** parameter is to be specified in the input. If both parameters are specified, the **DT** value will be ignored

5.31.2.5.1 Soft Story Checking

STAAD will perform this check when the option **CHECK SOFT STORY** command is used. If omitted, no soft story check is performed. Soft Story checking is only valid for structures having vertical elements in the form of columns and shear

wall (without opening). At present the program will exclude the effect of any other forms of lateral force resisting structural elements.

As per the IS1893-2002 code Clause 7.1, to perform well during an earthquake a building must have simple and regular configuration, adequate lateral strength, stiffness and ductility. This is because a building with simple regular geometry and uniformly distributed mass and stiffness in plan as well as in elevation, will suffer much less damage than buildings with irregular configurations.

According to this standard, a building can be considered irregular, if at least one of the conditions given in Table 4 - Plan Irregularities and Table 5 - Vertical Irregularities, of IS1893-2002 is applicable.

STAAD has implemented the methodology to find vertical stiffness irregularities, as given in Table 5 Sl No. (1) i) a) and Sl No. (1) i) b), in the form of soft story checking.

Stiffness Irregularities: Soft Story

As per this provision of the code, a soft story is one in which the lateral stiffness is less than 70 percent of that in the story above or less than 80 percent of the average lateral stiffness of the three story above.

Stiffness Irregularities: Extreme Soft Story

As per this provision of the code, a extreme soft story is one in which the lateral stiffness is less than 60 percent of that in the story above or less than 70 percent of the average lateral stiffness of the three story above.

Thus, if any story of a building is found to be soft or extremely soft, the building is likely to suffer much damage in an earthquake than a similar type of building but has more regular vertical stiffness.

Note: STAAD identifies column and shear wall (without opening) as vertical component for the purpose of computing lateral stiffness of the story. The vertical stiffness of a column is calculated as $12EI / L^3$ where E is the Young's modulus, I is the moment of inertia and L is the length of the column respectively and that for a shear wall (without opening) is calculated as $Ph^3/3EI + 1.2Ph/AG$ (i.e., summation of flexural stiffness and shear stiffness, obtained as deflection of a cantilever wall under a single lateral load P at its top) where h is the height, A is the cross-sectional area and G is the shear modulus of the wall (E and I carry usual meaning). The summation of lateral stiffness of all columns and

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

shear walls at a particular floor level constitute the total lateral stiffness of that particular story or floor level. The program checks soft story of a building along both global X and Z directions respectively. This computation is valid *only* for those structures whose floors are treated as rigid diaphragm.

5.31.2.5.2 Identification of Floor Level

The following two ways can identify floor level:

- Program calculated
- User defined

For regular building which has well defined floors (i.e., does not contain shear wall, staggered flooring, etc), STAAD can identify floor level on its own. However, if floor level is not so well defined, it is better to define floor height to have more accurate result for the purpose of torsion and soft story checking.

Program calculated

In general, STAAD identifies floor levels in order of increasing magnitude of Y-coordinates of joints. The program sorts different values of Y-coordinates, from minimum to maximum values, in ascending order and consider each Y-coordinate value as each floor level. This is the method used by the **DEFINE UBC** or similar load generation features.

This feature has been enhanced to identify the beam-column junctions at each floor level, as identified by the method above. If no beam-column junctions are identified at that level, a floor level will not be considered at that level in the structure. Where beam-column junctions are found, the program identifies two beams, at the same level which span in two different directions from the same beam-column junction. If this true, this identified floor level will be considered as truly existing floor level in the structure.

The enhanced feature for finding floor level is being used by lateral load generation feature for response spectrum analysis and soft story checking as per IS1893-2002 code. For response spectrum analysis, story drift and soft story checking is performed only when floor heights are specified.

User defined

Floor heights should be defined before using the **DEFINE** command for any primary response spectrum load case. The following general format is used for a user-defined floor height:

FLOOR HEIGHT

$$h_1; h_2; h_3; \dots; h_i -$$

$$h_{i+1}; \dots; h_n$$

$$(\text{BASE } h_b)$$

Where:

$h_1 \dots h_n$ = the different floor heights in current length unit and n is the number of floor levels.

h_b = the base level with respect to which 1st story height will be calculated. If h_b is not defined, the minimum Y-coordinate value present in the model will be taken as base level.

User defined floor heights are used by lateral load generation for response spectrum analysis and soft story checking as per IS1893-2002 code.

See "Generation of Seismic Loads" on page 653 for an example of the correct usage of this command.

5.31.2.6 IBC 2000/2003 Load Definition

The specifications of the IBC 2000, and 2003 codes for seismic analysis of a building using a static equivalent approach have been implemented as described in this section. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the **SET Z UP** command in Section 5.5). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

The implementation details of the respective codes are as follows:

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

IBC 2000

On a broad basis, the rules described in section 1617.4 of the IBC 2000 code document have been implemented. These are described in pages 359 thru 362 of that document. The specific section numbers, those which are implemented, and those which are not implemented, are as follows:

Table 5-9: Sections of IBC 2000 implemented and omitted in the program

Implemented sections of IBC 2000	Omitted sections of IBC 2000
1617.4.1	1617.4.4.1
1617.4.1.1	1617.4.4.2
1617.4.2	1617.4.4.3
1617.4.2.1	1617.4.4.5
1617.4.3	1617.4.5
1617.4.4	1617.4.6
1617.4.4.4	

IBC 2003

On a broad basis, the rules described in section 1617.4 of the IBC 2003 code document have been implemented. This section directs the engineer to Section 9.5.5 of the ASCE 7 code. The specific section numbers of ASCE 7-2002, those which are implemented, and those which are not implemented, are shown in the table below. The associated pages of the ASCE 7-2002 code are 146 thru 149.

Table 5-10: Sections of IBC 2003 (ASCE 7-02) implemented and omitted in the program

Implemented sections of IBC 2003 (ASCE 7-02)	Omitted sections of IBC 2003 (ASCE 7-02)
9.5.5.2	9.5.5.5.1
9.5.5.2.1	9.5.5.5.2
9.5.5.3	9.5.5.6
9.5.5.3.1	9.5.5.7
9.5.5.3.2	
9.5.5.4	
9.5.5.5	
Portions of 9.5.5.5.2	

Methodology

The design base shear is computed in accordance with Eqn. 16-34 of IBC 2000 and Eqn. 9.5.5.2-1 of ASCE 7-02:

$$V = C_s W$$

The seismic response coefficient, C_s , is determined in accordance with Eqn. 16-35 of IBC 2000 / Eqn. 9.5.5.2.1-1 of ASCE 7-02:

$$C_s = \frac{S_{DS}}{R / I_E}$$

C_s need not exceed the limit given in Eqn. 16-36 of IBC 2000 / Eqn. 9.5.5.2.1-2 of ASCE 7-02:

$$C_s = \frac{S_{D1}}{(R / I_E)T}$$

C_s shall not be taken less than the lower limit given in Eqn. 16-37 of IBC 2000 / Eqn. 9.5.5.2.1-3 of ASCE 7-0:

$$C_s = 0.044 S_{DS} I_E$$

In addition, for structures for which the 1-second spectral response, S_1 , is equal to or greater than 0.6g, the value of the seismic response coefficient, C_s , shall not be

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

taken less than the limit given in Eqn. 16-38 of IBC 2000 / Eqn. 9.5.5.2.1-4 of ASCE 7-02:

$$C_s = \frac{0.5S_1}{R/I_E}$$

For an explanation of the terms used in the above equations, please refer to the relevant IBC and ASCE 7-02 codes.

Procedure Used by the Program

Steps used to calculate and distribute the base shear are as follows:

1. The Time Period of the structure is calculated based on section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02 (IBC 2003) This is reported in the output as T_a .
2. The period is also calculated in accordance with the Rayleigh method. This is reported in the output as T .
3. you may override the Rayleigh based period by specifying a value for PX or PZ (Items f7 and f8) depending on the direction of the IBC load.
4. The governing Time Period of the structure is then chosen between the above two periods, and the additional guidance provided in clause 1617.4.2 of IBC 2000, section 9.5.5.3 of ASCE 7-02 (IBC 2003) or section 12.8.2.1 of ASCE 7-05 (IBC 2006). The resulting value is reported as "Time Period used".
5. The Design Base Shear is calculated based on equation 16-34 of IBC 2000, equation 9.5.5.2-1 of ASCE 7-02 (IBC 2003) or equation 12.8-1 of ASCE 7-05 (IBC 2006). It is then distributed at each floor using the rules of clause 1617.4.3, equations 16-41 and 16-42 of IBC 2000. For IBC 2003, using clause 9.5.5.4, equations 9.5.5.4-1 & 9.5.5.4-2 of ASCE 7-02.
6. If the ACCIDENTAL option is specified, the program calculates the additional torsional moment. The lever arm for calculating the torsional moment is obtained as 5% of the building dimension at each floor level perpendicular to the direction of the IBC load (clause 1617.4.4.4 of IBC 2000, and section 9.5.5.5.2 of ASCE 7-02 for IBC 2003). At each joint where a weight is located, the lateral seismic force acting at that joint is multiplied by this lever arm to obtain the torsional moment at that joint.

General Format

There are 2 stages of command specification for generating lateral loads. This is the first stage and is activated through the DEFINE IBC 2000 or 2003 LOAD command.

```

DEFINE IBC ( { 2000 | 2003 } ) (ACCIDENTAL) LOAD
SDS f1 ibc-spec
SELFWEIGHT
JOINT WEIGHT
joint-list WEIGHT w
...

```

Note: See "UBC 1994 or 1985 Load Definition" on page 438 for complete weight input definition.

Where:

```

ibc-spec = { SD1 f2 S1 f3 IE f4 RX f5 RZ f6 SCLASS f7 (CT
f8) (PX f9) (PZ f10) }

```

f1 = Design spectral response acceleration at short periods. See equation 16-18, Section 1615.1.3 of IBC 2000 and equation 9.4.1.2.5-1 of ASCE7-02

f2 = Design spectral response acceleration at 1-second period. See equation 16-19, Section 1615.1.3. of IBC 2000 and equation 9.4.1.2.5-2 of ASCE7-02

f3 = Mapped spectral acceleration for a 1-second period. See equation 16-17 of IBC 2000, and 9.4.1.2.4-2 of ASCE 7-02

f4 = Occupancy importance factor determined in accordance with Section 1616.2 of IBC 2000 and 2003, and section 9.1.4 (page 96) of ASCE 7-02

f5 = The response modification factor for lateral load along the X direction. See Table 1617.6 of IBC 2000 (pages 365-368) and Table 1617.6.2 of IBC 2003 (page 334-337). It is used in equations 16-35, 16-36 & 16-38 of IBC 2000

f6 = The response modification factor for lateral load along the Z direction. See Table 1617.6 of IBC 2000 (pages 365-368) and Table

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

1617.6.2 of IBC 2003 (page 334-337). It is used in equations 16-35, 16-36 & 16-38 of IBC 2000.

f7 = Site class as defined in Section 1615.1.1 of IBC 2000 (page 350) & 2003 (page 322). Enter 1 through 6 in place of A through F, see table below).

f8 = Optional CT value to calculate time period. See section 1617.4.2.1, equation 16-39 of IBC 2000 and section 9.5.5.3.2, equation 9.5.5.3.2-1 of ASCE 7-02.

If the value of Ct is not specified, the program scans the Modulus of Elasticity (E) values of all members and plates to determine if the structure is made of steel, concrete or any other material. If the average E is smaller than 2000 ksi, Ct is set to 0.016(0.044) [0.016 is the value in FPS units, 0.044 is the value in Metric units]. If the average E is between 2000 & 10000 ksi, Ct is set to 0.02(0.055). If the average E is greater than 10000 ksi, Ct is set to 0.028(0.068). If the building material cannot be determined, Ct is set to 0.02(0.055).

In Table 9.5.5.3.2 on page 147 of ASCE 7-02, a structure type called "Eccentrically braced steel frames" is listed, with a Ct of 0.03(0.07). STAAD does not choose this Ct on its own presently. If one wishes to use this value, Ct should be specified as an input accordingly.

f9 = Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure instead of the value derived from section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02.

f10 = Optional Period of structure (in sec) in Z or Y direction to be used as fundamental period of the structure instead of the value derived from section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02.

The Soil Profile Type parameter SCLASS can take on values from 1 to 6. These are related to the values shown in Table 1615.1.1, Site Class Definitions of the IBC 2000/2003 code in the following manner :

Table 5-11: Values of IBC soil class (SCLASS) used in STAAD

STAAD Value	IBC Value
1	A
2	B
3	C
4	D
5	E
6	F

Example 1

```

DEFINE IBC 2003 LOAD
SDS 0.6 SD1 0.36 S1 0.31 I 1.0 RX 3 RZ 4 SCL 4 CT 0.032
SELFWEIGHT
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720

```

Example 2

The following example shows the commands required to enable the program to generate the lateral loads. Users may refer to Section 5.32.12 of the Technical Reference Manual for this information.

```

LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
IBC LOAD X 0.75

```

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

LOAD 2 (SEISMIC LOAD IN Z DIRECTION)
IBC LOAD Z 0.75

The Examples manual contains examples illustrating load generation involving IBC and UBC load types.

5.31.2.7 CFE (Comisión Federal De Electricidad) Seismic Load

The purpose of this command is to define and generate static equivalent seismic loads as per MANUAL DE DISEÑO POR SISMO - SEISMIC DESIGN HANDBOOK COMISIÓN FEDERAL DE ELECTRICIDAD - ELECTRIC POWER FEDERAL COMMISSION - October 1993 (Chapters 3.1, 3.2, 3.3 and 3.4) specifications. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s). This is a code used in the country of Mexico.

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the **SET Z UP** command in Section 5.5). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Methodology

Seismic zone coefficient and parameter values are supplied by the user through the **DEFINE CFE LOAD** command.

Program calculates the natural period of building T utilizing Rayleigh-Quotient method. If time period is provided in the input file, that is used in stead of calculated period.

The acceleration a is calculated according to the following:

$$a = \begin{cases} a_0 + \left(c - a_0 \right) \frac{T}{T_a} & \text{when } T < T_a \\ c & \text{when } T_a \leq T \leq T_b \\ c(T_b / T)^r & \text{when } T_b < T \end{cases}$$

Where:

c = Seismic coefficient is extracted from table 3.1

a₀, T_a, T_b, and r are obtained form table 3.1

The ductility reduction factor Q' is calculated according to section 3.2.5.

$$Q' = \begin{cases} Q & \text{when } T \geq T_a \\ 1 + \left(\frac{T}{T_a}\right)(Q - 1) & \text{when } T < T_a \end{cases}$$

If not regular, then $Q' = Q' \times 0.8$

If the period T_s of the soil is known and the soil type II or III T_a and T_b will be modified according to section 3.3.2.

Lateral loads for each direction are calculated for:

When $T \leq T_b$, Eq. 4.5. Section 3.4.4.2 is used:

$$P_n = W_n h_n \frac{\sum_{n=1}^N (W_n) a}{\sum_{n=1}^N (W_n h_n) Q'}$$

When $T > T_b$, Eq. 4.6/7/8. Section 3.4.4.2 is used:

$$P_n = W_n \left(a / Q \right) \left(K_1 h_i + K_2 h_i^2 \right)$$

Where:

$$K_1 = \frac{q[1 - r(1 - q)] \sum W_i}{\sum (W_i / h_i)}$$

$$K_2 = \frac{1.5rq(1 - q) \sum W_i}{\sum (W_i / h_i^2)}$$

$$q = (T_b / T)^r$$

The base shears are distributed proportionally to the height if $T \leq T_b$ or with the quadratic equation mentioned if $T > T_b$.

The distributed base shears are subsequently applied as lateral loads on the structure.

General Format

DEFINE CFE (ACCIDENTAL) LOAD

ZONE f1 *cfe-spec*

SELFWEIGHT

JOINT WEIGHT

***joint-list* WEIGHT w**

...

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Note: See "UBC 1994 or 1985 Load Definition" on page 438 for complete weight input definition.

Where:

cfe-spec = { QX f2 QZ f3 GROUP f4 STYP f5 (REGULAR) (TS f6) (PX f7) (PZ f8) }

f1 = Zone number specified in number such as 1, 2, 3 or 4

f2 = seismic behavior factor of the structure along X direction as a parameter according 3.2.4.

f3 = seismic behavior factor of the structure along Z direction as a parameter according 3.2.4.

f4 = Group of structure entered as A or B

f5 = Soil type entered as 1 or 2 or 3

The optional parameter **REGULAR** is entered to consider the structure as a regular structure. By default, all structures are considered as irregular.

f6 = site characteristic period

f7 = Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure instead of the value calculated by the program using Rayleigh-Quotient method

f8 = Optional Period of structure (in sec) in Z direction (or Y if SET Z UP is used) to be used as fundamental period of the structure instead of the value calculated by the program using Rayleigh-Quotient method

To provide a CFE Seismic load in any load case:

LOAD i

CFE LOAD {X | Y | Z} (f)

Where:

i = the load case number

f = factor to multiply horizontal seismic load. Choose horizontal directions only.

Example

```

UNIT KGS METER
DEFINE CFE LOAD
ZONE 2 QX .5 QZ 0.9 STYP 2 GROUP B TS 0.2
SELFWEIGHT
MEMBER WEIGHT
1 TO 36 41 TO 50 UNI 300
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
FLOOR WEIGHT
YRA 11.8 12.2 FLOAD 400 -
XRA -1 11 ZRA -1 21
LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
CFE LOAD X 1.0
LOAD 2 ( SEISMIC LOAD IN -Z DIRECTION )
CFE LOAD Z -1.0

```

5.31.2.8 NTC (Normas Técnicas Complementarias) Seismic Load

The purpose of this command is to define and generate static equivalent seismic loads as per Code of the México Federal District (Reglamento de Construcciones del Distrito Federal de México) and Complementary Technical Standards for Seismic Design (y Normas Técnicas Complementarias (NTC) para Diseño por Sismo -Nov. 1987) (Chapters 8.1 8.2 8.6 and 8.8) specifications. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the **SET Z UP** command in Section 5.5). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Methodology

The design base shear is computed in accordance with Sections 8.1 or 8.2 of the NTC as decided by the user.

A. Base Shear is given as

$$V_o / W_o = c / Q$$

Where:

Seismic Coefficient, c , is obtained by the program from the following table

Table 5-12: Seismic coefficient per NTC

Seismic Coefficient, c	Group A	Group B
I	0.24	0.16
II not shaded	0.48	0.32
III (and II where shaded)	0.60	0.40

Q : is entered by the user as a parameter.

B. Base shear is given as

$$V_o / W_o = a / Q'$$

Where Reduction of Shear Forces are requested

Time Period T of the structure is:calculated by the program based on using Rayleigh quotient technique.

you may override the period that the program calculates by specifying these in the input

a and Q' are calculated according to the sections 3 and 4 of the NTC, that is to say:

$$\begin{aligned} & \left(1 + 3\frac{T}{T_a}\right)^c \text{ when } T < T_a \\ a = & c \text{ when } T_a \leq T \leq T_b \\ & q \cdot c \text{ when } T_b < T \end{aligned}$$

Where:

$$\begin{aligned} q = & (T_b/T)^f \\ & Q \text{ when } T \geq T_a \\ Q' = & 1 + \left(\frac{T}{T_a}\right)(Q - 1) \text{ when } T < T_a \end{aligned}$$

If not regular, then Q' = Q' x 0.8

T_a, T_b and r are taken from table 5-13 (Table 3.1 in the NTC).

Table 5-13: Values of T_a, T_b and r per NTC

Zone	T _a	T _b	r
I	0.2	0.6	1/2
II not shaded	0.3	1.5	2/3
III (and II where shaded)	0.6	3.9	1.0

a shall not be less than c/4

V_o for each direction is calculated:

$$V_o = \begin{aligned} & W_o a / Q' \text{ when } T \leq T_b \\ & \frac{\Sigma W_i a}{Q'(K_1 h_i + K_2 h_i^2)} \text{ when } T > T_b \end{aligned}$$

Where:

$$K_1 = \frac{q[1 - r(1 - q)]\Sigma W_i}{\Sigma(W_i / h_i)}$$

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

$$K_2 = \frac{1.5rq(1-q)\Sigma W_i}{\Sigma(W_i / h_i^2)}$$

W_i and h_i the weight and the height of the i^{th} mass over the soil or embedment level.

The base shears are distributed proportionally to the height if $T \leq T_b$ or with the quadratic equation mentioned if $T > T_b$. The distributed base shears are subsequently applied as lateral loads on the structure.

General Format

```
DEFINE NTC LOAD  
ZONE f1 ntc-spec  
SELFWEIGHT  
JOINT WEIGHT  
joint-list WEIGHT w  
...
```

Note: See "UBC 1994 or 1985 Load Definition" on page 438 for complete weight input definition.

Where:

```
ntc-spec = { QX f2 QZ f3 GROUP f4 (SHADOWED) (REGULAR)  
(REDUCE) (PX f6) (PZ f7) }
```

f1 = Zone number specified in number such as 1, 2, 3 or 4

f2 = seismic behavior factor of the structure along X direction as a parameter according 3.2.4.

f3 = seismic behavior factor of the structure along Z direction as a parameter according 3.2.4.

f4 = Group of structure entered as A or B

REGULAR optional parameter is entered to consider the structure as a regular structure. By default, all structures are considered as irregular.

SHADOWED optional parameter is used to define the shaded zone II as the site of the structure. By default regular zone II is used.

REDUCE optional parameter allows to reduce the seismic factors as described above. Otherwise the following formula is used to calculate base shear,

$$V = \frac{c}{Q'} \sum_{n=1}^N W_n$$

f6 = Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure instead of the value calculated by the program using Rayleigh-Quotient method

f7 = Optional Period of structure (in sec) in Z-direction to be used as fundamental period of the structure instead of the value calculated by the program using Rayleigh-Quotient method

To provide NTC Seismic load in any load case:

LOAD i

NTC LOAD {X/Y/Z} (f)

where i and f are the load case number and factor to multiply horizontal seismic load respectively.

Example

```

UNIT KGS METER
DEFINE NTC LOAD
ZONE 2 QX .5 QZ 0.9 GROUP B
SELFWEIGHT
ELEMENT WEIGHT
1577 TO 1619 PRESSURE 275
LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
NTC LOAD X 1.0
LOAD 2 ( SEISMIC LOAD IN Z DIRECTION )
NTC LOAD Z 1.0

```

5.31.2.9 RPA (Algerian) Seismic Load

The purpose of this command is to define and generate static equivalent seismic loads as per RPA specifications using a static equivalent approach similar to those

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

outlined by RPA. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the **SET Z UP** command in Section 5.5). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Methodology

The design base shear is computed in accordance with section 4.2.3 of the RPA 99 code. The primary equation, namely 4-1, as shown below, is checked.

$$V = (A D Q)W / R$$

Where:

W = total weight on the structure

A = zone coefficient

D = average dynamic amplification factor

R = lateral R factor

Q = structural quality factor

Seismic zone coefficient and parameter values are supplied by the user through the **DEFINE RPA LOAD** command.

Program calculates the natural period of building T utilizing clause 4.2.4 of RPA 99.

Design spectral coefficient (D) is calculated utilizing T as,

$$\begin{aligned} D &= 2.5\eta \text{ when } 0 \leq T \leq T_2 \\ &= 2.5\eta(T_2/T)^{2/3} \text{ when } T_2 \leq T \leq 0.3 \text{ sec.} \\ &= 2.5\eta(T_2/3)^{2/3}(3/T)^{5/3} \text{ when } T > 0.3 \text{ sec.} \end{aligned}$$

Where:

η = factor of damping adjustment (Eq. 4.3)

T_2 = specific period (Table 4.7)

Total lateral seismic load, V is distributed by the program among different levels.

There are 2 stages of command specification for generating lateral loads. This is the first stage and is activated through the **DEFINE RPA LOAD** command.

General Format

DEFINE RPA (ACCIDENTAL) LOAD

A f1 rpa-spec

SELFWEIGHT

JOINT WEIGHT

joint-list WEIGHT w

...

Note: See "UBC 1994 or 1985 Load Definition" on page 438 for complete weight input definition.

Where:

rpa-spec = { Q f2 RX f3 FZ f4 STYP f5 CT f6 CRDAMP f7 (PX f8) (PZ f9) }

f1 = Seismic zone coefficient. Instead of using an integer value like 1, 2, 3 or 4, use the fractional value like 0.08, 0.15, 0.2, 0.3, 0.05, etc.

f2 = Importance factor

f3 = Coefficient R for lateral load in X direction – table 4.3

f4 = Coefficient R for lateral load in Z direction – table 4.3

f5 = Soil Profile Type

f6 = Coefficient from table 4.6 of RPA 99

f7 = Damping factor

f8 = Optional Period of structure (in sec) in X direction

f9 = Optional Period of structure (in sec) in Z direction (or Y if SET Z UP is used) to be used as fundamental period of the structure instead of the value calculated by the program using Rayleigh method

General format to provide RPA Seismic load in any load case:

LOAD i

RPA LOAD {X | Y | Z} (f10) (ACC f11)

Where:

i = the load case number

f10 = factor to multiply horizontal seismic load.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

f_{11} = multiplying factor for Accidental Torsion, to be used to multiply the RPA accidental torsion load (default = 1.0). May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

Note: If the **ACCIDENTAL** option is specified, the accidental torsion will be calculated per the RPA specifications. The value of the accidental torsion is based on the "center of mass" for each level. The "center of mass" is calculated from the SELFWEIGHT, JOINT WEIGHTs and MEMBER WEIGHTs you have specified.

Example

```
DEFINE RPA LOAD
A 0.15 Q 1.36 STYP 2 RX 3 RZ 4 CT 0.0032 -
CRDAMP 30 PX .027 PZ 0.025
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
RPA LOAD X 1.0
```

5.31.2.10 Canadian Seismic Code (NRC) - 1995

This set of commands may be used to define the parameters for generation of equivalent static lateral loads for seismic analysis per National Building Code (NRC/CNRC) of Canada- 1995 edition. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the **SET Z UP** command in Section 5.5). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Methodology

The minimum lateral seismic force or base shear (V) is automatically calculated by STAAD using the appropriate equation(s); namely sentence 4, section 4.1.9.1 of NRC.

$$V = 0.6 \cdot V_e / R$$

Where:

V_e , the equivalent lateral seismic force representing elastic response (per sentence 5, section 4.1.9.1) is given by:

$$V_e = v \cdot S \cdot I \cdot W$$

Where:

v = Zonal velocity ratio per appendix C

S = Seismic Response Factor per table 4.9.1.A

I = Seismic importance factor per sentence 10 section 4.1.9.1

F = Foundation factor conforming to Table 4.9.1.C and sentence 11 section 4.1.9.1

W = Total load lumped as weight per sentence 2 section 4.1.9.1

R = Force modification factor conforming to Table 4.9.1.B that reflects the capability of a structure to dissipate energy through inelastic behavior.

STAAD utilizes the following procedure to generate the lateral seismic loads.

1. User provides seismic zone co-efficient and desired "nrc-spec" (1995) through the DEFINE NRC LOAD command.
2. The program calculates the fundamental period(T) of the structure by
 - a. finding out whether the structure being analysed is a moment resisting frame made primarily of steel or of concrete or it is a structure of any other type. Alternatively, the software uses the optional parameter CT if provided. The calculation is done per sentence 7(a) & 7(b) of section 4.1.9.1.
 - b. using the Rayleigh method or using the optional parameters PX , PZ – if provided. The stipulations of sentence 7(c) of section 4.1.9.1 are also considered while calculating.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

- c. taking the conservative value of T between those calculated by methods (a) and (b) above.
3. The program finds out the value of Seismic Response Factor(S) per table 4.9.1.A utilizing the values of T as calculated above and the values of ZA & ZV input by the user.
4. The program calculates V per sentence 4 section 4.1.9.1. W is obtained from the weight data (SELFWEIGHT, JOINT WEIGHT(s), etc.) provided by the user through the DEFINE NRC LOAD command. The weight data must be in the order shown.
5. The total lateral seismic load (base shear) is then distributed by the program among different levels of the structure per applicable NRC guidelines like sentence 13(a) section 4.1.9.1.

General Format

There are two stages of command specification for generating lateral loads. This is the first stage and is activated through the **DEFINE NRC LOAD** command.

```
DEFINE NRC LOAD  
nrc-spec  
SELFWEIGHT  
JOINT WEIGHT  
joint-list WEIGHT w  
...
```

Note: See "UBC 1994 or 1985 Load Definition" on page 438 for complete weight input definition.

Where:

$$nrc-spec = *{ \underline{V} f_1 \underline{ZA} f_2 \underline{ZV} f_3 \underline{RX} f_4 \underline{RZ} f_5 \underline{I} f_6 \underline{F} f_7 (\underline{CT} f_8) (\underline{PX} f_9) (\underline{PZ} f_{10}) }$$

f_1 = Zonal velocity ratio per Appendix C

f_2 = Factor for acceleration related seismic zone per Appendix C

f_3 = Factor for velocity related seismic zone per Appendix C

f_4 = Force modification factor along X-direction that reflects the capability of a structure to dissipate energy through inelastic behavior. Please refer Table 4.1.9.1B

f_5 = Force modification factor along Z-direction that reflects the capability of a structure to dissipate energy through inelastic behavior. Please refer Table 4.1.9.1B

f_6 = Seismic importance factor per sentence 10 section 4.1.9.1

f_7 = Foundation factor conforming to Table 4.1.9.1C and sentence 11 section 4.1.9.1

f_8 = Factor to be used to calculate the fundamental period of structure .This is an optional parameter.

f_{10} = Period of structure (in seconds) in the X- direction. This is an optional parameter.

f_{11} = Period of structure (in seconds) in the Z- direction (or Y if SET Z UP is used). This is an optional parameter.

w = joint weight associated with list

v_1, v_2, v_3 = Used when specifying a uniformly distributed load with a value of v_1 starting at a distance of v_2 from the start of the member and ending at a distance of v_3 from the start of the member. If v_2 and v_3 are omitted, the load is assumed to cover the entire length of the member.

v_4, v_5 = Used when specifying a concentrated force with a value of v_4 applied at a distance of v_5 from the start of the member. If v_5 is omitted, the load is assumed to act at the center of the member.

p_1 = weight per unit area for the plates selected. Assumed to be uniform over the entire plate.

Element Weight is used if plate elements are part of the model, and uniform pressures on the plates are to be considered in weight calculation.

Floor Weight is used if the pressure is on a region bounded by beams, but the entity which constitutes the region, such as a slab, is not defined as part of the structural model. It is used in the same sort of situation in which one uses FLOOR LOADS (see Section 5.32.4 of STAAD Technical Reference Manual for details of the Floor Load input).

The weights have to be input in the order shown.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Generation of NRC Load

The load so defined as above is applied on the structure in the NRC loadcases. These loadcases have to be the first loadcases in the input file. Built-in algorithms will automatically distribute the base shear among appropriate levels and the roof per the relevant code specifications.

The following general format should be used to generate loads in a particular direction.

```
LOAD i
NRC LOAD { X | Y | Z } (f1)
```

Where:

i = load case number

f₁ = factor to be used to multiply the NRC Load (default = 1.0). May be negative.

Notes

1. By providing either PX or PZ or both, you may override the period calculated by STAAD using Rayleigh method. If you do not define PX or PZ, the period for Method 2(b) above will be calculated by the program using Rayleigh method and the stipulations of sentence 7(c) of section 4.1.9.1
2. Some of the items in the output for the NRC analysis are explained below.

T_a = Time period calculated per sentence 7(a) or 7(b) of section 4.1.9.1

T_c = Time period calculated per sentence 7(c) of section 4.1.9.1

CALC / USED PERIOD

The CALC PERIOD is the period calculated using the Rayleigh method. For NRC in the x-direction, the USED PERIOD is PX. For the NRC in the z-direction (or Y direction if SET Z UP is used), the USED PERIOD is PZ. If PX and PZ are not provided, then the used period is the same as the calculated period for that direction. The used period is the one utilized to find out the value of S.

3. In the analysis for NRC loads, all the supports of the structure have to be at the same level and have to be at the lowest elevation level of the structure.

Example

```

DEFINE NRC LOAD
V 0.2 ZA 4 ZV 4 RX 4 RZ 4 I 1.3 F 1.3 CT 0.35 PX 2 PZ 2
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 7
49 TO 64 WEIGHT 3.5
LOAD 1 EARTHQUAKE ALONG X
NRC LOAD X 1.0
PERFORM ANALYSIS PRINT LOAD DATA
CHANGE

```

See also Example Problem EXAMP14_NRC.STD

5.31.2.11 Canadian Seismic Code (NRC) – 2005 Volume 1

This set of commands may be used to define the parameters for generation of equivalent static lateral loads for seismic analysis per National Building Code (NRC/CNRC) of Canada- 2005 Volume 1 edition. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the **SET Z UP** command in Section 5.5). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Methodology

The minimum lateral seismic force or base shear (V) is automatically calculated by STAAD using the appropriate equation (s).

$$V = \frac{S(T_a)M_v I_E W}{R_d R_o}$$

as per section 4.1.8.11(2) of NBC of Canada 2005, Volume 1

Except that V shall not be less than:

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

$$V_{\min} = \frac{S(2.0)M_v I_E W}{R_d R_o}$$

and for an $R_d = 1.5$, V need not be greater than:

$$V_{\max} = \frac{2S(0.2)I_E W}{3 R_d R_o}$$

(i.e., the upper limit of V)

Description of the terms of the equation to calculate V :

- T_a is the fundamental lateral period in the direction under consideration and is determined as:
 - a. For moment-resisting frames that resist 100% of the required lateral forces and where the frame is not enclosed by or adjoined by more rigid elements that would tend to prevent the frame from resisting lateral forces, and is calculated by the empirical formulae as described below provided h_n is in meter
 - i. $0.085(h_n)^{3/4}$ for steel moment frames,
 - ii. $0.075(h_n)^{3/4}$ for concrete moment frames, or
 - b. The period is also calculated in accordance with the Rayleigh method but could be overridden by user specified time period (PX, PZ).

If design spectral acceleration, $S(T_a)$, calculated considering structural time period calculated based on method (b) is greater than 0.8 times the same calculated considering structural time period calculated based on method (a), the former is used for further calculation. Otherwise, the later time period is used.
 - c. Other established methods of mechanics using a structural model that complies with the requirements of Sentence 4.1.8.3.(8), except that
 - i. for moment resisting frames, T_a shall not be greater than 1.5 times that determined in Clause (a).
 - ii. for braced frames, T_a shall not be greater than 2.0 times that determined in Clause (b).
 - iii. for shear wall structures, T_a shall not be greater than 2.0 times that determined in Clause (c), and
 - iv. for the purpose of calculating the deflections, the period without the upper limit specified is referred from Appendix A.
- $S(T_a)$ is the design spectral acceleration and is determined as follows, using

linear interpolation for intermediate values of T_a :

$$\begin{aligned} S(T_a) &= F_a S_a(0.2) \text{ for } T_a \leq 0.2s \\ &= F_v S_a(0.5) \text{ or } F_a S_a(0.2), \text{ whichever is smaller for } T_a = 0.5s \\ &= F_v S_a(1.0) \text{ for } T_a = 1.0s \\ &= F_v S_a(2.0) \text{ for } T_a = 2.0s \\ &= F_v S_a(2.0)/2 \text{ for } T_a \geq 4.0s \end{aligned}$$

The above terms $S_a(0.2)$, $S_a(0.5)$, $S_a(1.0)$ and $S_a(2.0)$ are the Seismic Data and are obtained as user input from the *Table C-2*.

Based on the above values of $S_a(T_a)$, F_a and F_v the acceleration- and velocity based site coefficients are determined from the Tables 4.1.8.4.B and 4.1.8.4.C, using linear interpolation for intermediate values of $S_a(0.2)$ and $S_a(1.0)$. It is to be mentioned that, these are the user inputs based on the site classes from A to E and the desired $S_a(0.2)$ and $S_a(1.0)$ values as required as per the above equations.

- M_v is the factor to account for higher mode effect on base shear and the associated base overturning moment reduction factor is J which are obtained as user input from the Table 4.1.8.11. To get this higher mode factor (M_v) and numerical reduction coefficient for base overturning moment (J), user has to get the ratios of $S_a(0.2)/S_a(2.0)$ as also the “Type of Lateral Resisting System”.

For values of M_v between fundamental lateral periods, T_a of 1.0 and 2.0 s, the product $S(T_a) \cdot M_v$ shall be obtained by linear interpolation.

Values of J between fundamental lateral periods, T_a of 0.5 and 2.0 s shall be obtained by linear interpolation.

- I_E is the earthquake importance factor of the structure and is determined from the Table 4.1.8.5. This is an user input depending on Importance Category and ULS / SLS
- W is the weight of the building and shall be calculated internally using the following formula:

$$W = \sum_{i=1}^n W_i$$

Where W_i is the portion of W that is located at or assigned to level i .

- R_d is the ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behavior as described in article 4.1.8.9.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

- R_o is the over-strength-related force modification factor accounting for the dependable portion of reserve strength in a structure designed according to the provision of the Article 4.1.8.9.

These R_d and R_o values are the user inputs depending on the type of SFRS. As per 4.1.8.11(6), the total lateral seismic force, V , shall be distributed such that a portion, F_t shall be concentrated at the top of the building, where,

$$F_t = 0.07T_a V$$

but F_t is not greater than $0.25V$ and $F_t = 0$ when T_a is not greater than $0.7s$.

The remainder ($V - F_t$), shall be distributed along the height of the building, including the top level, in accordance with the following formula [as per section 4.1.8.11(6)]:

$$F_x = \frac{(V - F_t)W_x h_x}{\sum_{i=1}^n W_i h_i}$$

Where:

F_x is the lateral force applied to level x .

F_t is the portion of V to be concentrated at the top of the structure.

W_i, W_x are the portion of W that is located at or assigned to level i or x respectively.

h_i, h_x are the height above the base ($i=0$) to level i or x respectively.

level i is any level in the building, $i=1$ for first level above the base.

level n is uppermost in the main portion of the structure.

General Format

STAAD utilizes the following format to generate the lateral seismic loads.

```
DEFINE NRC2005LOAD  
nrc-spec  
SELFWEIGHT  
JOINT WEIGHT  
joint-list WEIGHT w  
...
```

Note: See "UBC 1994 or 1985 Load Definition" on page 438 for complete weight input definition.

Where:

nrc-spec = { SA1 f1 SA2 f2 SA3 f3 SA4 f4 MVX f5 MVZ f6 JX f7
JZ f8 IE f9 RDX f10 ROX f11 RDZ f12 ROZ f13 SCLASS f14 }

$f_1, f_2, f_3,$ and f_4 = Seismic Data as per Table C-2 and are you input.

f_5, f_6 = The higher mode factor along X and Z directions respectively. These are user input and please refer Table 4.1.8.11.

f_7, f_8 = The numerical reduction coefficient for base overturning moment along X and Z directions respectively. These are user input and please refer Table 4.1.8.11.

f_9 = the earthquake importance factor of the structure. This is a user input depending on Importance Category and ULS / SLS and please refer Table 4.1.8.11.

f_{10}, f_{12} = are the ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behavior as described in article 4.1.8.9. along X and Z directions respectively. These are user input and please refer Table 4.1.8.9.

f_{11}, f_{13} = are the over strength-related force modification factor accounting for the dependable portion of reserve strength in a structure designed according to the provision of the Article 4.1.8.9. along X and Z directions respectively. These are user input and please refer Table 4.1.8.9.

f_{14} = is the Site Class starting from A to E. This is an user input. F_a and F_v are determined based on Site Class as per Table 4.1.8.4.B and Table 4.1.8.4.C.

w = joint weight associated with list

v_1, v_2, v_3 = Used when specifying a uniformly distributed load with a value of v_1 starting at a distance of v_2 from the start of the member and ending at a distance of v_3 from the start of the member. If v_2 and v_3 are omitted, the load is assumed to cover the entire length of the member.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

v_4, v_5 = Used when specifying a concentrated force with a value of v_4 applied at a distance of v_5 from the start of the member. If v_5 is omitted, the load is assumed to act at the center of the member.

p_1 = weight per unit area for the plates selected. Assumed to be uniform over the entire plate.

Element Weight is used if plate elements are part of the model, and uniform pressures on the plates are to be considered in weight calculation.

Floor Weight is used if the pressure is on a region bounded by beams, but the entity which constitutes the region, such as a slab, is not defined as part of the structural model. It is used in the same sort of situation in which one uses FLOOR LOADS (See "Floor Load Specification" on page 562 for details). The weights have to be input in the order shown.

Example

```
DEFINE NRC 2005 LOAD
SA1 .33 SA2 .25 SA3 .16 SA4 .091 MVX 1.2 MVZ 1.5 JX .7 JZ
.5 IE 1.3 -
RDX 4.0 ROX 1.5 RDZ 3.0 ROZ 1.3 SCLASS 4
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 7
49 TO 64 WEIGHT 3.5
LOAD 1 EARTHQUAKE ALONG X
NRC LOAD X 1.0
PERFORM ANALYSIS PRINT LOAD DATA
CHANGE
```

See Example Problem EXAMP14_NRC_2005.STD

5.31.2.12 Turkish Seismic Code

This set of commands may be used to define the parameters for generation of equivalent static lateral loads for seismic analysis per the specifications laid out in *Specification for Structures to be Built in Disaster Areas Part – III – Earthquake*

Disaster Prevention Amended on 2.7.1998, Official Gazette No. 23390 (English Translation). This is referred to as the Turkish Seismic Provisions.

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the **SET Z UP** command in Section 5.5). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Base Shear

The minimum lateral seismic force or base shear, V_t , is automatically calculated per equation 6.4 in section 6.7.1:

$$V_t = \frac{WA(T_1)}{R_a(T_1)}$$

Where:

T_1 is the fundamental time period of the structure.

Except that V_t shall not be less than:

$$V_{t,\min} = 0.10A_0IW$$

Seismic Load Reduction Factor, $R_a(T_1)$, in above equation is determined based on the following equations 6.3a and 6.3b in the code:

$$R_a(T_1) = \begin{cases} 1.5 + \left(R - 1.5\right) \frac{T_1}{T_a} & \text{when } 0 \leq T_1 \leq T_a \\ R & \text{when } T_1 > T_a \end{cases}$$

The Structural Behavior Factor in either direction, RX and RZ, are provided through user input (variables f5 and f5) along the direction of calculation. Spectrum Characteristics Period, TA and TB, are also provided by user through the parameters (variables f2 and f3).

Where:

R_a is the Structural Behavior Factor in either direction, RX and RZ, are provided through user input (variables f5 and f5) along the direction of calculation.

T_a , T_b are the Spectrum Characteristics Period are provided by user through the parameters (variables f2 and f3).

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

T_1 is the fundamental Lateral period in the direction under consideration and is determined as:

- a. Calculated by the empirical formulae as described below provided h_n is in meter:

$$T_1 = C_T [h_n]^{3/4}$$

Where C_T is assumed to be 0.075 for steel moment frames, 0.085 for concrete moment frames, or any user specified value.

- b. The period is also calculated in accordance with the Rayleigh method but could be overridden by user specified time period (PX, PZ).

The time period calculated based on method (a) is used in further calculation unless it is greater than 1.0 sec and 1.3 times of this is greater than the same calculated based on method (b). In that case time period calculated based on method (b) is used.

$A(T_1)$ is the Spectral Acceleration Coefficient is determined as follows as per eq. 6.1,

$$A(T_1) = A_0 I S(T_1)$$

A_0 and I in above equation are Effective Ground Acceleration Coefficient and Building Importance Factor are provided by the user through the load definition parameter and could be found in table 6.2 and 6.3 respectively in the code.

$S(T_1)$ is the Spectrum Coefficient is found by following equations, could be found in eq. 6.2a, 6.2b and 6.3c in original code.

$$S(T_1) = \begin{cases} 1 + 1.5 \frac{T_1}{T_A} & \text{when } 0 \leq T < T_A \\ 2.5 & \text{when } T_A \leq T \leq T_B \\ 2.5 \left(\frac{T_B}{T_1} \right)^{0.8} & \text{when } T_B < T \end{cases}$$

W is the weight of the building and shall be calculated internally using the following formula:

$$W = \sum_{i=1}^n W_i$$

W_i is the portion of W that is located at or assigned to level i .

Vertical Distribution

As per 4.1.8.11(6), the total lateral seismic force, V_t , shall be distributed such that a portion, F_t shall be concentrated at the top of the building, where,

$$\Delta F_N = 0.07 T_1 V_t$$

but ΔF_N is not greater than $0.20V_t$

and $\Delta F_N = 0$ when $H_N \leq 25$ m.

The remainder ($V - \Delta F_N$), shall be distributed along the height of the building, including the top level, in accordance with the following formula (per equation 6.9):

$$F_i = (V_t - \Delta F_N) w_i H_i / \sum w_j H_j$$

Where:

F_i is the lateral force applied to level i .

ΔF_N is the portion of V_t to be concentrated at the top of the structure.

w_i, w_j are the portion of W that is located at or assigned to level i or j respectively.

level i is any level in the building, $i = 1$ for first level above the base.

level N is uppermost in the main portion of the structure.

General Format

STAAD utilizes the following format to generate the lateral seismic loads.

DEFINE TURKISH LOAD

tur-spec

SELFWEIGHT

JOINT WEIGHT

joint-list **WEIGHT w**

...

Note: See "UBC 1994 or 1985 Load Definition" on page 438 for complete weight input definition.

Where:

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

tur-spec = { A **f1** TA **f2** TB **f3** I **f4** RX **f5** RZ **f6** (CT **f7**) (PX **f8**) (PZ **f9**) }

f_1 = Effective Ground Acceleration Coefficient, A_0 . Refer table 6.2

f_2, f_3 = Spectrum Characteristic Periods, T_A and T_B . These are user input and found in Table 6.4

f_5 and f_6 = Structural Behavior Factors (R) along X and Z directions respectively. These are user input and please refer Table 6.5

f_4 = the earthquake importance factor of the structure.

f_{10}, f_{12} = are the ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behavior as described in article 4.1.8.9. along X and Z directions respectively. These are user input and please refer Table 4.1.8.9.

f_8 = Optional CT value to calculate time period. See section 1617.4.2.1, equation 16-39 of IBC 2000 and section 9.5.5.3.2, equation 9.5.5.3.2-1 of ASCE 7-02.

f_9 = Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure instead of the value derived from section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02.

f_{10} = Optional Period of structure (in sec) in Z-direction to be used as fundamental period of the structure instead of the value derived from section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02.

w = joint weight associated with list

v_1, v_2, v_3 = Used when specifying a uniformly distributed load with a value of v_1 starting at a distance of v_2 from the start of the member and ending at a distance of v_3 from the start of the member. If v_2 and v_3 are omitted, the load is assumed to cover the entire length of the member.

v_4, v_5 = Used when specifying a concentrated force with a value of v_4 applied at a distance of v_5 from the start of the member. If v_5 is omitted, the load is assumed to act at the center of the member.

p_1 = weight per unit area for the plates selected. Assumed to be uniform over the entire plate.

Element Weight is used if plate elements are part of the model, and uniform pressures on the plates are to be considered in weight calculation.

Floor Weight is used if the pressure is on a region bounded by beams, but the entity which constitutes the region, such as a slab, is not defined as part of the structural model. It is used in the same sort of situation in which one uses FLOOR LOADS (See "Floor Load Specification" on page 562 for details).

Example

```

DEFINE TUR LOAD
A 0.40 TA 0.10 TB 0.30 I 1.4 RX 3.0 RZ 3.0
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 7
49 TO 64 WEIGHT 3.5
LOAD 1 EARTHQUAKE ALONG X
TUR LOAD X 1.0
PERFORM ANALYSIS PRINT LOAD DATA
CHANGE

```

See Example Problem EXAMP14_TURKISH.STD

5.31.2.13 IBC 2006/2009 Seismic Load Definition

The specifications of the seismic loading chapters of the International Code Council's IBC 2006 & 2009 code and the ASCE 7-05 (including Supplement #2) code for seismic analysis of a building using a static equivalent approach have been implemented as described in this section. Depending on the definition, equivalent lateral loads will be generated in the horizontal direction(s).

Note: This feature requires STAAD.Pro 2007 (Build 03) or higher. Refer to AD.2007-03.1.6 of the What's New section for additional information on using this feature.

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the **SET Z UP** command in Section 5.5). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

The rules described in section 1613 of the ICC IBC-2006 code (except 1613.5.5) have been implemented. This section directs the engineer to the ASCE 7-2005 code. The specific section numbers of ASCE 7— those which are implemented, and those which are not implemented—are shown in the table below.

Table 5-14: Sections of IBC 2006 implemented and omitted in the program

Implemented sections of IBC 2006/2009 (ASCE 7-2005)	Omitted sections of IBC 2006/2009 (ASCE 7-2005)
11.4	12.8.4.1
11.5	12.8.4.3 and onwards
12.8	

Additionally, Supplement #2 of ASCE 7-05—as referenced by IBC 2009—specifies a different equation to be used for the lower bound of the seismic response coefficient, which has also been implemented.

Steps used to calculate and distribute the base shear are as follows:

1. The Time Period of the structure is calculated based on section 12.8.2.1 of ASCE 7-05 (IBC 2006/2009). This is reported in the output as T_a .
2. The period is also calculated in accordance with the Rayleigh method. This is reported in the output as T .
3. you may override the Rayleigh based period by specifying a value for PX or PZ (Items f7 and f8) depending on the direction of the IBC load.
4. The governing Time Period of the structure is then chosen between the above two periods, and the additional guidance provided in section 12.8.2 of ASCE 7-05 (IBC 2006). The resulting value is reported as "Time Period used" in the output file.
5. The Design Base Shear is calculated based on equation 12.8-1 of ASCE 7-05 (IBC 2006). It is then distributed at each floor using the rules of clause 12.8.3, equations 12.8-11, 12.8-12 and 12.8-13 of ASCE 7-05.
6. If the **ACCIDENTAL** option is specified, the program calculates the additional torsional moment. The lever arm for calculating the torsional moment is obtained as 5% of the building dimension at each floor level perpendicular

to the direction of the IBC load (section 12.8.4.2 of ASCE 7-05 for IBC 2006). At each joint where a weight is located, the lateral seismic force acting at that joint is multiplied by this lever arm to obtain the torsional moment at that joint.

7. The amplification of accidental torsional moment, as described in Section 12.8.4.3 of the ASCE 7-05 code, is not implemented.
8. The story drift determination as explained in Section 12.8.6 of the ASCE 7-05 code is not implemented in STAAD.

Methodology

The design base shear is computed in accordance with the following equation (equation 12.8-1 of ASCE 7-05):

$$V = C_s W$$

The seismic response coefficient, C_s , is determined in accordance with the following equation (equation 12.8-2 of ASCE 7-05):

$$C_s = S_{DS}/[R/I_E]$$

For IBC 2006, C_s need not exceed the following limits defined in ASCE 7-05 (equations 12.8-3 and 12.8-4):

$$C_s = S_{D1}/[T \cdot (R/I)] \text{ for } T \leq T_L$$

$$C_s = S_{D1} \cdot T_L/[T^2(R/I)] \text{ for } T > T_L$$

However, C_s shall not be less than (equation 12.8-5 of ASCE 7-05, supplement #2):

$$C_s = 0.044 \cdot S_{DS} \cdot I \geq 0.01$$

In addition, per equation 12.8-6 of ASCE 7-05, for structures located where S_1 is equal to or greater than 0.6g, C_s shall not be less than

$$C_s = 0.5 \cdot S_1/(R/I)$$

For an explanation of the terms used in the above equations, please refer to the IBC 2006/2009 and ASCE 7-05 codes.

General Format

There are two stages of command specification for generating lateral loads. This is the first stage and is activated through the **DEFINE IBC 2006 LOAD** command.

DEFINE IBC 2006 (ACCIDENTAL) LOAD

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

map-spec *ibc06-spec*

SELFWEIGHT

JOINT WEIGHT

joint-list WEIGHT *w*

...

Note: See "UBC 1994 or 1985 Load Definition" on page 438 for complete weight input definition.

Where:

map-spec = { ZIP f_1 | LAT f_2 LONG f_3 | SS f_4 S1 f_5 }

ibc06-spec = { RX f_6 RZ f_7 I f_8 TL f_9 SCLASS f_{10} (CT f_{11}) (PX f_{12}) (PZ f_{13}) (K f_{14}) (FA f_{15}) (FV f_{16}) }

f_1 = The zip code of the site location to determine the latitude and longitude and consequently the S_s and S_1 factors. (ASCE 7-05 Chapter 22).

f_2, f_3 = The latitude and longitude, respectively, of the site used with the longitude to determine the S_s and S_1 factors. (ASCE 7-05 Chapter 22).

f_4 = The mapped MCE for 0.2s spectral response acceleration. (IBC 2006/2009 Clause 1613.5.1, ASCE 7-05 Clause 11.4.1).

f_5 = The mapped MCE spectral response acceleration at a period of 1 second as determined in accordance with Section 11.4.1 ASCE7-05

f_6 = The response modification factor for lateral load along the X direction, (ASCE Table 12.2.1). This is the value used in "R" in the equations shown above for calculating C_s .

f_7 = The response modification factor for lateral load along the Z direction, (ASCE Table 12.2.1) This is the value used in "R" in the equations shown above for calculating C_s .

f_8 = Occupancy importance factor. (IBC 2006/2009 Clause 1604.5, ASCE 7-05 Table 11.5-1)

f_9 = Long-Period transition period in seconds. (ASCE 7-05 Clause 11.4.5 and Chapter 22).

f_{10} = Site class. Enter 1 through 6 in place of A through F, see table below. (IBC 2006/2009 Table 1613.5.2, ASCE 7-05 Section 20.3)

The Soil Profile Type parameter **SCLASS** can take on values from 1 to 6. These relate to the values shown in Site Class Definitions Table in the following manner:

Table 5-15: Values of IBC soil class (SCLASS) used in STAAD

IBC Class	SCLASS value
A	1
B	2
C	3
D	4
E	5
F	6

f_{11} = Optional CT value to calculate time period. (ASCE 7-05 Table 12.8-2).

f_{12} = Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. (ASCE 7-05 Table 12.8-2).

f_{13} = Optional Period of structure (in sec) in Z-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. (ASCE 7-05 Table 12.8-2).

f_{14} = Exponent value, x , used in equation 12.8-7, ASCE 7. (ASCE 7-2005 table 12.8-2 page 129).

f_{15} = Optional Short-Period site coefficient at 0.2s. Value must be provided if **SCLASS** set to F (i.e., 6). (IBC 2006/2009 Clause 1613.5.3, ASCE 7-05 Section 11.4.3).

f_{16} = Optional Long-Period site coefficient at 1.0s. Value must be provided if **SCLASS** set to F (i.e., 6). (IBC 2006/2009 Clause 1613.5.3, ASCE 7-05 Section 11.4.3).

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Example 1

```
DEFINE IBC 2006
LAT 38.0165 LONG -122.105 I 1.25 RX 2.5 RZ 2.5 SCLASS 4 -
TL 12 FA 1 FV 1.5
SELFWEIGHT
JOINT WEIGHT
51 56 93 100 WEIGHT 650
MEMBER WEIGHT
151 TO 156 158 159 222 TO 225 324 TO 331 UNI 45
```

Example 2

The following example shows the commands required to enable the program to generate the lateral loads. Refer to Section 5.32.12 of the Technical Reference Manual for this information.

```
LOAD 1 (SEISMIC LOAD IN X DIRECTION)
IBC LOAD X 0.75
LOAD 2 (SEISMIC LOAD IN Z DIRECTION)
IBC LOAD Z 0.75
```

5.31.2.14 Chinese Static Seismic per GB50011-2001

This set of commands may be used to define and generate static equivalent seismic loads as per Chinese specifications GB50011-2001. This load uses a static equivalent approach, similar to that found in the UBC. Depending on this definition, equivalent lateral loads will be generated in the horizontal direction (s).

Note: This feature is available in STAAD.Pro V8i (SELECTseries 1) (release 20.07.06) and later.

The seismic load generator can be used to generate lateral loads in the X and Z directions for Y up and the X and Y directions for Z up; where Y up or Z up is the

vertical axis parallel to the direction of gravity loads (See the **SET Z UP** command in Section 5.5).

Note: All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

This method of seismic load generation is limited in use to buildings not taller than 40 meters, with deformations predominantly due to shear, and a rather uniform distribution of mass and stiffness in elevation. Alternately, for buildings modeled as a single-mass system, a simplified method such as this base shear method, may be used.

Gravity Loads for Design

In the computation of seismic action, the representative value of gravity load of the building shall be taken as the sum of characteristic values of the weight of the structure and members plus the combination values of variable loads on the structure. The combination coefficients for different variable loads shall be taken from the following table.

Table 5-16: Combinations of different load effects per GB50011-2001

Type of Variable	land Combination coefficient	
Snow load	0.5	
Dust load on roof	0.5	
Live load on roof	Not considering	
Live load on the floor, calculated according to actual state	1.0	
Live load on the floor, calculated according to equivalent uniform state	Library, archives	0.8
	Other civil buildings	0.5

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Type of Variable	land Combination coefficient	
	Gravity for hanging object of crane	Hard hooks
	Soft hooks	Not considering

Seismic influence coefficient

This shall be determined for building structures according to the Intensity, Site-class, Design seismic group, and natural period and damping ratio of the structure. The maximum value of horizontal seismic influence coefficient shall be taken from Table 2.2; the characteristic period shall be taken as Table 2.3 according to Site-class and Design seismic group, that shall be increased 0.05s for rarely earthquake of Intensity 8 and 9.

Table 5-17: Earthquake influence per GB50011-2001

Earthquake influence	Intensity 6	Intensity 7	Intensity 8	Intensity 9
Frequent earthquake	0.04	0.08 (0.12)	0.16(0.24)	0.32
Rarely earthquake	-	0.50(0.72)	0.90(1.20)	1.40

Note: The values in parenthesis are separately used for where the design basic seismic acceleration is 0.15g and 0.30g.

Table 5-18: Earthquake group per GB50011-2001

Earthquake Group	Site class			
	I	II	III	IV
1	0.2- 5	0.3- 5	0.4- 5	0.6- 5
2	0.3- 0	0.4- 0	0.5- 5	0.7- 5

Earthquake Group	Site class			
	I	II	III	IV
3	0.3- 5	0.4- 5	0.6- 5	0.9- 0

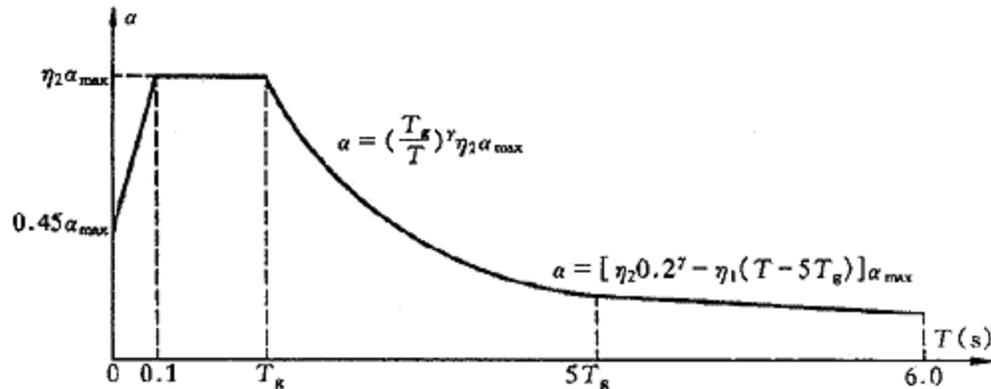
Calculation of seismic influence coefficient

The design base shear is computed in accordance with the equations shown below.

The damping adjusting and forming parameters on the building seismic influence coefficient curve (Fig.2.1) shall comply with the following requirements:

1. The damping ratio of building structures shall select 0.05 except otherwise provided, the damping adjusting coefficient of the seismic influence coefficient curve shall select 1.0, and the coefficient of shape shall conform to the following provisions:
 1. Linear increase section, whose period (T) is less than 0.1 s;
 2. Horizontal section, whose period from 0. is thought to characteristic period, shall select the maximum value (α_{max});
 3. Curvilinear decrease section, whose period from characteristic period thought to 5 times of the characteristic period, the power index (γ) shall choose 0.9.
 4. Linear decrease section, whose period from 5 times characteristic period thought to 6s, the adjusting factor of slope (η_1) shall choose 0.02.

Figure 5-33: Seismic influence coefficient curve



Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

2. When the damping adjusting and forming parameters on the seismic influence coefficient curve shall comply with the following requirements:

1. The power index of the curvilinear decreased section shall be determined according to the following equation E2.1

$$r = 0.9 + \frac{0.05 - \zeta}{0.5 + 5\zeta} \quad \text{E2.1}$$

Where:

γ - the power index of the curvilinear decrease section;

ξ - the damping ratio.

2. The adjusting factor of slope for the linear decrease section shall be determined from following equation:

$$\eta_1 = 0.02 + (0.05 - \zeta) / 8 \quad \text{E2.2}$$

Where:

η_1 - the adjusting factor of slope for the linear decrease section, when it is less than 0, shall equal 0.

3. The damping adjustment factor shall be determined according to the following equation:

$$\eta_2 = 1 + \frac{0.05 - \zeta}{0.06 + 1.7\zeta} \quad \text{E2.3}$$

Where:

η_2 - the damping adjustment factor, when it is smaller than 0.55 shall equal 0.55.

Calculation of horizontal seismic action

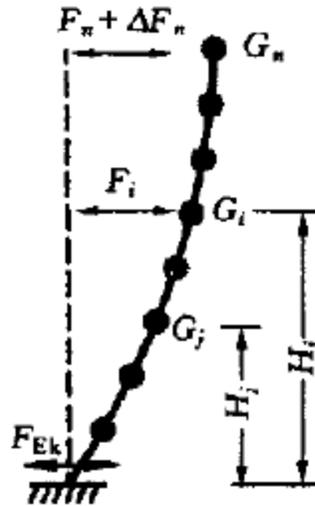
When the base shear force method is used, only one degree of freedom may be considered for each story; the characteristic value of horizontal seismic action of the structure shall be determined by the following equations (Fig. 2.2):

$$F_{Ek} = a_1 G_{eq} \quad \text{E2.4}$$

$$F_i = \frac{G_i H_i}{\sum_{j=1}^n G_j H_j} F_{Ek} (1 - \delta_n) \quad \text{E2.5}$$

$$\Delta F_n = \delta_n F_{Ek} \quad \text{E2.6}$$

Figure 5-34: Calculation of horizontal seismic action



Where:

F_{Ek} - characteristic value of the total horizontal seismic action of the structure.

α_1 - horizontal seismic influence coefficient corresponding to the fundamental period of the structure, which shall be determined by using Clause 2.3. For multistory masonry buildings and multi-story brick buildings with bottom-frames or inner-frames, the maximum value of horizontal seismic influence coefficient should be taken.

G_{eq} - equivalent total gravity load of a structure. When the structure is modeled as a single-mass system, the representative value of the total gravity load shall be used; and when the structure is modeled as a multi-mass system, the 85% of the representative value of the total gravity load may be used.

F_i — characteristic value of horizontal seismic action applied on mass i -th .

G_i, G_j — representative values of gravity load concentrated at the masses of i -th and j -th respectively, which shall be determined by Clause 2.1.

H_i, H_j - calculated height of i -th and j -th from the base of the building respectively.

δ_n — additional seismic action factors at the top of the building; for multi-story reinforced concrete buildings, it may be taken using Table 2.4; for multi-story brick buildings with inner-frames, a value of 0.2 may be used; no need to consider for other buildings

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

ΔF_n - additional horizontal seismic action applied at top of the building.

Table 2.4 Additional seismic action factors at top of the building

Note: T, is the fundamental period of the structure.

The horizontal seismic shear force at each floor level of the structure shall comply with the requirement of the following equation:

$$V_{Eki} > \lambda \sum_{j=1}^n G_j \quad E2.7$$

Where:

V_{Eki} - the floor i-th shear corresponding to horizontal seismic action characteristic value.

λ - Shear factor, it shall not be less than values in Table 2.5; for the weak location of vertical irregular structure, these values shall be multiplied by the amplifying factor of 1.15.

G_j - the representative value of gravity load in floor j-th of the structure.

Table 5-19: Minimum seismic shear factor value of the floor level per GB50011-2001

Structures	Intensity		
	7	8	9
structures with obvious torsion effect or fundamental period is less than 3.5s	0.16 (0.024)	0.032 (0.048)	0.064
Structures with fundamental period greater than 5.0s	0.012 (0.018)	0.024 (0.032)	0.040

Note:

The values may be selected through interpolation method for structures whose fundamental period is between 3.5s and 5s.

Values in the brackets are used at the regions with basic seismic acceleration as 0.15g and 0.30g respectively.

Calculation of vertical seismic action

For tall buildings for Intensity 9, the characteristic value of vertical seismic action shall be determined by the following equations (figure 2.3). The effects of vertical seismic action at floor level may be distributed in proportion of representative value of gravity load acting on the members, which should multiply with the amplified factor 1.5:

$$F_{Evk} = \alpha_{vmax} G_{eq} \quad E2.8$$

$$F_i = \frac{G_i H_i}{\sum_{j=1}^n G_j H_j} F_{Evk} \quad E2.9$$

Where:

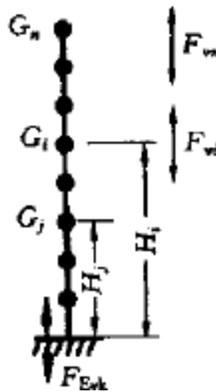
F_{Evk} = characteristic value of the total vertical seismic actions of the structure.

F_v = characteristic value of vertical seismic action at the level of mass i-th.

α_{vmax} = maximum value of vertical seismic influence coefficient, which may be taken as 65% of the maximum value of the horizontal seismic influence coefficient.

G_{eq} = equivalent total gravity load of the structure, which may be taken as 75% of the representative value of the total gravity load of the structure.

Figure 5-35: Sketch for the computation of vertical seismic action



Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Complementarities

1. Structures having the oblique direction lateral-force-resisting members and the oblique angle to major orthogonal axes is greater than 15°, the horizontal seismic action along the direction of each lateral-force-resisting member shall be considered respectively. So we could consider this through the item, the action of the oblique member could be multiplied by this factor as design force.
2. Eccentricity: similar to UBC code. The eccentricity value of gravity center on each floor should be $e_i = \pm 0.05L_i$,

where:

e_i - Eccentricity value of gravity center on i-th floor.

L_i - maximum width of calculated story of the building.

3. Structures having obviously asymmetric mass and stiffness distribution, the torsion effects caused by both two orthogonal horizontal direction seismic action shall be considered; and other structures, it is permitted that a simplified method, such as adjusting the seismic effects method, to consider their seismic torsion effects.

So we may use the to decide which whether torsion effects should be considered.

General Format

The following general format should be used to generate loads in a particular direction.

DEFINE GB (ACCIDENTAL) LOAD

**INTENSITY s_1 { FREQUENT | RARE } GROUP i_1 SCLASS i_2 (DAMP f_1)
(DELN f_2) (SF f_3) (AV f_4) (PX f_5) (PZ f_6)**

Where:

- s_1 = the Fortification Intensity (ref. table 5.1.4-1). Acceptable values are 6, 7, 7A, 8, 8A, or 9.
- i_1 = Design Seismic Group (ref. table 5.1.4-2). Acceptable values are 1, 2, or 3.
- i_2 = Site-Class (ref. table 5.1.4-2). Acceptable values are 1, 2, 3, or 4.
- f_1 = damping ratio (default = 0.05 for 5% damping)

- $f_2 = \delta_n$, Additional seismic action factor at the top of the building (default as calculated from Table 5.2.1)
- $f_3 = \lambda$, Minimum seismic shear factor of the floor (default as calculated from Table 5.2.5)
- $f_4 = \alpha_{(v,max)}$, maximum value of vertical seismic influence coefficient (default=0.0) (ref. section 5.3)

Frequency of seismic action, as specified by either FREQUENT or RARE (ref. table 5.1.2-2)

As a fraction of total vertical load is to be considered such as $0.75G_{eq}$, specify the product of the factor on maximum horizontal seismic influence factor and factor of total gravity load as f_4 . For instance,

- $\alpha_{(v,max)} = 0.65\alpha_{max}$

And

- $G_{v,eq} = 0.75G_{eq}$
- Specify f_4 as $(0.65 * 0.75)$ i.e., equal to 0.4875
- f_5 and f_6 = optional time period along two horizontal direction (X and Y, respectively).

Steps described in section 5.1.4, 5.1.5, 5.2.1, 5.2.5, 5.3.1 will be used in generating load due to seismic action.

To apply the load in any load case, following command would be used

LOAD CASE i

GB LOAD X (f_7) (ACC f_8)

Where:

i = load case number

f_7 = optional factor to multiply horizontal seismic load.

f_8 = multiplying factor for Accidental Torsion, to be used to multiply the accidental torsion load (default = 1.0). May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

Note: If the **ACCIDENTAL** option is specified, the accidental torsion will be calculated per the GB specifications. The value of the accidental torsion is based on the "center of mass" for each level. The "center of mass" is

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

calculated from the SELFWEIGHT, JOINT WEIGHTs and MEMBER WEIGHTs you have specified.

Example Output

```
*****
* *
* EQUIV. SEISMIC LOADS AS PER SEISMIC DESIGN CODE FOR BUILDINGS *
* (GB50011-2001) OF CHINA ALONG X *
* T CALCULATED = 0.252 SEC. T USER PROVIDED = 1.200 SEC. *
* T USED = 1.200 SEC. *
* MAX. HORIZONTAL SEISMIC INFLUENCE COEFFICIENT = 0.240 *
* CHARACTERISTIC PERIOD = 0.750 SEC. *
* DAMPING RATIO = 0.030 POWER INDEX (GAMMA) = 0.931 *
* DAMPING ADJUSTMENT FACTOR (ETA2) = 1.180 *
* ADJUSTING FACTOR (ETA1) = 0.022 *
* HORIZONTAL SEISMIC INFLUENCE COEFFICIENT (ALPHA1) = 0.183 (18.288%) *
* MINIMUM SHEAR FACTOR AS PER SEC. 5.2.5 (LAMBDA) = 0.050 ( 5.000%) *
* TOTAL HORIZONTAL SEISMIC ACTION = *
* = 0.183 X 285.529 = 52.218 KIP *
* DESIGN BASE SHEAR = 0.750 X 52.218 *
* = 39.164 KIP *
* ADDITIONAL SEISMIC ACTION FACTOR (DELTAN) = 0.020 *
* VERTICAL SEISMIC INFLUENCE COEFFICIENT (ALPHA,VMAX) = -0.108 *
* TOTAL VERTICAL SEISMIC ACTION = *
* = -0.108 X 285.529 = -30.837 KIP *
* TOTAL DESIGN VERTICAL LOAD = 0.750 X -30.837 *
* = -23.128 KIP *
* *
*****
CHECK FOR MINIMUM LATERAL FORCE AT EACH FLOOR [GB50011-2001:5.2.5]
LOAD - 1 FACTOR - 0.750
FLOOR LATERAL GRAVITY LAMBDA LAMBDA ADJUSTMENT
HEIGHT (KIP ) LOAD (KIP ) LOAD (KIP ) (%) MIN (%) FACTOR
-----
30.000 23.406 79.045 29.61 5.00 1.00
20.000 19.208 180.888 10.62 5.00 1.00
10.000 9.604 282.731 3.40 5.00 1.47
JOINT LATERAL TORSIONAL VERTICAL LOAD - 1
LOAD (KIP ) MOMENT (KIP -FEET) LOAD (KIP ) FACTOR - 0.750
```

```

-----
17 FX 0.541 MY 0.000 FY -0.221
18 FX 0.663 MY 0.000 FY -0.271
19 FX 0.663 MY 0.000 FY -0.271
20 FX 0.541 MY 0.000 FY -0.221
21 FX 0.663 MY 0.000 FY -0.271
22 FX 0.785 MY 0.000 FY -0.321
23 FX 0.785 MY 0.000 FY -0.321
24 FX 0.663 MY 0.000 FY -0.271
25 FX 0.663 MY 0.000 FY -0.271
26 FX 0.785 MY 0.000 FY -0.321
27 FX 0.785 MY 0.000 FY -0.321
28 FX 0.663 MY 0.000 FY -0.271
29 FX 0.541 MY 0.000 FY -0.221
30 FX 0.663 MY 0.000 FY -0.271
31 FX 0.663 MY 0.000 FY -0.271
32 FX 0.541 MY 0.000 FY -0.221
-----
TOTAL = 10.602 0.000 -0.221 AT LEVEL 10.000 FEET

```

5.31.3 Definition of Wind Load

This set of commands may be used to define some of the parameters for generation of wind loads on the structure. See Section 5.32.12 for the definition of wind direction and the possible surfaces to be loaded. Section 1.17.3 of this manual describes the two types of structures on which this load generation can be performed.

The wind load generator can be used to generate lateral loads in the horizontal—X and Z (or Y if Z up) —directions only.

The graphical user interface can be used to automatically generate the appropriate intensity values via the ASCE-7: Wind Load dialog.

General Format

```

DEFINE WIND LOAD
TYPE j
INTENSITY p1 p2 p3 ... pn HEIGHT h1 h2 h3 ... hn
EXPOSURE e1 { JOINT joint-list | YRANGE f1 f2 | ZRANGE f1 f2
}

```

Repeat **EXPOSURE** command up to 98 times.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Where:

j = wind load system type number (integer)

$p_1, p_2, p_3 \dots p_n$ = wind intensities (pressures) in force/area. Up to 100 different intensities can be defined in the input file per type.

$h_1, h_2, h_3 \dots h_n$ = corresponding heights in global vertical direction, measured in terms of actual Y (or Z for Z UP) coordinates up to which the corresponding intensities occur.

$e_1, e_2 \dots e_m$ = exposure factors. A value of 1.0 means that the wind force may be applied on the full influence area associated with the joint(s) if they are also exposed to the wind load direction. Limit: 99 factors.

joint-list = Joint list associated with Exposure Factor (joint numbers or "TO" or "BY") or enter only a group name.

f_1 and f_2 = global coordinate values to specify Y (or Z if Z UP) vertical range for Exposure Factor. Use **YRANGE** when Y is Up and **ZRANGE** when Z is Up (See the **SET Z UP** command in Section 5.5).

If the command **EXPOSURE** is not specified or if a joint is not listed in an Exposure, the exposure factor for those joints is chosen as 1.0.

Description

All intensities and heights are in current unit system. The heights specified are in terms of actual Y coordinate (or Z coordinates for Z UP) and not measured relative to the base of the structure. The first value of intensity (p_1) will be applied to any part of the structure for which the Y coordinate (or Z coordinate for Z UP) is equal to or less than h_1 . The second intensity (p_2) will be applied to any part of the structure that has vertical coordinates between the first two heights (h_1 and h_2) and so on. Any part of the structure that has vertical coordinates greater than h_n will be loaded with intensity p_n .

Only exposed surfaces bounded by members (not by plates or solids) will be used. The joint influence areas are computed based on surface member selection data entered in section 5.32.12 and based on the wind direction for a load case. Only joints actually exposed to the wind and connected to members will be loaded. The individual bounded areas must be planar surfaces, to a close tolerance, or they will not be loaded.

Exposure factor (e) is the fraction of the influence area associated with the joint (s) on which the load may act if it is also exposed to the wind load. Total load on a particular joint is calculated as follows.

$$\text{Joint load} = (\text{Exposure Factor}) \times (\text{Influence Area}) \times (\text{Wind Intensity})$$

The exposure factor may be specified by a joint-list or by giving a vertical range within which all joints will have the same exposure. If an exposure factor is not entered or not specified for a joint, then it defaults to 1.0 for those joints; in which case the entire influence area associated with the joint(s) will be considered.

For load generation on a closed type structure defined as a PLANE FRAME, influence area for each joint is calculated considering unit width perpendicular to the plane of the structure. You can accommodate the actual width by incorporating it in the Exposure Factor as follows.

Exposure Factor (User Specified) = (Fraction of influence area) X (influence width for joint)

Notes

- a. All intensities, heights and ranges must be provided in the current unit system.
- b. If necessary, the INTENSITY, EXPOSURE command lines can be continued on to additional lines by ending all but last line with a space and hyphen (-). Use up to 11 lines for a command.

Example

```

UNIT FEET
DEFINE WIND LOAD
TYPE 1
INTENSITY 0.1 0.15 HEIGHT 12 24
EXPOSURE 0.90 YRANGE 11 13
EXPOSURE 0.85 JOITN 17 20 22
LOAD 1 WIND LOAD IN X-DIRECTION
WIND LOAD X 1.2 TYPE 1

```

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

For additional examples, see section 5.32.12 and example 15 in the Examples manual.

The Intensity line can be continued in up to 12 lines.

So the following

```
INT 0.008 0.009 0.009 0.009 0.01 0.01 0.01 0.011 0.011 0.012  
0.012 0.012 HEIG 15 20 25 30 40 50 60 70 80 90 100 120
```

could be split as

```
INT 0.008 0.009 0.009 0.009 0.01 0.01 0.01 0.011 0.011 0.012  
0.012 0.012 -
```

```
HEIG 15 20 25 30 40 50 60 70 80 90 100 120
```

or

```
INT 0.008 0.009 0.009 0.009 0.01 0.01-  
0.01 0.011 0.011 0.012 0.012 0.012 HEIG 15 20 25 -  
30 40 50 60 70 80 90 100 120
```

etc.

Persistency of Parameters used to Generate ASCE Wind Loads

Note: This feature is effective STAAD.Pro 2006

In the graphical environment, under the General-Load-Definitions-Wind Definitions page, there is a facility to generate the pressure versus height table per the ASCE 7 wind load specifications per the 1995, 2002, or 2010 editions. The parameters which go into the derivation of this table are not retained by the graphical environment but rather added into the STAAD input file so they may be edited as needed. These values are not read by the STAAD engine directly and, therefore, are not directly processed as a load but are rather used to generate the wind intensity values which are used by the engine. An example of it is shown below.

Example

```
DEFINE WIND LOAD  
TYPE 1  
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
```

```

ASCE-7-2002:PARAMS 85.000 MPH 0 1 0 0 0.000 FT 0.000 FT
0.000 FT 1 -
1 40.000 FT 30.000 FT 25.000 FT 2.000 0.010 0 -
0 0 0 0.761 1.000 0.870 0.850 0 -
0 0 0 0.866 0.800 0.550
!> END GENERATED DATA BLOCK
INT 0.0111667 0.0111667 0.0113576 0.0115336 0.0116972
0.0118503 0.0119944 -
0.0121307 0.0122601 0.0123834 0.0125012 0.0126141
0.0127226 0.012827 0.0129277 -
HEIG 0 15 16.9231 18.8461 20.7692 22.6923 24.6154 26.5385
28.4615 -
30.3846 32.3077 34.2308 36.1538 38.0769 40

```

5.31.4 Definition of Time History Load

This set of commands may be used to define parameters for Time History loading on the structure. The time history data may be specified using either explicit definition, function specification, a spectrum specification, or time history data provided in an external file.

General Format

```

DEFINE TIME HISTORY (DT x)
TYPE i { ACCELERATION | FORCE | MOMENT } (SCALE f7) (SAVE)
{ t1 p1 t2 p2 ... tn pn | function-spec | spectrum-spec | READ
filename (f8) }

```

Repeat **TYPE** and Amplitude vs Time sets until all are entered, then:

```

ARRIVAL TIME
a1 a2 a3 ... an
{ DAMPING d | CDAMP | MDAMP }

```

The time history data can be explicitly defined using pairs of time and values of acceleration, force, or moment, where:

- **ACCELERATION** indicates that the time varying load type is a ground motion.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

- **FORCE** indicates that it is a forcing function.
- **MOMENT** indicates that it is a moment forcing function.

Table 5-20: Parameters for explicitly defined time history load

Variable or Command	Default Value	Description
x	-	Solution time step used in the step-by-step integration of the uncoupled equations. Values smaller than 0.00001 will be reset to the default DT value of 0.0013888 seconds.
i	-	Type number of time varying load (integer). Up to 136 types may be provided. This number should be sequential.
SCALE f ₇	1.0	The scale factor option multiplies all forces, accelerations, and amplitudes entered, read or generated within this Type. Primarily used to convert acceleration in g's to current units (9.80665, 386.08858, etc.).
SAVE	-	The save option results in the creation of two files (input file name with .TIM and .FRC file extensions). The .TIM file contains the history of the displacements of every node. The .FRC file contains the history of the 12 end forces of every member of the structure at every time step, and the 6 reactions at each support at every step. Syntax: TYPE 1 FORCE SAVE

Variable or Com- mand	Default Value	Description
t_1 p_1 t_2 p_2	-	<p>Values of time (in sec.) and corresponding force (current force unit) or acceleration (current length unit/sec²) depending on whether the time varying load is a forcing function or a ground motion. If the data is specified through the input file, up to 499 pairs can be provided for each type in the ascending value of time. More than one line may be used if necessary. However, if the data is provided through an external file, an unlimited number of time-force pairs may be specified.</p> <p>If the first point is not at zero time, then the forces before the first time (but after the arrival time) will be determined by extrapolation using the first two points entered. If the first point has a nonzero force, there will be a sudden application of that force over a single integration step (DT) at that time. Zero force will be assumed for all times after the last data point.</p>

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Variable or Command	Default Value	Description
$a_1 a_2 a_3 \dots a_n$	-	<p>Values of the various possible arrival times (seconds) of the various dynamic load types. Arrival time is the time at which a load type begins to act at a joint (forcing function) or at the base of the structure (ground motion). The same load type may have different arrival times for different joints and hence all those values must be specified here.</p> <p>The arrival times and the times from the time-force pairs will be added to get the times for a particular set of joints in the TIME LOAD data (see section 5.32.10.2). The arrival times and the time-force pairs for the load types are used to create the load vector needed for each time step of the analysis. Refer to Section 5.32.10.2 for information on input specification for application of the forcing function and/or ground motion loads. Up to 999 arrival time values may be specified.</p>

Variable or Command	Default Value	Description
d	0.05	<p>Modal damping ratio.</p> <p>To enter different damping for each mode, enter a DEFINE DAMP command elsewhere and MDAMP here; or for composite damping, enter CDAMP here and CDAMP ratios for all of the members / elements/ springs in the Constants and Set commands.</p> <p>The default value of 0.05 is used if no value is entered, if 0.0 is entered, or if less than 0.0000011 is entered. Use 0.0000011 to get exactly 0.0 for damping.</p>

The *function-spec* option can be used to specify harmonic loads. Both sine and cosine harmonic functions may be specified. The program will automatically calculate the harmonic load time history based on the following specifications.

For the function and amplitude option (:

function-spec =

{ SINE | COSINE }

AMPLITUDE f_0 { FREQUENCY | RPM } f_2 (PHASE f_3) CYCLES f_4 {
SUBDIV f_5 | STEP f_6 }

Note: Please be aware that if a Cosine function or Sine with nonzero phase angle is entered, the force at the arrival time will be nonzero; there will be a sudden application of force over a single integration step (**DT**) at that time.

Where:

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Table 5-21: Parameters for use with function time history load

Variable or Command	Default Value	Description
x	-	Solution time step used in the step-by-step integration of the uncoupled equations. Values smaller than 0.00001 will be reset to the default DT value of 0.0013888 seconds.
i	-	Type number of time varying load (integer). Up to 136 types may be provided. This number should be sequential.
SCALE f ₇	1.0	The scale factor option multiplies all forces, accelerations, and amplitudes entered, read or generated within this Type. Primarily used to convert acceleration in g's to current units (9.80665, 386.08858, etc.).

Variable or Command	Default Value	Description
SAVE	-	<p>The save option results in the creation of two files (input file name with .TIM and .FRC file extensions).</p> <p>The .TIM file contains the history of the displacements of every node.</p> <p>The .FRC file contains the history of the 12 end forces of every member of the structure at every time step, and the 6 reactions at each support at every step.</p> <p>Syntax: TYPE 1 FORCE SAVE</p>
f_0	-	Max. Amplitude of the forcing function in current units.
f_2	-	<p>If FREQUENCY, then cyclic frequency (cycles / sec.)</p> <p>If RPM, then revolutions per minute.</p>
f_3	0	Phase Angle in degrees.
f_4	-	No. of cycles of loading.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Variable or Command	Default Value	Description
f_5	3	<p>Used to subdivide a $\frac{1}{4}$ cycle into this many integer time steps.</p> <p>Note: Only used to digitize the forcing function. It is not the DT used to integrate for the responses. More subdivisions will make the digitized force curve more closely match a sine wave. The default is usually adequate.</p>

Variable or Command	Default Value	Description
f_6	(p/12)	<p>Time step of loading. Default is equal to one twelfth of the period corresponding to the frequency of the harmonic loading. (It is best to use the default)</p> <p>Note: Only used to digitize the forcing function. It is not the DT used to integrate for the responses. More subdivisions or smaller step size will make the digitized force curve more closely match a sine wave.</p>

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Variable or Command	Default Value	Description
$a_1 a_2 a_3 \dots a_n$	-	<p>Values of the various possible arrival times (seconds) of the various dynamic load types. Arrival time is the time at which a load type begins to act at a joint (forcing function) or at the base of the structure (ground motion). The same load type may have different arrival times for different joints and hence all those values must be specified here.</p> <p>The arrival times and the times from the time-force pairs will be added to get the times for a particular set of joints in the TIME LOAD data (see section 5.32.10.2). The arrival times and the time-force pairs for the load types are used to create the load vector needed for each time step of the analysis. Refer to Section 5.32.10.2 for information on input specification for application of the forcing function and/or ground motion loads. Up to 999 arrival time values may be specified.</p>

Variable or Command	Default Value	Description
d	0.05	<p>Modal damping ratio.</p> <p>To enter different damping for each mode, enter a DEFINE DAMP command elsewhere and MDAMP here; or for composite damping, enter CDAMP here and CDAMP ratios for all of the members / elements/ springs in the Constants and Set commands.</p> <p>The default value of 0.05 is used if no value is entered, if 0.0 is entered, or if less than 0.000001 is entered. Use 0.000001 to get exactly 0.0 for damping.</p>

The *spectrum-spec* option can be used to specify a synthetic ground motion acceleration time history based statistically on a user supplied acceleration spectrum.

The program will automatically calculate the acceleration time history based on the following specifications. Enter f_{12} , f_{13} , and f_{14} to indicate the rise, steady, & decay times, respectively.

For the spectrum option:

```

spectrum-spec =
SPECTRUM TMAX f9 DTI f10 DAMP f11 T1 f12 T2 f13 T3 f14 SEED f15
OPTIONS NF f16 NITR f17 ( THPRINT f18 ) ( SPRINT f19 ) ( FREQ
)

```

Starting on the next line, enter Spectra in the following input form:

$$P_1, V_1; P_2, V_2; \dots; P_n, V_n$$

Where:

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Table 5-22: Parameters used with the spectrum time history load

Variable or Command	Default Value	Description
x		Solution time step used in the step-by-step integration of the uncoupled equations. Values smaller than 0.00001 will be reset to the default DT value of 0.0013888 seconds.
i	-	Type number of time varying load (integer). Up to 136 types may be provided. This number should be sequential.
SCALE f ₇	1.0	The scale factor option multiplies all forces, accelerations, and amplitudes entered, read or generated within this Type. Primarily used to convert acceleration in g's to current units (9.80665, 386.08858, etc.).

Variable or Command	Default Value	Description
SAVE		<p>The save option results in the creation of two files (input file name with .TIM and .FRC file extensions).</p> <p>The .TIM file contains the history of the displacements of every node.</p> <p>The .FRC file contains the history of the 12 end forces of every member of the structure at every time step, and the 6 reactions at each support at every step.</p> <p>Syntax: TYPE 1 FORCE SAVE</p>
f_9	20 seconds	The Max. time (in seconds) in the generated time history.
f_{10}	0.2	Delta time step (in seconds) in the generated time history.
f_{11}	0.05	Damping ratio (5% is entered as 0.05) associated with the input spectrum.
f_{12}	4 seconds	Ending time of the acceleration rise time.
f_{13}	9 seconds	Ending time of the steady acceleration.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Variable or Command	Default Value	Description
f_{14}	14 seconds	Ending time of the acceleration decay.
f_{15}		Random Seed. Enter a positive integer (1 to 2147483647) to be used as a unique random number generation “seed”. A unique time history will be produced for each seed value. Change this value when you want to produce a “different (from the time history generated with the prior seed value)” but statistically equivalent time history. Omit this entry to get the default value (normal option).
f_{16}		The input shock spectrum will be re-digitized at NF equally spaced frequencies by interpolation. Default is the greater of 35 or the number of points in the input spectrum.
f_{17}	10	The number of iterations which will be used to perfect the computed time history.

Variable or Command	Default Value	Description
f_{18}	1	<p>Print the time history that is generated. Omit the THPRINT parameter to avoid printing.</p> <p>1 = print beginning 54 values and last 54 values</p> <p>2 = Print entire curve.</p> <p>>10 = print beginning f_{18} values and last f_{18} values</p>
f_{19}	1	<p>Print the spectrum generated from the time history that is generated. Omit the SPRINT parameter to avoid printing</p>
FREQ		<p>If entered, then frequency-spectra pairs are entered rather than period-spectra pairs.</p>

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Variable or Command	Default Value	Description
P_1, V_1 P_2, V_2 ... P_n, V_n		<p>Data is part of input, immediately following SPECTRUM command. Period (or frequency if FREQ option entered above)</p> <p>Value pairs (separated by semi colons) are entered to describe the Spectrum curve. Enter the period in seconds (or frequency in Hz.) and the corresponding Value is in acceleration (current length unit/sec²) units. Continue the curve data onto as many lines as needed (up to 999 spectrum pairs). Spectrum pairs must be in ascending or descending order of period (or frequency).</p> <p>Note: If data is in g acceleration units, then set SCALE to a conversion factor to the current length unit (9.807, 386.1, etc.). Also note, do not end these lines with a hyphen. Commas and</p>

Variable or Command	Default Value	Description
$a_1 a_2 a_3 \dots a_n$		<p>Values of the various possible arrival times (seconds) of the various dynamic load types. Arrival time is the time at which a load type begins to act at a joint (forcing function) or at the base of the structure (ground motion). The same load type may have different arrival times for different joints and hence all those values must be specified here.</p> <p>The arrival times and the times from the time-force pairs will be added to get the times for a particular set of joints in the TIME LOAD data (see section 5.32.10.2). The arrival times and the time-force pairs for the load types are used to create the load vector needed for each time step of the analysis. Refer to Section 5.32.10.2 for information on input specification for application of the forcing function and/or ground motion loads. Up to 999 arrival time values may be specified.</p>

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Variable or Command	Default Value	Description
d	0.05	<p>Modal damping ratio.</p> <p>To enter different damping for each mode, enter a DEFINE DAMP command elsewhere and MDAMP here; or for composite damping, enter CDAMP here and CDAMP ratios for all of the members / elements/ springs in the Constants and Set commands.</p> <p>The default value of 0.05 is used if no value is entered, if 0.0 is entered, or if less than 0.0000011 is entered. Use 0.0000011 to get exactly 0.0 for damping.</p>

Note: Please be aware that if a Cosine function or Sine with nonzero phase angle is entered, the force at the arrival time will be nonzero; there will be a sudden application of force over a single integration step (**DT**) at that time.

The time history data can also be defined in an external file, where:

Table 5-23: Parameters for a time history load defined in an external file

Variable or Command	Default Value	Description
x	-	Solution time step used in the step-by-step integration of the uncoupled equations. Values smaller than 0.00001 will be reset to the default DT value of 0.0013888 seconds.
i	-	Type number of time varying load (integer). Up to 136 types may be provided. This number should be sequential.
SCALE f ₇	1.0	The scale factor option multiplies all forces, accelerations, and amplitudes entered, read or generated within this Type. Primarily used to convert acceleration in g's to current units (9.80665, 386.08858, etc.).
SAVE	-	The save option results in the creation of two files (input file name with .TIM and .FRC file extensions). The .TIM file contains the history of the displacements of every node. The .FRC file contains the history of the 12 end forces of every member of the structure at every time step, and the 6 reactions at each support at every step. Syntax: TYPE 1 FORCE SAVE
filename		Filename for an external file containing time varying load history data.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

Variable or Command	Default Value	Description
f_8		The delta time spacing used for the external file data
$a_1 a_2 a_3 \dots a_n$		<p>Values of the various possible arrival times (seconds) of the various dynamic load types. Arrival time is the time at which a load type begins to act at a joint (forcing function) or at the base of the structure (ground motion). The same load type may have different arrival times for different joints and hence all those values must be specified here.</p> <p>The arrival times and the times from the time-force pairs will be added to get the times for a particular set of joints in the TIME LOAD data (see section 5.32.10.2). The arrival times and the time-force pairs for the load types are used to create the load vector needed for each time step of the analysis. Refer to Section 5.32.10.2 for information on input specification for application of the forcing function and/or ground motion loads. Up to 999 arrival time values may be specified.</p>

Variable or Com- mand	Default Value	Description
d	0.05	<p>Modal damping ratio.</p> <p>To enter different damping for each mode, enter a DEFINE DAMP command elsewhere and MDAMP here; or for composite damping, enter CDAMP here and CDAMP ratios for all of the members / elements/ springs in the Constants and Set commands.</p> <p>The default value of 0.05 is used if no value is entered, if 0.0 is entered, or if less than 0.000001 is entered. Use 0.000001 to get exactly 0.0 for damping.</p>

Examples

Using Force and Acceleration options:

```

UNIT ...
DEFINE TIME HISTORY
TYPE 1 FORCE
0.0 1.0 1.0 1.2 2.0 1.8 3.0 2.2
4.0 2.6 5.0 2.8
TYPE 2 ACCELERATION SCALE 9.80665
0.0 1.0 1.0 1.2 2.0 1.8 3.0 2.2
4.0 2.6 5.0 2.8
ARRIVAL TIME
0.0 1.0 1.8 2.2 3.5 4.4
DAMPING 0.075

```

Using the Spectrum option:

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

```
UNIT ...
DEFINE TIME HISTORY
TYPE 1 ACCELERATION SCALE 9.80665
SPECTRUM TMAX 19 DTI 0.01 DAMP 0.03
OPTIONS NF 40
0.03 1.00 ; 0.05 1.35
0.1 1.95 ; 0.2 2.80
0.5 2.80 ; 1.0 1.60
ARRIVAL TIME
0.0 1.0 1.8 2.2 3.5 4.4
DAMPING 0.075
```

Using the Harmonic Loading Generator:

```
UNIT ...
DEFINE TIME HISTORY
TYPE 1 FORCE
*FOLLOWING LINES FOR HARMONIC LOADING GENERATOR
FUNCTION SINE
AMPLITUDE 6.2831 FREQUENCY 60 CYCLES 100 STEP 0.02
ARRIVAL TIME
0.0
DAMPING 0.075
```

To define more than one sinusoidal load, the input specification is as follows:

```
DEFINE TIME HISTORY
TYPE 1 FORCE
FUNCTION SINE
AMPLITUDE 1.925 RPM 10794.0 CYCLES 1000
TYPE 2 FORCE
FUNCTION SINE
AMPLITUDE 1.511 RPM 9794.0 CYCLES 1000
```

```

TYPE 3 FORCE
FUNCTION SINE
AMPLITUDE 1.488 RPM 1785.0 CYCLES 1000
ARRIVAL TIME
0.0 0.0013897 0.0084034
DAMPING 0.04

```

The data in the external file must be provided as one or more time-force pairs per line as shown in the following example.

Data in Input file

```

UNIT ...
DEFINE TIME HISTORY
TYPE 1 FORCE
READ THFILE
ARRIVAL TIME
0.0
DAMPING 0.075

```

Data in the External file THFILE:

```

0.0 1.0 1.0 1.2
2.0 1.8
3.0 2.2
4.0 2.6

```

Notes

- a. By default the response (displacements, forces etc.) will contain the contribution of only those modes whose frequency is less than or equal to 108 cps. Use the **CUT OFF FREQUENCY** command to change this limit. Contributions of modes with frequency greater than the Cut Off Frequency are not considered.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

- b. Results are the individual maximums over the time period. Thus, derived quantities such as section forces and stresses, plate surface stresses and principal stresses should not be used.
- c. Results from harmonic input are the maximum over the time period including the start-up transient period. These results are not the steady-state results.
- d. By default, the results do not include the time period after the time loads end. Use the **CUT OFF TIME** command to lengthen (or shorten) the time period. If an intense short-term loading is used, the loading should be continued until after the expected peak response is reached.
- e. The **READ *filename*** command is to be provided only if the history of the time varying load is to be read from an external file. *filename* is the file name and may be up to 72 characters long. If the data on the file consists only of amplitudes, then enter f_8 as the delta time spacing.

5.31.5 Definition of Snow Load

This set of commands may be used to define some of the parameters for generation of snow loads on the structure. See section 5.32.13, Generation of Snow Loads, for the definition of additional parameters and the surfaces to be loaded.

General Format

DEFINE SNOW LOAD

TYPE f_1 PG f_2 CE f_3 CT f_4 IM f_5

Where:

f_1 - Type No: (limit of 100). The "Type No." is an integer value (1, 2, 3, etc.) which denotes a number by which the snow load type will be identified. Multiple snow load types can be created in the same model. Include as many types as needed.

f_2 - Ground Snow Load (Default = 0.0). The pressure or, weight per unit area, to be used for the calculation of the design snow load. Use a negative value to indicate loading acting towards the roof (upwards) as per section 7.2 of SEI/ASCE 7-02.

f_3 - Exposure Factor (Default = 1.0). Exposure factor as per Table 7-2 of the SEI/ASCE-7-02 code. It is dependent upon the type of exposure

of the roof (fully exposed/partially exposed/sheltered) and the terrain category, as defined in section 6.5.6 of the code.

f_4 - Thermal Factor (Default = 0.0). Thermal factor as per Table 7-3 of the SEI/ASCE-7-02 code. It is dependent upon the thermal condition.

f_5 - Importance Factor (Default = 1.0). Importance factor as per Table 7-4 of the SEI/ASCE-7-02 code. This value depends on the category the structure belongs to, as per section 1.5 and Table 1-1 of the code.

Example

```
START GROUP DEFINITION
FLOOR
  _ROOFSNOW 102 TO 153 159 160 TO 170 179 195 TO 197
END GROUP DEFINITION
UNIT FEET POUND
DEFINE SNOW LOAD
TYPE 1 PG 50 CE 0.7 CT 1.1 IM 1.1
```

5.31.6 Defining Reference Load Types

Large models can include multiple load cases which do not require analysis in their own right and are simply the building blocks for inclusion in primary load cases. Thus Reference Loads may be defined for this purpose. This is similar to a **REPEAT LOAD** command (See Section 5.32.11), but has the added benefit of not being solved in its own right.

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

Note: This feature requires STAAD.Pro 2007 Build 01 or higher.

See "Reference Load Cases - Application" on page 669 for a description of the procedure for specifying the reference load information in active load cases.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

General Format

```
DEFINE REFERENCE LOADS
LOAD R(i) LOADTYPE (type) TITLE load_title
(Load items)
...
END DEFINE REFERENCE LOADS
```

Example

```
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE DEAD TITLE REF DEAD
SELFWEIGHT Y -1
JOINT LOAD
4071 4083 4245 4257 FY -4.04
4090 FY -0.64
ELEMENT LOAD
378 TO 379 406 TO 410 422 TO 426 PR GY -1.44
MEMBER LOAD
5006 TO 5229 UNI GY -0.64
PMEMBER LOAD
1 TRAP GY -0.347 -0.254 35.5 42
LOAD R2 LOADTYPE LIVE TITLE REF LIVE
JOINT LOAD
4209 FY -6.63
4071 4083 4245 4257 FY -1.71
LOAD R3 LOADTYPE SNOW TITLE REF SNOW
JOINT LOAD
4109 FY -8.69
4071 4083 4245 4257 FY -3.29
LOAD R4 LOADTYPE SOIL TITLE REF SOIL
ELEMENT LOAD
```

```

1367 TO 1394 1396 1398 1522 1539 TO 1574 -
1575 TRAP JT -0.78 -0.78 -0.719167 -0.719167
LOAD R4 LOADTYPE MASS TITLE MASS MODEL
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
JOINT LOAD
17 TO 48 FY -2.5
49 TO 64 FY -1.25
END DEFINE REFERENCE LOADS

```

Mass Modeling Using Reference Loads

A reference load case of type **MASS** can be created which can then be used to define the structure mass used for all dynamic analyses (i.e., seismic, response spectrum, time history, etc.). Some analysis methods require you to create separate weight tables in the form of **SELFWEIGHT**, **MEMBER WEIGHT**, **JOINT WEIGHT**, etc. for each analysis, thus resulting in repetition of the same information. Using a **LOADTYPE MASS** reduces the repetitive data entry and the need for manually creating a weight table.

A mass model using this method is defined once and then used for all dynamic analyses.

Note: This feature requires STAAD.Pro V8i (SELECTseries 3) (release 20.07.08) or higher.

If the **LOADTYPE MASS** is missing and no mass is defined in the corresponding seismic or dynamic analysis load cases, the program will report error as mass is missing. If a mass model using reference load type **MASS** is defined and also a seismic weight table is defined within a dynamic load (i.e., static equivalent seismic load, time history, response spectrum, etc.) command, the program will simply add the seismic weight table with mass model already defined. This will increase the masses of the structure for equivalent seismic static analysis. The program will issue a warning message. Precaution should be taken so that mass model is not defined twice.

5.31.7 Definition of Direct Analysis Members

This set of commands may be used to define the members whose flexural stiffness or axial stiffness is considered to contribute to the lateral stability of the structure. Also the initial value of τ_b for each member can be set here. See Appendix 7 ANSI/AISC 360-05.

Note: This feature is available effective STAAD.Pro 2007 Build 03 and greater.

Members listed with **FLEX** will have their EI factored by 0.80 times τ_b while performing the global solution. The final member forces and code check will be with 100% of the flexural stiffness. Members listed with **AXIAL** will have their EA factored by 0.80 while performing the global solution. The final member forces and code check will be with 100% of the axial stiffness.

The **FYLD** value here will only be used for the calculation of τ_b .

General Format

DEFINE DIRECT

FLEX (f_1) *list-spec*

AXIAL *list-spec*

FYLD (f_2) *list-spec*

NOTIONAL LOAD FACTOR (f_3)

list-spec = * { **XR** f_4 f_5 | **YR** f_4 f_5 | **ZR** f_4 f_5 | **MEMBERS** *mem-list* | **LIST** *memb-list* | **ALL** }

Where:

f_1 = τ_b value. Default is 1.0. See Appendix 7 ANSI/AISC 360-05

f_2 = Yield Strength in current units. Default is 36.0 KSI.

f_3 = Notional Load Factor. Default is 0.002. If 0.003 or greater is entered, then τ_b will be set to 1.0 and no iterations will be performed.

f_4, f_5 = Upper and lower range values, respectively, used when specifying a range for **FLEX** or **FYLD**.

For specifying **NOTIONAL LOADS**, please see section 5.32.14. The notional loads and the factor used is specified entirely in the loading data.

Notes

- a. The **NOTIONAL LOAD FACTOR** f_3 command does *not* instruct the program to create Notional Loads. Instead, it is a control parameter used solely to make the following decision :
 - If f_3 is 0.002, the program calculates Tau-b on an iterative basis, as described in section 1.18.2.1.4 of this manual.
 - If f_3 is greater than 0.00299, the program sets the iteration limit to 1 and does not perform any additional iterations.
- b. τ_b is the value entered in the **FLEX** command. τ_b defaults to 1.0 if not entered.

5.31.8 Mass Model Using Reference Load

In STAAD.Pro, there is a facility to create mass model which will be used for all types of analysis, including seismic, response spectrum, time history, and any other dynamic analysis.

Note: This feature requires STAAD.Pro V8i (SELECTseries 3) (release 20.07.08) or higher.

When a rigid floor diaphragm is present in the model, the program follows the following logic:

- a. If a reference load type **MASS** is present, then the mass model is formed by combining all **MASS** reference loads.
- b. If reference load type **MASS** is not present, then the mass model is formed by combining all gravity reference loads. Warnings are displayed in the analysis output to alert you to this fact.
- c. If neither reference load types **MASS** nor gravity are present, then the mass model is formed by combining all **DEAD** reference loads. If any **LIVE** reference loads are present, then they will also be combined to this mass model. Warnings are displayed in the analysis output to alert you to this fact.

Mass model will be formed from Gravity or Dead/Live reference loads in case Mass reference load types are not present, only when rigid floor diaphragm is present in the model.

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

General Format

```
DEFINE REFERENCE LOADS  
LOAD Ri title  
...  
Load header and items  
...  
LOAD Rj LOADTYPE MASS title  
...  
Load header and items  
...  
END DEFINE REFERENCE LOADS  
DEFINE UBC/1893/... LOAD  
ZONE ...  
LOAD 1 title  
UBC/1893/... LOAD X f1  
LOAD 2 title  
UBC/1893/... LOAD Z f2  
LOAD 3 title  
SPECTRUM ...
```

The mass model is defined using the **LOADTYPE MASS** command. This mass model will be used for all types of analysis, including static equivalent, response spectrum, time history, and simply Eigen solution. If the load type **MASS** is missing and also no mass is defined in the corresponding seismic or dynamic analysis load cases, the program will report error as mass is missing

If a mass model using reference load type **MASS** is defined and also a seismic weight table is defined in **DEFINE UBC/1893/... DEFINITION**, the program will simply add seismic weight table with mass model already defined. This will increase the masses of the structure for equivalent seismic static analysis. The program will issue a warning message in the analysis output. Precaution should be taken so that the mass model is not defined twice. Similar warnings are issued in the cases of response spectrum and time history analysis.

Example

```

UNIT FEET KIP
DEFINE REF LOAD
LOAD R1 LOADTYPE MASS
* MASS MODEL
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
JOINT LOAD
17 TO 48 FX 2.5 FY 2.5 FZ 2.5
49 TO 64 FX 1.25 FY 1.25 FZ 1.25
*
LOAD R2
SELFWEIGHT Y -1
JOINT LOAD
17 TO 48 FY -2.5
49 TO 64 FY -1.25
END DEF REF LOAD
*****
***
DEFINE UBC LOAD
ZONE 0.38 I 1 RWX 5.6 RWZ 5.6 STYP 2 CT 0.032 NA 1.3 NV
1.6
*****
**
DEFINE TIME HISTORY
TYPE 1 FORCE
0.0 -0.0001 0.5 0.0449 1.0 0.2244 1.5 0.2244 2.0 0.6731
2.5 -0.6731
TYPE 2 ACCELERATION

```

Section 5 Commands and Input Instructions

5.31 Definition of Load Systems

```
0.0 0.001 0.5 -7.721 1.0 -38.61 1.5 -38.61 2.0 -115.82
2.5 115.82
ARRIVAL TIMES
0.0
DAMPING 0.05
*****
**
LOAD 1
UBC LOAD X 0.75
*****
**
LOAD 2
UBC LOAD Z 0.75
*****
**
LOAD 3
SPECTRUM CQC X 0.174075 ACC DAMP 0.05 SCALE 32.2
0.03 1.00 ; 0.05 1.35 ; 0.1 1.95 ; 0.2 2.80 ; 0.5 2.80 ;
1.0 1.60
*****
**
LOAD 4
SPECTRUM CQC Z 0.174075 ACC DAMP 0.05 SCALE 32.2
0.03 1.00 ; 0.05 1.35 ; 0.1 1.95 ; 0.2 2.80 ; 0.5 2.80 ;
1.0 1.60
*****
**
LOAD 5
TIME LOAD
53 57 37 41 21 25 FX 1 1
GROUND MOTION X 2 1
*****
**
```

5.32 Loading Specifications

This section describes the various loading options available in STAAD. The following command may be used to initiate a new load case.

General Format

```
LOADING  $i_1$  ( LOADTYPE  $a_1$  ) ( REDUCIBLE ) ( TITLE any_Load_title )
```

Where:

i_1 = any unique integer number (up to five digits) to identify the load case. This number need not be sequential with the previous load number.

a_1 = one of the following:

Dead	Rain Water/ Ice	Wind on Ice
Live	Ponding	Crane Hook
Roof Live	Dust	Mass
Wind	Traffic	Gravity
Seismic	Temperature	Push
Snow	Accidental	None
Fluids	Flood	
Soil	Ice	

The words **LOADTYPE a** are necessary only if you intend to use the Automatic Load Combination generation tool in the graphic interface. For details, refer to the User Interface Manual.

The keyword **REDUCIBLE** should be used only when the loadtype is **LIVE**. It instructs the program to reduce—according to the provisions of UBC 1997, IBC 2000, or IBC 2003 codes—a floor live load specified using **FLOOR LOAD** or **ONEWAY LOAD** commands. For details, see Section 5.32.4 of this manual.

Under this heading, all different loads related to this loading number can be input. These different kinds of loads are described in the remaining sub-sections below.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Notes

- a. For Mass Model Loading in Dynamics, it is strongly recommended that you read Section 1.18.3.2. For the purpose of entering the mass distribution for the first dynamic load case, use the following sections: 5.32.1 through 5.32.4 plus 5.32.9. A reference load can be used to create a mass model for all dynamic analyses.

The purpose of the mass modeling step is to create lumped masses at the joints that the eigensolution can use. The member/element loading is only a convenience in generating the joint masses. Analytically the masses are not in the elements but are lumped at the joints.

- b. The absolute value of joint loads or loads distributed to joints from member/element loadings will be treated as weights. The moments applied to member/elements or computed at joints as a result of member/element loadings will be ignored. Only moments (actually weight moment of inertia, force-length² units) applied in the Joint.
- c. The load command will be used in defining the weight moment of inertias at joints. For slave joint directions, the associated joint weight or weight moment of inertia will be moved to the master. In addition, the translational weights at slave joint directions will be multiplied by the square of the distance to the master to get the additional weight moment of inertia at the master. Cross product weight moment of inertias at the master will be ignored.

5.32.1 Joint Load Specification

This set of commands may be used to specify joint loads on the structure. For dynamic mass modeling see sections 5.32 and 1.18.3.

General Format

JOINT LOAD

```
joint-list *{ FX  $f_7$  | FY  $f_8$  | FZ  $f_9$  | MX  $f_{10}$  | MY  $f_{11}$  | MZ  $f_{12}$  }
```

Where:

f_7, f_8, f_9 = force values in the corresponding global direction (even at inclined support joints).

f_{10}, f_{11}, f_{12} = moment values in the corresponding global direction.

Example

```

UNIT FEET KIP
...
JOINT LOAD
3 TO 7 9 11 FY -17.2 MZ 180.0
5 8 FX 15.1
UNIT INCH KIP
12 MX 180.0 FZ 6.3

```

Notes

- Joint numbers may be repeated where loads are meant to be additive in the joint.
- A **UNIT** command may be on lines in between joint-list lines.
- If moments are for dynamic mass, then the units are assumed to be force-length².

5.32.2 Member Load Specification

This set of commands may be used to specify member loads on frame members

General Format

MEMBER LOAD

```

member-list { { UNI | UMOM } dir-spec f1 f2 f3 f4 | { CON |
CMOM } dir-spec f5 f6 f4 | LIN dir-spec f7 f8 f9 | TRAP dir-
spec f10 f11 f12 f13 }

```

Where:

dir-spec = { X | Y | Z | GX | GY | GZ | PX | PY | PZ }

X, Y, & Z specify the direction of the load in the local (member) x, y and z-axes.

GX, GY, & GZ specify the direction of the load in the global X, Y, and Z-axes.

Section 5 Commands and Input Instructions

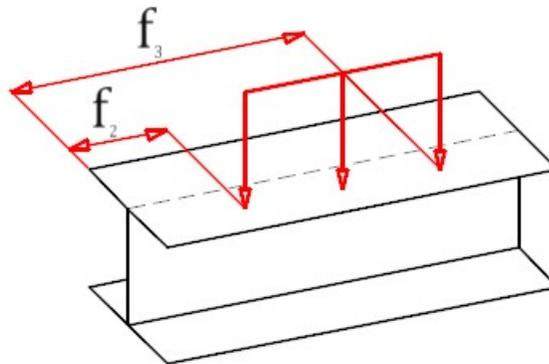
5.32 Loading Specifications

PX, PY and PZ may be used if the load is to be along the projected length of the member in the corresponding global direction.

Note: Load start and end distances are measured along the member length and *not* the projected length.

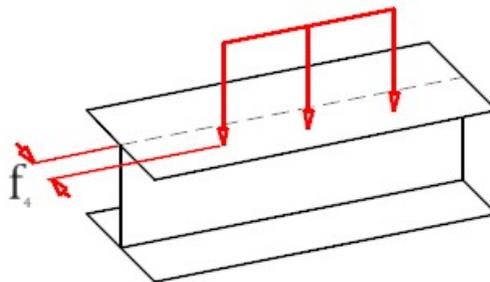
f_1 = value of uniformly distributed load (**UNI**) or moment (**UMOM**).

f_2, f_3 = distance of from the start of the member to the start of the load, and distance from the start of the member to the end of the load, respectively. The load is assumed to cover the full member length if f_2 and f_3 are omitted.



Note: Uniformly distributed moments can not be assigned to tapered members for analysis.

f_4 = Perpendicular distance from the member shear center to the local plane of loading. The value is positive in the general direction of the parallel (or close to parallel) local axis. If global or projected load is selected, then the local Y component of load is offset the f_4 distance; the local Z component is offset the f_4 distance; and the local X component is not offset.



Note: The local x component of force is not offset (i.e., no secondary moment is caused by axial load).

f_5 = value of concentrated force (**CON**) or moment (**CMOM**)

f_6 = distance of from the start of the member to concentrated force or moment. f_6 will default to half the member length if omitted.

f_7, f_8 = **LIN** specifies a linearly decreasing or increasing, or a triangular load. If the load is linearly increasing or decreasing then f_7 is the value at the start of the member and f_8 is the value at the end.⁷

f_9 = If the load is triangular, then f_7 and f_8 are input as zero and f_9 is the value of the load in the middle⁷ of the member.

f_{10}, f_{11} = The starting and ending load value for a trapezoidal linearly varying load (**TRAP**), respectively. The TRAP load may act over the full or partial length of a member and in a local, global or projected direction.

f_{12}, f_{13} = the loading starting point and stopping point, respectively. Both are measured from the start of the member. If f_{12} and f_{13} are not given, the load is assumed to cover the full member length.¹³

Notes

- In earlier versions of STAAD, the LINear type of member load could be applied only along the local axis of the member. It has been modified to allow for global and projected axes directions also.
- If the member being loaded has offset distances (see MEMBER OFFSET specification), the location of the load is measured not from the coordinates of the starting node but from the offset distance.
- Trapezoidal loads are converted into a uniform load and 8 or more concentrated loads.
- A UNIT command may be on lines in between member-list lines.
- If a load location is less than zero, it is reset to o.o.
- If a load location is greater than the length, it is reset to the length.

Example

```
MEMBER LOAD
619 CON GY -2.35 5.827
68 TO 72 UNI GX -0.088 3.17 10.0
186 TRAP GY -0.24 -0.35 0.0 7.96
3212 LIN X -5.431 -3.335
41016 UNI PZ -0.075
3724 LIN GY -6.2 -7.8
```

5.32.3 Element Load Specifications

This set of commands may be used to specify various types of loads on plate and solid elements.

5.32.3.1 Element Load Specification - Plates

This command may be used to specify various types of **ELEMENT LOADS** for plates. Plate element loads must be applied following the expression

ELEMENT LOAD (PLATE)

using the format explained under the following options.

Option 1

element-list {PRESSURE {GX | GY | GZ} p_1 (x_1 y_1 x_2 y_2) }

This is for specifying a pressure of magnitude p_1 in one of the global axis directions on the full element or a small rectangular part of an element. If applied on a small part, (x_1, y_1, x_2 and y_2) define the corners of the rectangular region where the load is applied. If only x_1, y_1 is provided, the load is assumed as a concentrated load applied at the specified point defined by (x_1, y_1). If (x_1, y_1, x_2, y_2) is not provided, the load is assumed to act over the full area of the element. (x_1, y_1, x_2 and y_2) are measured from the center of the element in the local axis system (see figure later in this section). There is no option to apply the load over a projected area.

p_1 has units of force per square of length for pressure and units of force for concentrated load.

GX, GY and GZ represent the global axis directions.

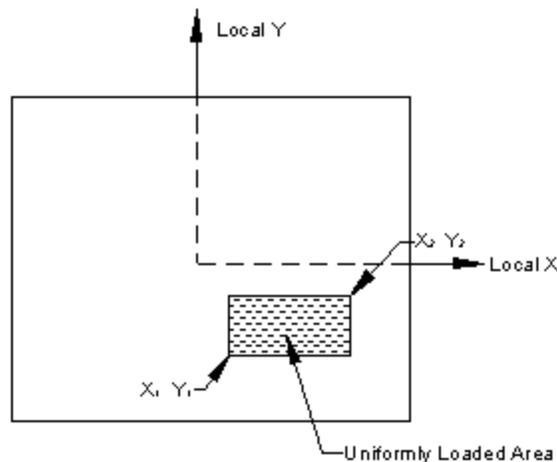
Option 2

element-list PRESSURE (Z) p_1 (x_1 y_1 x_2 y_2)

This is for specifying a constant pressure of magnitude p_1 acting perpendicular to the plane of the element on the full element or a small rectangular part of an element. This coincides with the element's local Z axis. If applied on a small part, (x_1, y_1, x_2 and y_2) define the corners of the rectangular region where the load is applied. If only x_1, y_1 is provided, the load is assumed as a concentrated load applied at the specified point defined by (x_1, y_1). If (x_1, y_1, x_2, y_2) is not provided, the load is assumed to act over the full area of the element. (x_1, y_1, x_2 and y_2) are measure from the center of the element in the local axis system (see figure later in this section). There is no option to apply the load over a projected area.

p_1 has units of force per square of length for pressure and units of force for concentrated load.

Figure 5-36: Coordinate values, x_1, y_1 & x_2, y_2 , in the local coordinate system used in options 1 and 2

**Option 3**

element-list PRESSURE {LX | LY} { p_2 }

This is for specifying a constant pressure of magnitude p_2 along the local X (LX) or Y (LY) axis of the element (parallel to the element surface). An example of this type of load is friction load. With this option, a load can be applied only on the full area of the element.

p_2 has units of force per square of length.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Option 4

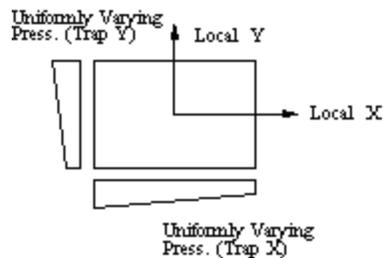
element-list TRAP {GX | GY | GZ | LX | LY} {X | Y} f1 f2

This is for applying a trapezoidally varying load with the following characteristics:

- The direction of action of the load is global (GX, GY or GZ), parallel to the surface (LX or LY as in friction type loads), or normal to the element surface (local Z). The last becomes the automatic direction of action if the global or tangential directions are not specified).
- The load varies along the local X or Y directions (imagine the wall of a tank with hydrostatic pressure where pressure at the lower nodes is higher than at the upper nodes, and hence the load varies as one travels from the bottom edge of the elements to the top edge.)

This type of load has to be applied over the full area of the element. f_1 is the intensity at the I-J (or J-K) edge, and f_2 is the intensity at the K-L (or L-I) edge depending on whether the load varies along "X" or "Y".

Figure 5-37: Center of the element is the origin defining the rectangular area on which the pressure is applied in option 4



The TRAP option should be used when a linearly varying pressure needs to be specified. The variation must be provided over the entire element.

X or Y - Direction of variation of element pressure. The TRAP X/Y option indicates that the variation of the Trapezoid is in the local X or in the local Y direction. The load acts in the global or local direction if selected, otherwise along the local Z axis.

f_1 - Pressure intensity at start.

f_2 - Pressure intensity at end.

Option 5

element-list TRAP {GX | GY | GZ | LX | LY} JT f3 f4 f5 f6

This is for specifying a trapezoidally varying load over the full area of the element where one happens to know the intensity at the joints (JT) of the element. The load is defined by intensities of f_3 , f_4 , f_5 and f_6 at the 4 corners of a 4-noded element. For triangular elements, f_6 is not applicable. The load can act along the global directions (GX, GY and GZ) or along the local X and Y directions (LX and LY, like a friction load).

Notes

- “Start” and “end” defined above are based on positive directions of the local X or Y axis.
- Pressure intensities at the joints allows linear variation of pressure in both the X and Y local directions simultaneously.
- The TRAP load with global directions may be used to apply a volumetric type of pressure. For example, consider a grain silo with a sloping wall. In the event of modeling it using non-uniform elements, by which we mean elements whose 3 or 4 nodes are all at different elevations, the grain height at each node will depend on the elevation of the node. One can apply the pressure by specifying the intensity at each node of each element.

Example

```
LOAD 4
ELEMENT LOAD
1 7 TO 10 PR 2.5
11 12 PR 2.5 1.5 2.5 5.5 4.5
15 TO 25 TRAP X 1.5 4.5
15 TO 20 TRAP GY JT 1.5 4.5 2.5 5.5
34 PR 5.0 2.5 2.5
35 TO 45 PR -2.5
15 25 TRAP GX Y 1.5 4.5
29 95 TRAP LX Y 3.7 8.7
```

5.32.3.2 Element Load Specification - Solids

The following types of loads can be assigned on the individual faces of solid elements:

Section 5 Commands and Input Instructions

5.32 Loading Specifications

- A uniform pressure
- A volumetric type of pressure on a face where the intensity at one node of the face can be different from that at another node on the same face.

An example of such a load is the weight of water on the sloping face of a dam. If the dam is modeled using solids, for the individual elements, the water height at the lower elevation nodes will be larger than those at the higher elevation nodes.

The syntax is as follows.

General Format

ELEMENT LOAD SOLIDS

elem-list FACE i_1 PRESSURE { GX | GY | GZ } f_1 f_2 f_3 f_4

Where:

f_1 f_2 f_3 f_4 = Pressure values at the joints for each 3 or 4 joint face defined. Only f_1 needs to be specified for uniform pressure. In any case the pressure is provided over the entire face.

i_1 = one of six face numbers to receive the pressure for the solids selected. See figure in section 1.6.2 for the following face definitions. Enter a pressure on all 4 joints even if the face is collapsed to 3 points.

Description

The first option above loads the solid by specifying one or more of the 6 faces to receive pressure.

The PRESSURE may be provided either in GLOBAL (GX, GY, GZ) directions or in local Z direction (normal to the element). If the GLOBAL direction is omitted, the applied loading is assumed to be normal to the face and a positive pressure is into the solid. The loads are proportional to the area, not the projected area.

Face Number	Surface Joints			
	f1	f2	f3	f4
1 front	Jt 1	Jt 4	Jt 3	Jt 2

Face Number	Surface Joints			
	f1	f2	f3	f4
2 bottom	Jt 1	Jt 2	Jt 6	Jt 5
3 left	Jt 1	Jt 5	Jt 8	Jt 4
4 top	Jt 4	Jt 8	Jt 7	Jt 3
5 right	Jt 2	Jt 3	Jt 7	Jt 6
6 back	Jt 5	Jt 6	Jt 7	Jt 8

Example

```
LOAD 4
ELEMENT LOAD SOLIDS
11 12 FACE 3 PR 2.5 1.5 2.5 5.5
```

5.32.3.3 Element Load Specification - Joints

This command may be used to specify various types of element like loads for joints. Three or four joints are specified that form a plane area; pressure is specified for that area; then STAAD computes the equivalent joint loads. This command may be used as an alternative or supplement for the Area Load, Floor Load, and the other Element Load commands.

The PRESSURE may be provided either in GLOBAL (GX, GY, GZ) directions or in local Z direction (normal to the element). If the GLOBAL direction is omitted, the applied loading is assumed to be in the local Z direction as if the joints defined a plate. The loads are proportional to the area, not the projected area.

General Format**ELEMENT LOAD JOINTS**

```
i1 (BY i2) i3 (BY i4) i5 (BY i6) i7 (BY i8) -
FACETS j1 PRESSURE { GX | GY | GZ } f1 f2 f3 f4
```

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Note: If this data is on more than one line, the hyphens must be within the joint data.

Where:

f_1, f_2, f_3, f_4 = Pressure values at the joints for each 3 or 4 joint facet defined. Only f_1 needs to be specified for uniform pressure. In any case the pressure is provided over the entire element.

j_1 = number of facets loaded.

i_1, i_3, i_5, i_7 = Joint numbers that define the first facet.

i_2, i_4, i_6, i_8 = each joint number is incremented by the BY value (1 if omitted).

Example

```
LOAD 4
ELEMENT LOAD JOINT
1 BY 1 2 BY 1 32 BY 1 31 BY 1 -
FACETS 5 PR GY 10 10 15 15
```

The above data is equivalent to the following:

```
LOAD 4
ELEMENT LOAD JOINT
1 2 32 31 FACETS 1 PRESSURE GY 10 10 15 15
2 3 33 32 FACETS 1 PRESSURE GY 10 10 15 15
3 4 34 33 FACETS 1 PRESSURE GY 10 10 15 15
4 5 35 34 FACETS 1 PRESSURE GY 10 10 15 15
5 6 36 35 FACETS 1 PRESSURE GY 10 10 15 15
```

So, the value following the word **FACETS** is like a counter for generation, indicating how many element faces the load command must be created for. Thus a value of 5 for facets means, a total of 5 imaginary element faces have been loaded.

BY is the value by which the individual corner node number is being incremented during the generation. In this example, the value is 1, which is same as the default. Instead, if it had been say,

```
1 BY 22 BY 2 32 BY 1 31 BY 1 -
FACETS 5 PRESSURE GY 10 10 15 15
```

we would have obtained

```
1 2 32 31 FACETS 1 PRESSURE GY 10 10 15 15
3 5 33 32 FACETS 1 PRESSURE GY 10 10 15 15
5 8 34 33 FACETS 1 PRESSURE GY 10 10 15 15
8 11 35 34 FACETS 1 PRESSURE GY 10 10 15 15
9 14 36 35 FACETS 1 PRESSURE GY 10 10 15 15
```

Notes

If a pressure or volumetric load is acting on a region or surface, and the entity which makes up the surface, like a slab, is not part of the structural model, one can apply the pressure load using this facility. The load is defined in terms of the pressure intensity at the 3 or 4 nodes which can be treated as the corners of the triangular or quadrilateral plane area which makes up the region. This command may be used as an alternative or supplement for the Area Load, Floor Load, Wind Load, and other pressure load situations.

In other words, the element pressure load can be applied along a global direction on any surface, without actually having elements to model that surface. Thus, for a sloping face of a building, if one wants to apply a wind pressure on the sloping face, one can do so by specifying the joints which make up the boundary of that face. Three or four joints are specified that form a plane area; pressure is specified for that area; then STAAD computes the equivalent joint loads.

5.32.3.3.1 Surface Loads Specification

The following loading options are available for surface entities:

General Format

```
LOAD n
SURFACE LOAD
```

Section 5 Commands and Input Instructions

5.32 Loading Specifications

surface-List* PRESSURE { GX | GY | GZ } *Load-spec

Where:

GX, GY, and GZ = Global X, Y and Z directions. If the direction is omitted, the load will act along the local Z axis of the surface.

load-spec is provided for the following types of surface loads:

- Uniform pressure across the entire surface

$$\mathbf{Load-spec} = w$$

Where:

w = Value of pressure. Negative value indicates load acts opposite to the direction of the axis specified.

- Pressure load on a partial area of the surface:

$$\mathbf{Load-spec} = w \ x_1 \ y_1 \ x_2 \ y_2$$

Where:

w = Value of pressure. Negative value indicates load acts opposite to the direction of the axis specified.

x_1, y_1 = Local X and Y coordinates of the corner nearest to the surface origin of the loaded region. Measured from the origin of the surface, in the local coordinate system of the surface.

x_2, y_2 = Local X and Y coordinates of the corner farthest from the surface origin of the loaded region. Measured from the origin of the surface, in the local coordinate system of the surface.

- Concentrated force anywhere on the surface:

$$\mathbf{Load-spec} = p \ x \ y$$

Where:

p = Value of the concentrated force. Negative sign indicates load acts opposite to the direction of the axis specified.

x, y = Local X and Y coordinates of the point of action of the force. Measured from the origin of the surface, in the local coordinate system of the surface.

Example

```

LOAD 1 UNIFORM PRESSURE ON FULL SURFACE
SURFACE LOAD
1 2 PRESS GX 0.002
3 GX -0.0025
*
LOAD 2 CONCENTRATED FORCE
SURFACE LOAD
10 12 PRE GZ 400 3.5 4.5
*
LOAD 3 PARTIAL AREA LOAD
SURFACE LOAD
23 25 PRE FY -250 4 4.3 8 9.5

```

The attributes associated with the surface element, and the sections of this manual where the information may be obtained, are listed below:

Attributes	Related Sections
Surfaces Incidences	5.13.3
Openings in surface	5.13.3
Local Coordinates system for surfaces	1.6.3
Specifying sections for stress/force output	5.13.3
Property for surfaces	5.21.2
Material constants	5.26.3

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Attributes	Related Sections
Surface loading	5.32.3.4
Stress/Force output printing	5.42
Shear Wall Design	3.8.2, 5.55

5.32.4 Area, One-way, and Floor Load Specifications

These commands may be used to specify loading over an area enclosed by beam members and the program will distribute the loading onto the perimeter beams as either a one way or two-way system. They are used mostly when the entity transmitting the load, such as a slab, is not part of the structural model. The **AREA LOAD** or **ONEWAY LOAD** may be used for modeling one-way distribution and the **FLOOR LOAD** may be used for modeling two-way distribution. There are three commands which should be used in the following way:

Area Load

The program will establish the direction of the shorter span and load the beams in that direction. This command is used for distributing a pressure load onto the beams that define a closed loop.

Floor Load

This command is used for distributing a pressure load onto all beams that define a closed loop assuming a two way distribution of load.

One-way

This may appear similar to the **AREA LOAD** command, but this command defines an area from which the program will search out closed loops of beams, similar to the **FLOOR LOAD** command and also has the option to define the direction of span. This command is an development of the principals defined in the **FLOOR LOAD** command, but the load is defined to span in a single direction.

Note: Floor Load and One-Way Load can be made reducible according to the UBC/IBC rules if included in a Live Load case which is specified as Reducible.

5.32.4.1 Area Load Specification

Used for distributing a pressure load onto the beams that define a closed loop. The program will establish the direction of the shorter span and load the beams in that direction.

Note: The **AREA LOAD** command has been deprecated in favor of the favor of the **ONEWAY LOAD** or **FLOOR LOAD** commands.

General Format

AREA LOAD

```
memb-list ALOAD f1 { GX | GY | GZ }
```

Where:

f₁ = The value of the area load (units of weight over square length unit). If Global direction is omitted, then this load acts along the positive local y-axis [for the members of a floor analysis, the local y direction will coincide with global vertical axis in most cases]. If Global direction is included, then the load acts in that direction. The magnitude of the loads calculated is the same as if the positive local y axis option was selected. (For detailed description, refer to Section 1.)

Note: Area load should not be specified on members declared as **MEMBER CABLE**, **MEMBER TRUSS** or **MEMBER TENSION**.

Example

```
AREA LOAD
2 4 TO 8 A LOAD - 0.250
12 16 ALOAD - .0500
```

Notes

- The structure has to be modeled in such a way that the specified global axis remains perpendicular to the floor plane(s).

Section 5 Commands and Input Instructions

5.32 Loading Specifications

- b. For the FLOOR LOAD specification, a two-way distribution of the load is considered. For the ONEWAY and AREA LOAD specification, a one-way action is considered. For ONE WAY loads, the program attempts to find the shorter direction within panels for load generation purposes. So, if any of the panels are square in shape, no load will be generated for those panels. For such panels, use the FLOOR LOAD type.
- c. The global horizontal direction options (GX and GZ) enables one to consider AREA LOADs, ONEWAY LOADs, and FLOOR LOADs for mass matrix for frequency calculations.

5.32.4.2 One-way Load Specification

Defines an area from which the program will search out closed loops of beams, similar to the FLOOR LOAD command and also has the option to define the direction of span. This command is a development of the principals defined in the FLOOR LOAD command, but the load is defined to span in a single direction (similar to AREA LOAD).

The One-way Load specification be applied to groups and can also use live load reduction per IBC or UBC codes.

General Format

ONEWAY LOAD

{ YRANGE f1 f2 ONELOAD f3 (XRA f4 f5 ZRA f6 f7) { GX | GY | GZ } (INCLINED) (TOWARDS f8)

or

YRANGE f1 f2 ONELOAD f3 (XRA f4 f5 ZRA f6 f7) { GX | GY | GZ } (INCLINED) (TOWARDS f8)

or

YRANGE f1 f2 ONELOAD f3 (XRA f4 f5 ZRA f6 f7) { GX | GY | GZ } (INCLINED) (TOWARDS f8)

or

_FloorGroupName ONELOAD f3 { GX | GY | GZ } (INCLINED) (TOWARDS f8)

Where:

f1 f2 = Global coordinate values to specify Y, X, or Z range. The load

will be calculated for all members lying in that global plane within the first specified global coordinate range.

f_3 = The value of the load (unit weight over square length unit). If the global direction is omitted, then this load acts parallel to the positive global Y if command begins with YRA and based on the area projected on a X-Z plane. Similarly, for commands beginning with XRA, the load acts parallel to the positive global X and based on the area projected on a Y-Z plane. Similarly, for commands beginning with ZRA, the load acts parallel to the positive global Z and based on the area projected on a X-Y plane.

$f_4 - f_7$ = Global coordinate values to define the corner points of the area on which the specified floor load (f_3) acts. If not specified, the floor load will be calculated for all members in all floors within the first specified global coordinate range.

GX,GY,GZ = If a Global direction is included, then the load is re-directed to act in the specified direction(s) with a magnitude of the loads which is based on the area projected on a plane as if the Global direction was omitted. The Global direction option is especially useful in mass definition.

FloorGroupName = See "Listing of Entities (Members / Elements / Joints, etc.) by Specifying Groups" on page 318 for the procedure for creating FLOORGROUPS. The member-list contained in this name will be the candidates that will receive the load generated from the floor pressure.

f_8 = Defines a member onto which the loading is directed and defines the span direction for the one way loading. If the **TOWARDS** option is not used, the program will default to distributing a one-way load to the longest side.

INCLINED - This option must be used when a **ONEWAY LOAD** is applied on a set of members that form a panel(s) which is inclined to the global XY, YZ, or ZX planes.

Hint: The **SET FLOOR LOAD TOLERANCE** command may be used to specify a tolerance for out-of-plane nodes to be included in a floor load. See "Set Command Specification" on page 284

Section 5 Commands and Input Instructions

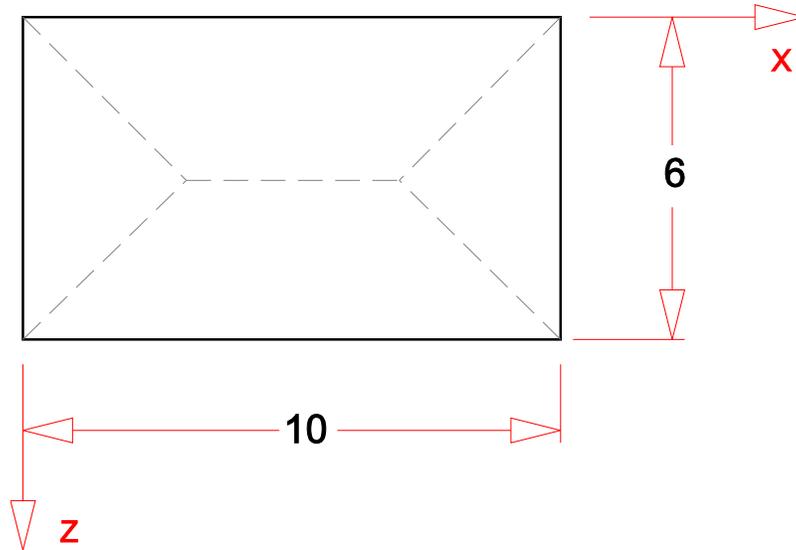
5.32 Loading Specifications

Notes

- a. The structure has to be modeled in such a way that the specified global axis remains perpendicular to the floor plane(s).
- b. For the **FLOOR LOAD** specification, a two-way distribution of the load is considered. For the **ONEWAY** and **AREA LOAD** specification, a one-way action is considered. For **ONE WAY** loads, the program attempts to find the shorter direction within panels for load generation purposes. So, if any of the panels are square in shape, no load will be generated for those panels. For such panels, use the **FLOOR LOAD** type.
- c. The load per unit area may not vary for a particular panel and it is assumed to be continuous and without holes.
- d. If the floor has a shape consisting of a mixture of convex and concave edges, then break up the floor load command into several parts, each for a certain region of the floor. This will force the program to localize the search for panels and the solution will be better. See illustrative example at the end of this section.
- e. At least one quadrilateral panel bounded on at least 3 sides by "complete" members has to be present within the bounds of the user-defined range of coordinates (**XRANGE**, **YRANGE** and **ZRANGE**) in order for the program to successfully generate member loads from the **FLOOR/ONEWAY LOAD** specification. A "complete" member is defined as one whose entire length between its start and end coordinates borders the specified panel.

The load distribution pattern depends upon the shape of the panel. If the panel is Rectangular, the distribution will be Trapezoidal and triangular as explained in the following diagram.

Figure 5-38: Trapezoidal and triangular load distribution for rectangular panels



For a panel that is not rectangular, the distribution is described in following diagram.

First, the CG of the polygon is calculated. Then, each corner is connected to the CG to form triangles as shown. For each triangle, a vertical line is drawn from the CG to the opposite side. If the point of intersection of the vertical line and the side falls outside the triangle, the area of that triangle will be calculated and an equivalent uniform distributed load will be applied on that side. Otherwise a triangular load will be applied on the side.

Live Load Reduction per UBC and IBC Codes

The UBC 1997, IBC 2000 and IBC 2003 codes permit reduction of floor live loads under certain situations. The provisions of these codes have been incorporated in the manner described further below.

To utilize this facility, the following conditions have to be met when creating the STAAD model.

- i. The live load must be applied using the **FLOOR LOAD** or **ONEWAY LOAD** option. This option is described earlier in this section of this manual, and

Section 5 Commands and Input Instructions

5.32 Loading Specifications

an example of its usage may be found in example problem 15 of the Examples manual.

- ii. As shown in section 5.32, the load case has to be assigned a Type called Live at the time of creation of that case. Additionally, the option called **Reducible**, also has to be specified as shown.

LOAD n LOADTYPE Live REDUCIBLE

Where:

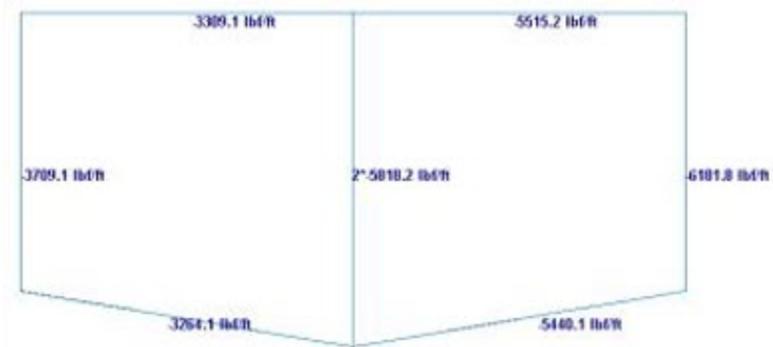
n is the load case number

The following figures show the load generated on members for the two situations.

Figure 5-39: Load generated for the previously described cases



i)



ii)

Table 5-24: Table - Details of the code implementation

Code Name	Section of code which has been implemented	Applicable Equations
UBC 1997	1607.5, page	Equation 7-1 R = $r(A-150)$ for FPS units R = $r(A-13,94)$ for SI units
IBC 2000	1607.9.2, page 302	Equation 16-2 R = $r(A-150)$ for FPS units R = $r(A-13,94)$ for SI units
IBC 2003	1607.9.2, page 277	Equation 16-22 R = $r(A-150)$ for FPS units R = $r(A-13,94)$ for SI units

In the above equations,

A = area of floor supported by the member

R = reduction in percentage

R = rate of reduction equal to 0.08 for floors.

Notes

- Only the rules for live load on **Floors** have been implemented. The rules for live load on **Roofs** have not been implemented.
- Since the medium of application of this method is the FLOOR LOAD or ONEWAY LOAD feature, and since STAAD performs load generation on beams only, the rules of the above-mentioned sections of the code for

Section 5 Commands and Input Instructions

5.32 Loading Specifications

vertical members (columns) has not been implemented. The distributed load on those members found to satisfy the requirements explained in the code would have a lowered value after the reduction is applied.

- c. Equation (7-2) of UBC 97, (16-3) of IBC 2000 and (16-23) of IBC 2003 have not been implemented.
- d. In the IBC 2000 and 2003 codes, the first note says “A reduction shall not be permitted in Group A occupancies.” In STAAD, there is no direct method for conveying to the program that the occupancy type is Group A. So, it is the user’s responsibility to ensure that when he/she decides to utilize the live load reduction feature, the structure satisfies this requirement. If it does not, then the reduction should not be applied. STAAD does not check this condition by itself.
- e. In the UBC 97 code, the last paragraph of section 1607.5 states that “The live load reduction shall not exceed 40 percent in garages for the storage of private pleasure cars having a capacity of not more than nine passengers per vehicle.” Again, there is no method to convey to STAAD that the structure is a garage for storing private pleasure cars. Hence, it is the user’s responsibility to ensure that the structure satisfies this requirement. If it does not, then the reduction should not be applied. STAAD does not check this condition by itself.
- f. Because all the three codes follow the same rules for reduction, no provision is made available in the command syntax for specifying the code name according to which the reduction is to be done.

5.32.4.3 Floor Load Specification

Used to distribute a pressure load onto all beams that define a closed loop assuming a two way distribution of load.

The Floor Load specification be applied to groups and can also use live load reduction per IBC or UBC codes.

General Format

FLOOR LOAD

```
{ YRANGE f1 f2 FLOAD f3 (XRA f4 f5 ZRA f6 f7) { GX  
| GY | GZ } (INCLINED)
```

or

YRANGE f1 f2 FLOAD f3 (XRA f4 f5 ZRA f6 f7) { GX | GY | GZ } (INCLINED)

or

YRANGE f1 f2 FLOAD f3 (XRA f4 f5 ZRA f6 f7) { GX | GY | GZ } (INCLINED)

or

_FloorGroupName FLOAD f3 { GX | GY | GZ } (INCLINED) }

Where:

$f_1 f_2$ = Global coordinate values to specify Y, X, or Z range. The load will be calculated for all members lying in that global plane within the first specified global coordinate range.

f_3 = The value of the load (unit weight over square length unit). If the global direction is omitted, then this load acts parallel to the positive global Y if command begins with YRA and based on the area projected on a X-Z plane. Similarly, for commands beginning with XRA, the load acts parallel to the positive global X and based on the area projected on a Y-Z plane. Similarly, for commands beginning with ZRA, the load acts parallel to the positive global Z and based on the area projected on a X-Y plane.

$f_4 - f_7$ = Global coordinate values to define the corner points of the area on which the specified floor load (f_3) acts. If not specified, the floor load will be calculated for all members in all floors within the first specified global coordinate range.

GX,GY,GZ = If a Global direction is included, then the load is re-directed to act in the specified direction(s) with a magnitude of the loads which is based on the area projected on a plane as if the Global direction was omitted. The Global direction option is especially useful in mass definition.

FloorGroupName = Please see section 5.16 of this manual for the procedure for creating FLOORGROUPS. The member-list contained in this name will be the candidates that will receive the load generated from the floor pressure.

INCLINED - This option must be used when a **FLOOR LOAD** is applied on a set of members that form a panel(s) which is inclined to the global XY, YZ, or ZX planes.

Section 5 Commands and Input Instructions

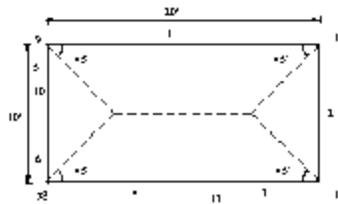
5.32 Loading Specifications

Hint: The **SET FLOOR LOAD TOLERANCE** command may be used to specify a tolerance for out-of-plane nodes to be included in a floor load. See "Set Command Specification" on page 284

Notes

- The structure has to be modeled in such a way that the specified global axis remains perpendicular to the floor plane(s).
- For the FLOOR LOAD specification, a two-way distribution of the load is considered. For the ONEWAY and AREA LOAD specification, a one-way action is considered. For ONE WAY loads, the program attempts to find the shorter direction within panels for load generation purposes. So, if any of the panels are square in shape, no load will be generated for those panels. For such panels, use the FLOOR LOAD type.
- FLOOR LOAD from a slab is distributed on the adjoining members as trapezoidal and triangular loads depending on the length of the sides as shown in the diagram. Internally, these loads are converted to multiple point loads.

Figure 5-40: Members 1 and 2 get full trapezoidal and triangular loads respectively. Members 3 and 4 get partial trapezoidal loads and 5 and 6 get partial triangular load.

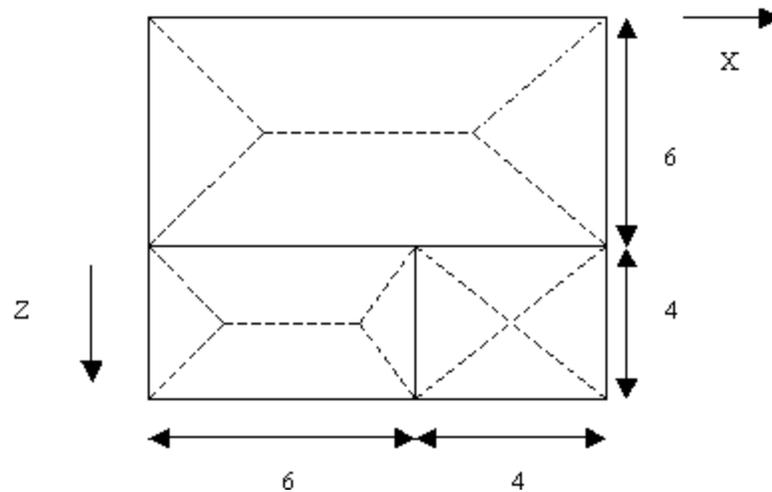


- The load per unit area may not vary for a particular panel and it is assumed to be continuous and without holes.
- The FLOOR LOAD facility is not available if the SET Z UP command is used (See Section 5.5.)
- If the floor has a shape consisting of a mixture of convex and concave edges, then break up the floor load command into several parts, each for a certain region of the floor. This will force the program to localize the search for panels and the solution will be better. See illustrative example at the end of this section.
- At least one quadrilateral panel bounded on at least 3 sides by "complete" members has to be present within the bounds of the user-defined range of

coordinates (XRANGE, YRANGE and ZRANGE) in order for the program to successfully generate member loads from the FLOOR/ONEWAY LOAD specification. A "complete" member is defined as one whose entire length between its start and end coordinates borders the specified panel.

The load distribution pattern depends upon the shape of the panel. If the panel is Rectangular, the distribution will be Trapezoidal and triangular as explained in the following diagram.

Figure 5-41: Trapezoidal and triangular loading patterns for rectangular panels



For a panel that is not rectangular, the distribution is described in following diagram.

First, the CG of the polygon is calculated. Then, each corner is connected to the CG to form triangles as shown. For each triangle, a vertical line is drawn from the CG to the opposite side. If the point of intersection of the vertical line and the side falls outside the triangle, the area of that triangle will be calculated and an equivalent uniform distributed load will be applied on that side. Otherwise a triangular load will be applied on the side.

Figure 5-42: Triangular and uniform dead load for a non-rectangular polygon


```
...
```

If in the above example, panel A has a load of 0.25 and panels B and C have a load of 0.5, then the input will be as follows:

Hint: Note the usage of XRANGE, YRANGE, and ZRANGE specifications.

```
...
LOAD 2
FLOOR LOAD
YRA 11.9 12.1 FLOAD -0.25 XRA 0.0 11.0 ZRA 0.0 16.0
YRA 11.9 12.1 FLOAD -0.5 XRA 11.0 21.0 ZRA 0.0 16.0
LOAD 3
...
```

The program internally identifies the panels (shown as A, B, and C in the figure). The floor loads are distributed as trapezoidal and triangular loads as shown by the dotted lines in the figure. The negative sign for the load signifies that it is applied in the downward global Y direction.

Illustration of Notes Item (f) for FLOOR LOAD

The attached example illustrates a case where the floor has to be subdivided into smaller regions for the floor load generation to yield proper results. The internal angle at node 6 between the sides 108 and 111 exceeds 180 degrees. A similar situation exists at node 7 also. As a result, the following command:

```
LOAD 1
FLOOR LOAD
YRANGE 11.9 12.1 FLOAD -0.35
```

will not yield acceptable results. Instead, the region should be subdivided as shown in the following example

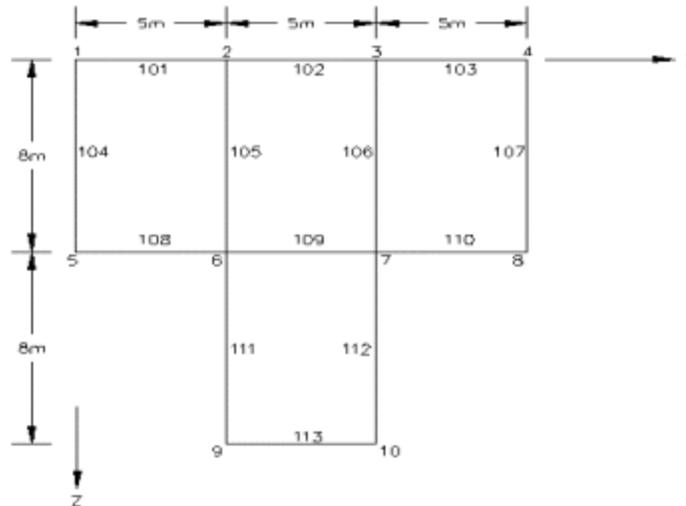
```
LOAD 1
FLOOR LOAD
```

Section 5 Commands and Input Instructions

5.32 Loading Specifications

```
YRANGE 11.9 12.1 FLOAD -0.35 XRA -.01 15.1 ZRA -0.1  
8.1  
YRANGE 11.9 12.1 FLOAD -0.35 XRA 4.9 10.1 ZRA 7.9  
16.1
```

Figure 5-44: Sub-divide floor area to avoid convex angles



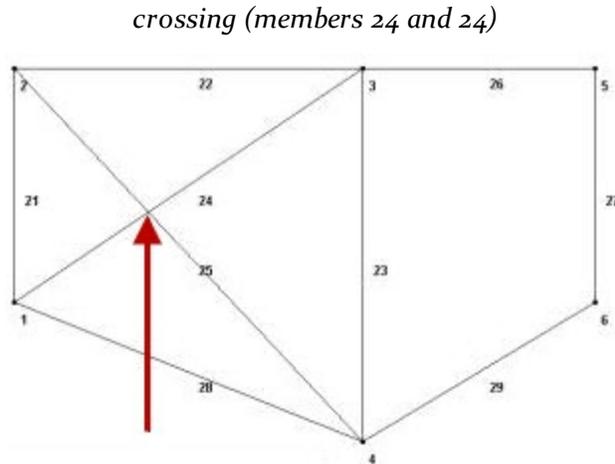
- h. The global horizontal direction options (GX and GZ) enables one to consider AREA LOADs, ONEWAY LOADSs and FLOOR LOADs for mass matrix for frequency calculations.
- i. For ONE WAY loads, the program attempts to find the shorter direction within panels for load generation purposes. So, if any of the panels are square in shape, no load will be generated on the members circumscribing those panels. In such cases, one ought to use the FLOOR LOAD type.

Applying FLOOR LOAD onto a Floor Group

When applying a floor load using X RANGE, Y RANGE and Z RANGE, there are two limitations that one may encounter:

- a. If panels consist of members whose longitudinal axis cross each other in an X type, and if the members are not connected to each other at the point of crossing, the panel identification and hence the load generation in that panel may fail. A typical such situation is shown in the plan drawing shown in the next figure.

Figure 5-45: Load generation on a panel with members not connected at a point of



- b. After the load is specified, if the user decides to change the geometry of the structure (X, Y or Z coordinates of the nodes of the regions over which the floor load is applied), she/he has to go back to the load and modify its data too, such as the XRANGE, YRANGE and ZRANGE values. In other words, the 2 sets of data are not automatically linked.

The above limitations may be overcome using a FLOOR GROUP. A GROUP name is a facility which enables us to cluster a set of entities – nodes, members, plates, solids, etc. into a single moniker through which one can address them. Details of this are available in section 5.16 of this manual.

The syntax of this command, as explained earlier in this section is:

FLOOR LOAD

***Floor-group-name* FLOAD f3 { GX | GY | GZ }**

Where:

f3 = pressure on the floor

To create equal loads in all 3 global directions for mass definition or other reasons, then enter direction labels for each direction desired; GY first then GX and/or GZ.

Example

```
START GROUP DEFINITION
FLOOR
_PNL5A 21 22 23 28
```

Section 5 Commands and Input Instructions

5.32 Loading Specifications

```
END GROUP DEFINITION
LOAD 2 FLOOR LOAD ON INTERMEDIATE PANEL @ Y = 10 FT
FLOOR LOAD
_PNL5A FLOAD -0.45 GY
_PNL5A FLOAD -0.45 GY GX GZ
```

```
LOAD 5 LOAD ON SLOPING ROOF
FLOOR LOAD
_SLOPINGROOF FLOAD -0.5 GY INCLINED
```

Live Load Reduction per UBC and IBC Codes

The UBC 1997, IBC 2000 and IBC 2003 codes permit reduction of floor live loads under certain situations. The provisions of these codes have been incorporated in the manner described further below.

To utilize this facility, the following conditions have to be met when creating the STAAD model.

1. The live load must be applied using the FLOOR LOAD or ONEWAY LOAD option. This option is described earlier in this section of this manual, and an example of its usage may be found in example problem 15 of the Examples manual.
2. As shown in section 5.32, the load case has to be assigned a Type called Live at the time of creation of that case. Additionally, the option called **Reducible**, also has to be specified as shown.

LOAD n LOADTYPE Live REDUCIBLE

Where:

n is the load case number

The following figures show the load generated on members for the two situations.

Figure 5-46: Distributed load on beams without live load reduction per IBC

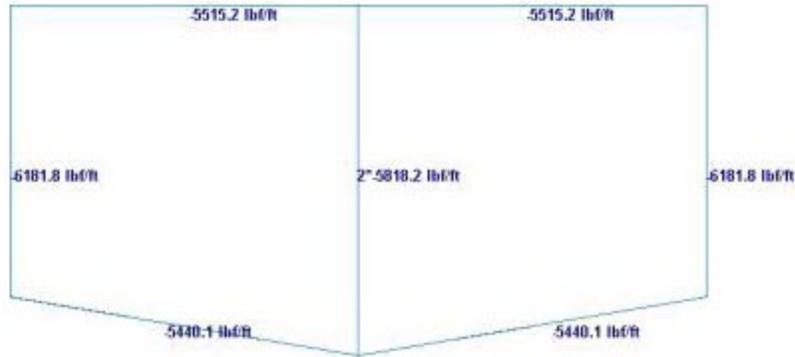


Figure 5-47: Distributed load on beams of a floor after live load reduction per IBC 2003

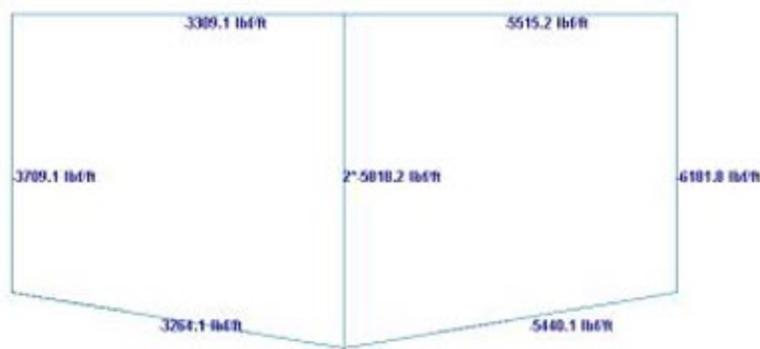


Table 5-25: Details of the code implementation

Code Name	Section of code which has been implemented	Applicable Equations
UBC 1997	1607.5, page	Equation 7-1 $R = r(A-150)$ for FPS units $R = r(A-13.94)$ for SI units
IBC 2000	1607.9.2, page 302	Equation 16-2 $R = r(A-150)$ for FPS units $R = r(A-13.94)$ for SI units

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Code Name	Section of code which has been implemented	Applicable Equations
IBC 2003	1607.9.2, page 277	Equation 16-22 $R = r(A-150)$ for FPS units $R = r(A-13.94)$ for SI units

In the above equations,

A = area of floor supported by the member

R = reduction in percentage

r = rate of reduction equal to 0.08 for floors.

Notes

- Only the rules for live load on **Floors** have been implemented. The rules for live load on **Roofs** have not been implemented.
- Since the medium of application of this method is the FLOOR LOAD or ONEWAY LOAD feature, and since STAAD performs load generation on beams only, the rules of the above-mentioned sections of the code for vertical members (columns) has not been implemented. The distributed load on those members found to satisfy the requirements explained in the code would have a lowered value after the reduction is applied.
- Equation (7-2) of UBC 97, (16-3) of IBC 2000 and (16-23) of IBC 2003 have not been implemented.
- In the IBC 2000 and 2003 codes, the first note says “A reduction shall not be permitted in Group A occupancies.” In STAAD, there is no direct method for conveying to the program that the occupancy type is Group A. So, it is the user’s responsibility to ensure that when he/she decides to utilize the live load reduction feature, the structure satisfies this requirement. If it does not, then the reduction should not be applied. STAAD does not check this condition by itself.

- e. In the UBC 97 code, the last paragraph of section 1607.5 states that “The live load reduction shall not exceed 40 percent in garages for the storage of private pleasure cars having a capacity of not more than nine passengers per vehicle.” Again, there is no method to convey to STAAD that the structure is a garage for storing private pleasure cars. Hence, it is the user’s responsibility to ensure that the structure satisfies this requirement. If it does not, then the reduction should not be applied. STAAD does not check this condition by itself.
- f. Because all the three codes follow the same rules for reduction, no provision is made available in the command syntax for specifying the code name according to which the reduction is to be done.

5.32.5 Prestress Load Specification

This command may be used to specify **PRESTRESS** loads on members of the structure.

General Format

MEMBER {PRESTRESS | POSTSTRESS } (LOAD)
memb-list **FORCE** f_1 *{ ES f_2 | EM f_3 | EE f_4 }

Where:

f_1 = Prestressing force. A positive value indicates precompression in the direction of the local x-axis. A negative value indicates pretension.

f_2 = specifies eccentricity (from the centroid) of the prestress force at the start of the member.

f_3 = specifies eccentricity (from the centroid) of the prestress force at the mid-point of the member.

f_4 = specifies eccentricity (from the centroid) of the prestress force at the end of the member.

Description

The first option, (**MEMBER PRESTRESS LOAD**), considers the effect of the prestressing force during its application. Thus, transverse shear generated at the ends of the member(s) subject to the prestressing force is transferred to the adjacent members.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

The second option, (**MEMBER POSTSTRESS LOAD**), considers the effect of the existing prestress load after the prestressing operation. Thus, transverse shear at the ends of the member(s) subject to the prestressing force is not transferred to the adjacent members.

Example

```
MEMBER PRESTRESS
2 TO 7 11 FORCE 50.0
MEMBER POSTSTRESS
8 FORCE 30.0 ES 3.0 EM -6.0 EE 3.0
```

In the first example, a prestressing force of 50 force units is applied through the centroid (i.e., no eccentricity) of members 2 to 7 and 11. In the second example, a poststressing force of 30 force units is applied with an eccentricity of 3 length units at the start, -6.0 at the middle, and 3.0 at the end of member 8.

One of the limitations in using this command is that under any one load case, on any given member, a prestress or poststress load may be applied only once. If the given member carries multiple stressed cables or has a **PRESTRESS** and **POSTSTRESS** load condition, such a situation will have to be specified through multiple load cases for that member. See example below.

Incorrect Input:

```
LOAD 1
MEMBER PRESTRESS
6 7 FORCE 100 ES 2 EM -3 EE 2
6 FORCE 150 ES 3 EM -6 EE 3
PERFORM ANALYSIS
```

Correct aInput:

```
LOAD 1 MEMBER PRESTRESS
6 7 FORCE 100 ES 2 EM -3 EE 2
LOAD 2
MEMBER PRESTRESS
6 FORCE 150 ES 3 EM -6 EE 3
```

```

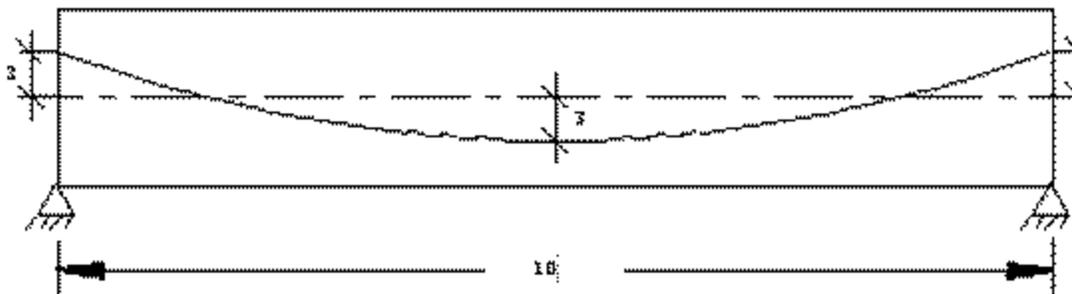
LOAD COMBINATION 3
1 1.0 2 1.0
PERFORM ANALYSIS

```

Examples for Modeling Techniques

The following examples describe the partial input data for the members and cable profiles shown below.

Figure 5-48: Example 1



```

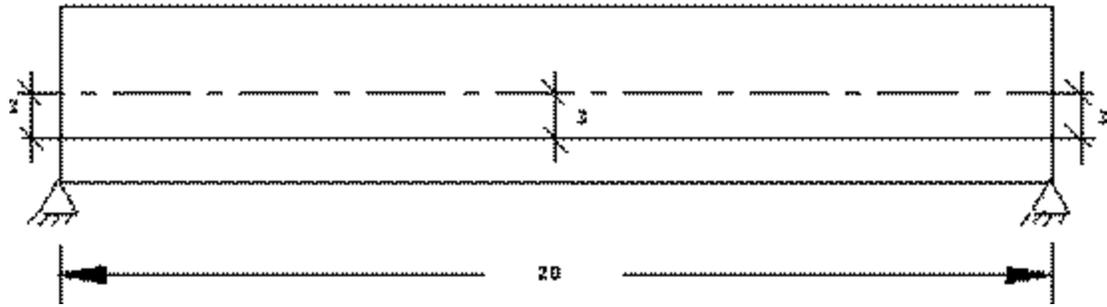
JOINT COORD
1 0 0 ; 2 10 0
MEMBER INCI
1 1 2
...
UNIT ...
LOAD 1
MEMBER POSTSTRESS
1 FORCE 100 ES 3 EM -3 EE 3
PERFORM ANALYSIS

```

Figure 5-49: Example 2

Section 5 Commands and Input Instructions

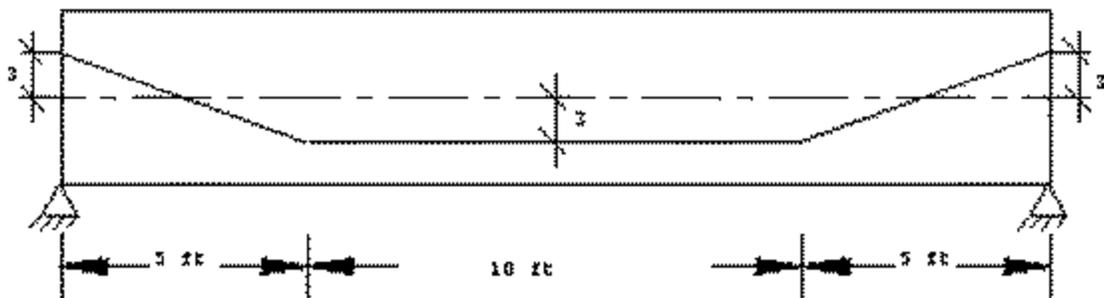
5.32 Loading Specifications



```

JOINT COORD
1 0 0 ; 2 20 0
MEMBER INCI
1 1 2
...
UNIT ...
LOAD 1
MEMBER PRESTRESS
1 FORCE 100 ES -3 EM -3 EE -3
PERFORM ANALYSIS
    
```

Figure 5-50: Example 3



```

JOINT COORD
1 0 0 ; 2 5 0 ; 3 15 0 0 ; 4 20 0
MEMBER INCI
1 1 2 ; 2 2 3 ; 3 3 4
...
UNIT ...
LOAD 1
    
```

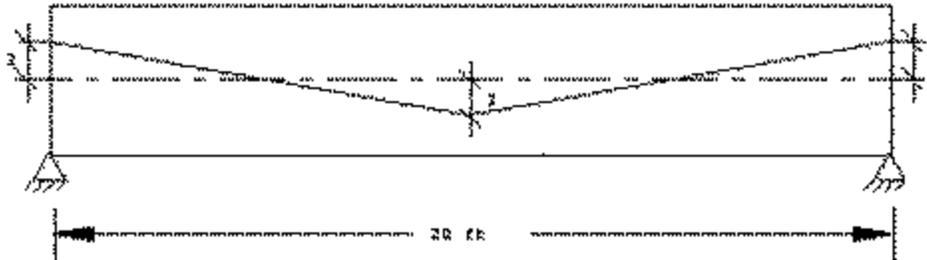
MEMBER PRESTRESS

```

1 FORCE 100 ES 3 EM 0 EE -3
2 FORCE 100 ES -3 EM -3 EE -3
3 FORCE 100 ES -3 EM 0 EE 3

```

PERFORM ANALYSIS

Figure 5-51: Example 4**JOINT COORD**

```

1 0 0 ; 2 10 0 ; 3 20 0 0

```

MEMBER INCI

```

1 1 2 ; 2 2 3

```

...

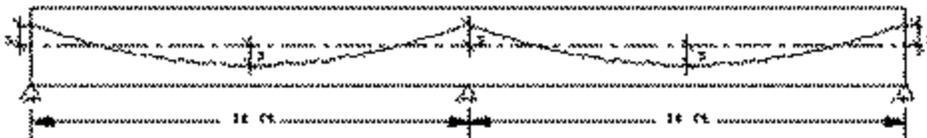
UNIT ...**LOAD 1****MEMBER PRESTRESS**

```

1 FORCE 100 ES 3 EM 0 EE -3
2 FORCE 100 ES -3 EM 0 EE 3

```

PERFORM ANALYSIS

Figure 5-52: Example 5**JOINT COORD**

```

1 0 0 ; 2 10 0 ; 3 20 0 0

```

Section 5 Commands and Input Instructions

5.32 Loading Specifications

```
MEMBER INCI
1 1 2 ; 2 2 3
...
UNIT ...
LOAD 1
MEMBER PRESTRESS
1 FORCE 100 ES 3 EM -3 EE 3
2 FORCE 100 ES 3 EM -3 EE 3
PERFORM ANALYSIS
```

5.32.6 Temperature Load Specification for Members, Plates, and Solids

This command may be used to specify **TEMPERATURE** loads or strain loads on members, plates, and solids; or strain loads on members.

General Format

TEMPERATURE LOAD

```
memb/elem-list { TEMP f1 f2 f4 | STRAIN f3 | STRAINRATE f5 }
```

Where:

f₁ = The change in temperature which will cause axial elongation in the members or uniform volume expansion in plates and solids. The temperature unit is the same as the unit chosen for the coefficient of thermal expansion ALPHA under the CONSTANT command.

(Members/Plates/Solids)

f₂ = The temperature differential from the top to the bottom of the member or plate ($T_{\text{top surface}} - T_{\text{bottom surface}}$). If f₂ is omitted, no bending will be considered. (Local Y axis) (Members/Plates). Section depth must be entered for prismatic.

f₄ = The temperature differential from side to side of the member. (Local Z axis) (Members). Section or flange width must be entered for prismatic.

f_3 = Initial axial elongation (+)/ shrinkage (-) in member due to misfit, etc. in length unit (Members only).

f_5 = Initial axial elongation (+)/ shrinkage (-) per unit length, of members only.

Example

```
UNIT MMS
TEMP LOAD
1 TO 9 15 17 TEMP 70.0
18 TO 23 TEMP 90.0 66.0
8 TO 13 STRAIN 3.0
15 27 STRAINRATE 0.4E-4
```

Notes

- It is not necessary or possible to specify the units for temperature or for **ALPHA**. The user must ensure that the value provided for ALPHA is consistent in terms of units with the value provided for the temperature load. (See "Specifying and Assigning Material Constants" on page 385).
If **ALPHA** was provided by a material name (**STEEL**, **CONCRETE**, **ALUMINUM**) then the temperature here must be in degree Fahrenheit units (if English length units were in effect when Alpha was defined) or degree Celsius units (if metric length units were in effect when Alpha was defined).

5.32.7 Fixed-End Load Specification

This command may be used to specify **FIXED END** loads on members (beams only) of the structure.

General Format

```
FIXED ( END ) LOAD  
memb-list FXLOAD  $f_1, f_2, \dots, f_{12}$ 
```

Where:

member_list = normal STAAD member list rules (TO and BY for

Section 5 Commands and Input Instructions

5.32 Loading Specifications

generation; and - to continue list to next line).

$f_1 \dots f_6$ = Force-x, shear-y, shear-z, torsion, moment-y, moment-z (all in local coordinates) at the start of the member.

$f_7 \dots f_{12}$ = Same as above except at the end of the member.

If less than 12 load values are entered, zero values will be added to the end. The loads may be extended to one additional line by ending the first load line with a - (hyphen).

These loads are given in the member local coordinate system and the directions are opposite to the actual load on the member.

5.32.8 Support Joint Displacement Specification

This command may be used to specify displacements (or generate loads to induce specified displacements) in supported directions (pinned, fixed, enforced, or spring).

General Format

SUPPORT DISPLACEMENT

support-joint-list { FX | FY | FZ | MX | MY | MZ } f_1

Where:

f_1 = Value of the corresponding displacement. For translational displacements, the unit is in the currently specified length unit, while for rotational displacements the unit is always in degrees.

FX, FY, FZ specify translational displacements in X, Y, and Z directions respectively. MX, MY, MZ specify rotational displacements in X, Y, and Z directions.

Description

There are two distinct modes of usage for this command. If any “Enforced” specifications were used in the Support command then the “Displacement mode” is used; otherwise the “Load mode” is used. Despite the name of this command, if displacements are specified in spring directions, the displacement is at the joint not at the grounded end of the support spring. Displacement cannot be specified in a direction that does not have a support or a spring.

1. **DISPLACEMENT MODE** - With this mode, the support joint displacement is modeled as an imposed joint displacement. The joint directions where displacement may be specified must be defined (same for all cases) in the **SUPPORT** command, see section 5.27.1. Any beam members, springs or finite elements will be considered in the analysis. Other loading, inclined supports, and master/slave are all considered. Any number of cases may have displacements entered. However, all cases will have zero displacements at the enforced directions if no displacement values are entered for that case. At inclined supports the displacement specification is assumed to be in the inclined direction. Displacements may not be specified at slave directions.

If some cases are to have spring supports and others enforced displacements at the same joint directions, then two **PERFORM ANALYSES** must be used with the **CHANGE** command in between. The first perform analysis could have the **SUPPORTS** with springs, no enforced directions, and with the load cases without displacements. The second perform analysis would then have **SUPPORTS** without springs but with enforced directions and the cases with displacements.

Displacement Mode Restrictions:

- The Support Displacement command may be entered only once per case.
 - Spring directions and Enforced directions may not both be specified at the same joint direction in the same Perform Analysis step.
2. **LOAD MODE** - With this mode, the support joint displacement is modeled as a load. Only beam members (not springs or finite elements) are considered in computing the joint load distribution necessary to cause the displacement. Other loading, inclined supports, and master/slave are also not considered. These unconsidered factors, if entered, will result in displacements other than those entered (results are superimposed). Only those cases with displacements entered will be affected.

Load Mode Restrictions:

- Support Displacements can be applied in up to four load cases only.
- The Support Displacement command may be entered only once per case.
- Finite elements should not be entered.
- Inclined supports must not be entered.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

- Spring supports are not considered in calculating the load so their use will lead to displacements different from the input values.

Example

```
UNIT ...  
SUPPORT DISPL  
5 TO 11 13 FY -0.25  
19 21 TO 25 MX 15.0
```

In this example, the joints of the first support list will be displaced by 0.25 units in the negative global Y direction. The joints of the second support list will be rotated by 15 degrees about the global X-axis.

5.32.9 Selfweight Load

Used to calculate and apply the self weight of structural elements for analysis.

This command may be used to calculate and apply the **SELFWEIGHT** of the structure for analysis.

General Format

```
SELFWEIGHT ( { X | Y | Z }  $f_1$ ) ( LIST member-list | ALL | _  
groupname )
```

Where:

f_1

The factor to be used to multiply the selfweight. The default value is -1.0.

X, Y, & Z

represent the global direction in which the selfweight acts. The default direction is along the Y axis.

member-list

member, plate and/or solid list or group. All list specifications such as explicit list including **TO** and **BY**, range (**XR**, **YR**, **ZR**), parallel to (X, Y, Z) and groups are supported. If a group is used, the

selfweight command must be repeated for each group. If list is not provided, all structural components are used for body weight calculation unless those are inactive either by **INACTIVE** command specification or internally set due to TENSION/COMPRESSION ONLY specification.

Description

This command is used if the self-weight of the members, plates, and solids within the structure is to be considered. The self-weight of every active element is calculated and applied as a uniformly distributed member load.

Note: Surfaces are not included in the selfweight command. The **SSELFWT** must be used to include the selfweight for surface elements. See "Surface Selfweight Load" on page 585

This command may also be used without any direction and factor specification. Thus, if specified as **SELFWEIGHT**, loads will be applied in the negative global Y direction with a factor of unity.

Notes

- a. Density must be provided for calculation of the self weight.
- b. The selfweight of finite elements is converted to joint loads at the connected nodes and is not used as an element pressure load.
- c. The selfweight of a plate is placed at the joints, regardless of plate releases.
- d. The **SELFWEIGHT** specification for definition of static method of seismic load generator also is capable of accepting a list. Only there is no direction specification for this command.
- e. Similarly, the **SELFWEIGHT** specification when used in a Reference Load is also capable of handling a list.

Example

```
LOAD 1 DEAD AND LIVE LOAD
* FACTORED WEIGHT FOR MEMBERS, PLATES, AND SOLIDS 4
  THROUGH 10 ALONG GLOBAL X DIRECTION
  SELF WEIGHT X 1.4 LIST 4 TO 10
```

Section 5 Commands and Input Instructions

5.32 Loading Specifications

```
* INCLUDES WEIGHT OF AL MEMBERS, PLATES, AND SOLIDS
ASSOCIATED WITH THE GROUP _PLATEGRP1
SELF X 1.0 _PLATEGRP1
* INCLUDES THE SELFWEIGHT OF ALL MEMBERS, PLATES, AND
SOLIDS
SELF
SELF WEIGHT Y -1.0 ALL
* MISC OTHER EXAMPLES
SELF Y -1.4
SELF X 1.4 ALL
SELF WEIGHT Z 1.4 _SOLIDGRP1
SELF Y -1.4 YR 10.5 10.51
SELF Y -1.4 X
```

5.32.9.0.1 Selfweight Load

Used to calculate and apply the self weight of members, plates, and solids in the structure for analysis.

General Format

SELFWEIGHT ({ X | Y | Z } f_1) (LIST *member-list* | ALL)

Where:

f_1

The factor to be used to multiply the selfweight. The default value is -1.0.

X, Y, & Z

represent the global direction in which the selfweight acts. The default direction is along the Y axis.

member-list

member, plate and/or solid list or group. All list specifications such as explicit list including **TO** and **BY**, range (**XR**, **YR**, **ZR**), parallel to (X, Y, Z) and groups are supported. If a group is used, the

selfweight command must be repeated for each group. If list is not provided, all structural components are used for body weight calculation unless those are inactive either by **INACTIVE** command specification or internally set due to TENSION/COMPRESSION ONLY specification.

Example

```
LOAD 1 DEAD AND LIVE LOAD
* INCLUDES THE SELFWEIGHT OF ALL MEMBERS, PLATES, AND
SOLIDS. THE FOLLOWING TWO COMMANDS ARE FUNCTIONALLY
IDENTICAL
SELF
SELF WEIGHT Y -1.0 ALL
* FACTORED WEIGHT FOR MEMBERS, PLATES, AND SOLIDS 4
THROUGH 10 ALONG GLOBAL X DIRECTION
SELF WEIGHT X 1.4 LIST 4 TO 10
* INCLUDES WEIGHT OF ALL MEMBERS, PLATES, AND SOLIDS
ASSOCIATED WITH THE GROUP _PLATEGRP1
SELF X 1.0 _PLATEGRP1
```

5.32.9.0.2 Surface Selfweight Load

Used to calculate and apply the self weight of surface elements in the structure for analysis.

Note: This feature requires STAAD.Pro V8i (SELECTseries 3) or higher.

General Format

SSELFWEIGHT ({ X | Y | Z } f_1) (LIST *surface-list* | ALL)

Where:

f_1

Section 5 Commands and Input Instructions

5.32 Loading Specifications

The factor to be used to multiply the selfweight. The default value is -1.0.

X, Y, & Z

represent the global direction in which the selfweight acts. The default direction is along the Y axis.

surface-list

surface element list or group. All list specifications such as explicit list including **TO** and **BY**, range (**XR, YR, ZR**), parallel to (X, Y, Z) and groups are supported. If a group is used, the self-weight command must be repeated for each group. If list is not provided, all structural components are used for body weight calculation unless those are inactive either by **INACTIVE** command specification or internally set due to **TENSION/COMPRESSION ONLY** specification.

See "Selfweight Load" on page 584 for Description and Notes pertaining to self weight.

Example

```
LOAD 1 LOADTYPE NONE TITLE DEAD LOAD
SURFACE LOAD
1 PR 1 0 0
SSELFWEIGHT Y -1
```

5.32.10 Dynamic Loading Specification

The command specification needed to perform response spectrum analysis and time-history analysis is explained in the following sections.

5.32.10.1 Response Spectrum Analysis

Various methods for performing response spectrum analysis have been implemented in STAAD. They include a generic method that is described in most text books, as well as code based methods like those required by the Eurocode 8, IS 1893, etc. These are described in the following sections.

5.32.10.1.1 Response Spectrum Specification - Generic Method

This command may be used to specify and apply the **RESPONSE SPECTRUM** loading for dynamic analysis.

This command should appear as part of a loading specification. If it is the first occurrence, it should be accompanied by the load data to be used for frequency and mode shape calculations. Additional occurrences need no additional information. The maximum number of response spectrum load cases allowed in one run is 50.

Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement. For dynamic mass modeling, see sections 5.32 and 1.18.3. An illustration of mass modeling is available, with explanatory comments, in Example Problem No.11.

General Format

```
SPECTRUM { SRSS | ABS | CQC | ASCE | TEN | CSM | GRP } *{ X
f1 | Y f2 | Z f3 } { ACC | DIS } (SCALE f4)
{DAMP f5 | CDAMP | MDAMP } ( {LIN | LOG} ) (MIS f6) (ZPA f7)
(FF1 f8) (FF2 f9) ( { DOMINANT f10 | SIGN } ) (SAVE) (IMR
f11) (STARTCASE f12)
```

Note: The data from SPECTRUM through SCALE above must be on the first line of the command, the remaining data can be on the first or subsequent lines with all but last ending with a hyphen (limit of four lines per spectrum).

Starting on the next line, enter Spectra in one of these two input forms:

```
{ p1 v1; p2 v2; p3 v3; ... | FILE filename }
```

Optionally enter a set of commands for individual modal response generation. See the section on Individual Modal Response Case Generation below for this command.

```
GENERATE INDIVIDUAL MODAL RESPONSE CASES
MODE LIST
CASE LIST
{ REPEAT | COMBINATION }
```

Section 5 Commands and Input Instructions

5.32 Loading Specifications

END

Where:

Table 5-26: Parameters used for generic model response spectrum

Parameter	Default Value	Description
f ₁ , f ₂ , f ₃	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.
f ₄		Linear scale factor by which the spectra data will be multiplied. Usually used to factor g's to length/sec ² units. This input is the appropriate value of acceleration due to gravity in the current unit system (thus, 9.81 m/s ² or 32.2 ft/s ²).
f ₅	0.05	Damping ratio for all modes when DAMP is used. Default value is 0.05 (5% damping if 0 or blank entered).

Parameter	Default Value	Description
f6		<p>Use Missing Mass method to include the static effect of the masses not represented in the modes. The spectral acceleration length/sec² for this missing mass mode is the f6 value entered in length per second squared units (this value is not multiplied by SCALE). If f6 is zero, then the spectral acceleration at the ZPA f7 frequency is used. If f7 is zero or not entered, then the spectral acceleration at 33Hz is used. The results of this calculation are SRSSed with the modal combination results.</p> <p>For SRSS, CQC, and TEN the results of this calculation are SRSSed with the modal combination results. For ABS, missing mass is ignored. For ASCE, the missing mass result is algebraically added with the rigid parts of the extracted modes. For ASCE, the MIS option is assumed to be on. If any of f6, f7, f8, or f9 are not entered, the defaults will be used. Missing mass does not include the effect of masses lumped at the supports unless the support is a stiff spring or an Enforced support.</p> <p>Note: If the MIS parameter is entered on any spectrum case it will be used for all spectrum cases.</p>

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Parameter	Default Value	Description
f7	33 [Hz]	For use with MIS option only. Defaults to 33 Hz if not entered. Value is printed but not used if MIS f6 is entered.
f8	2 [Hz]	The f1 parameter defined in the ASCE 4-98 standard in Hz units. For ASCE option only.
f9	33 [Hz]	The f2 parameter defined in the ASCE 4-98 standard in Hz units. For ASCE option only.
fio	1 (1st Mode)	<p>Dominant mode method. All results will have the same sign as mode number fio alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero).</p> <p>Note: Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode.</p>

Parameter	Default Value	Description
f11	1	The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through f11 will be used. Missing Mass modes are not output.
f12	Highest Load Case No. + 1	The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If f12 is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case no. plus one.

SRSS , **ABS** , **CQC** , **ASCE4-98** , **CSM** , **GRP** , & **TEN** Percent are methods of combining the responses from each mode into a total response. The **CQC&ASCE4** methods require damping, **ABS**, **SRSS**, **CRM**, **GRP**, and **TEN** do not use damping unless Spectra-Period curves are made a function of damping (see File option below). **CQC**, **ASCE**, **CRM**, **GRP**, and **TEN** include the effect of response magnification due to closely spaced modal frequencies. **ASCE** includes more algebraic summation of higher modes. **ASCE** & **CQC** are more sophisticated and realistic methods and are recommended.

SRSS

Square Root of Summation of Squares method.

CQC

Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of **SRSS**.

ASCE

Section 5 Commands and Input Instructions

5.32 Loading Specifications

NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.

ABS

Absolute sum. This method is very conservative and represents a worst case combination.

CSM

Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.

GRP

Closely Spaced Modes Grouping Method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

TEN

Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (Rev. 1.2.2, 1976).

ACC or **DIS** indicates whether Acceleration or Displacement spectra will be entered.

DAMP, **MDAMP**, and **CDAMP** Select source of damping input:

- **DAMP** indicates to use the f_2 value for all modes.
- **MDAMP** indicates to use the damping entered or computed with the **DEFINE DAMP** command if entered, otherwise default value of 0.05 will be used.
- **CDAMP** indicates to use the composite damping of the structure calculated for each mode. One must specify damping for different materials under the **CONSTANT** specification.

LIN or **LOG**: Select linear or logarithmic interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. **LIN** is default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

For **LIN** or **LOG**, when **FILE filename** is entered, the interpolation along the damping axis will be linear. The last **LIN-LOG** parameter entered on all of the spectrum cases will be used for all spectrum cases.

SAVE : This option results in the creation of a file (file name with .ACC extension) containing the joint accelerations in g's and radians/sec²

SIGN : This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for **ABS** option.

Warning: Do not enter **DOMINANT** parameter with this option.

Spectra data is input in one of these two input forms:

1. **p1, v1; p2, v2; ... ; pn, vn.** Data is part of input, immediately following SPECTRUM command. Period – Value pairs (separated by semi colons) are entered to describe the Spectrum curve. Period is in seconds and the corresponding Value is either acceleration (current length unit/sec²) or displacement (current length unit) depending on the ACC or DIS chosen. Continue the curve data onto as many lines as needed (up to 500 spectrum pairs). Spectrum pairs must be in ascending order of period. Note, if data is in g acceleration units, then set SCALE to a conversion factor to the current length unit (9.81, 386.4, etc.). Also note, do not end these lines with a hyphen (-). Each SPECTRUM command must be followed by Spectra data if this input form is used.
2. **FILE filename** data is in a separate file.

When the **File filename** command has been provided, then you must have the spectra curve data on a file named FILENAME prior to starting the analysis. The format of the FILE spectra data allows spectra as a function of damping as well as period.

Note: If the FILE fn command is entered, it must be with the first spectrum case and will be used for all spectrum cases.

No **File filename** command needs to be entered with the remaining spectrum cases. The format is shown below. FILENAME may not be more than 72 characters in length.

Example

```
LOAD 2 SPECTRUM IN X-DIRECTION
SELFWEIGHT X 1.0
```

Section 5 Commands and Input Instructions

5.32 Loading Specifications

```
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
SPECTRUM SRSS X 1.0 ACC SCALE 32.2
0.20 0.2 ; 0.40 0.25 ; 0.60 0.35 ; 0.80 0.43 ; 1.0 0.47
1.2 0.5 ; 1.4 0.65 ; 1.6 0.67 ; 1.8 0.55 ; 2.0 0.43
```

```
LOAD 2 SPECTRUM IN X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
SPECTRUM SRSS X 1.0 ACC SCALE 32.2
0.20 0.2 ; 0.40 0.25 ; 0.60 0.35 ; 0.80 0.43 ; 1.0 0.47
1.2 0.5 ; 1.4 0.65 ; 1.6 0.67 ; 1.8 0.55 ; 2.0 0.43
```

Multiple Response Spectra

If there is more than one response spectrum defined in the input file, the load data (representing the dynamic weight) should accompany the first set of spectrum data only. In the subsequent load cases, only the spectra should be defined. See example below.

```
LOAD 1 SPECTRUM IN X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
```

```

SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
SPECTRUM SRSS X 1.0 ACC SCALE 32.2 IMR 2 STARTCASE 11
0.20 0.2 ; 0.40 0.25 ; 0.60 0.35 ; 0.80 0.43 ; 1.0 0.47
1.2 0.5 ; 1.4 0.65 ; 1.6 0.67 ; 1.8 0.55 ; 2.0 0.43
PERFORM ANALYSIS
CHANGE
*
LOAD 2 SPECTRUM IN Y-DIRECTION
SPECTRUM SRSS Y 1.0 ACC SCALE 32.2
0.20 0.1 ; 0.40 0.15 ; 0.60 0.33 ; 0.80 0.45 ; 1.00 0.48
1.20 0.51 ; 1.4 0.63 ; 1.6 0.67 ; 1.8 0.54 ; 2.0 0.42

```

File Format for Spectra Data

The format of the **FILE** spectra data allows spectra as a function of damping as well as period. The format is:

Dataset 1	MDAMPCV NPOINTCV	(no of values = 2)
Dataset 2	Damping Values in ascending order	(no of values = Mdampcv)
Dataset 3a	Periods	(no of values = Npointcv)
3b	Spectra	(no of values = Npointcv)

For **ASCE**, the **MIS** option is assumed to be on. If any of **f6**, **f7**, **f8**, **f9** are not entered the defaults will be used.

Repeat Data set 3 Mdampcv times (3a,3b , 3a,3b , 3a,3b , etc.) (i.e., for each damping value).

Data sets 2, 3a and 3b must have exactly Npointcv values each. Blanks or commas separate the values. The data may extend to several lines. Do not end lines with a hyphen (-). No comment lines (*) or semi-colons. Multiple values may be entered per line.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

- MDAMPCV= Number of damping values for which there will be separate Spectra vs. Period curves.
- NPOINTCV= Number of points in each Spectra vs. Period curve. If NPOINTCV is negative, then the period-spectra values are entered as pairs.

Individual Modal Response Case Generation

Individual modal response (**IMR**) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with Repeat Load combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

Note: When the IMR option is entered for a Spectrum case, then a Perform Analysis & Change must be entered after each such Spectrum case.

Input for using individual modal response cases is as follows:

```
GENERATE INDIVIDUAL MODAL RESPONSE CASES  
MODE LIST mode_list  
CASE LIST case_list  
{ REPEAT | ( COMBINATION ) }  
END
```

Where:

MODE LIST =list of mode numbers that will have a corresponding IMR load case generated. Enter mode numbers separated by spaces and/or use the “ii to jj” form or **ALL**. Default is mode 1. This data can be continued on multiple lines by entering a hyphen at the end of each line but the last. If the **MODE LIST** is entered more than once the data will be concatenated.

CASE LIST = list of load case numbers that matches the mode list. These case numbers must not have been used for any other case. If more cases than modes, the unused numbers will be ignored and remain available. If there are more modes than cases, then unused case numbers will be selected. Default will use unused case numbers for all of the modes selected. This data can be continued on multiple lines by entering a hyphen at the end of each line but the last. If the **CASE LIST** is entered more than once the data will be concatenated..

REPEAT or **COMBINATION** = If **COMBINATION** is chosen then these cases must not be entered in a **REPEAT LOAD** command. If **REPEAT** is chosen then the equivalent applied loads will be saved in a file. In addition, a static analysis of this case will be solved; and the case can then be used in a **REPEAT LOAD** command. The default is **COMBINATION**.

END = Used to end this data.

Example

```
LOAD 1 SPECTRUM IN X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
SPECTRUM CQC X 1.0 ACC SCALE 32.2
0.20 0.2 ; 0.40 0.25 ; 0.60 0.35 ; 0.80 0.43 ; 1.0 0.47
1.2 0.5 ; 1.4 0.65 ; 1.6 0.67 ; 1.8 0.55 ; 2.0 0.43
```

Section 5 Commands and Input Instructions

5.32 Loading Specifications

```
GENERATE IND MOD RES
MODE LIST 1 2 TO 5
CASE LIST 11 12 TO 15
REPEAT
END
LOAD 2 SPECTRUM IN Y-DIRECTION
SPECTRUM SRSS Y 1.0 ACC SCALE 32.2
0.20 0.1 ; 0.40 0.15 ; 0.60 0.33 ; 0.80 0.45 ; 1.00 0.48
1.20 0.51 ; 1.4 0.63 ; 1.6 0.67 ; 1.8 0.54 ; 2.0 0.42
```

5.32.10.1.2 Response Spectrum Specification in Conjunction with the Indian IS: 1893 (Part 1)-2002

This command may be used to specify and apply the **RESPONSE SPECTRUM** loading as per IS: 1893 (Part 1)-2002 for dynamic analysis.

The seismic load generator can be used to generate lateral loads in the X and Z directions only. Y is the direction of gravity loads.

Note: This facility has *not* been developed for cases where the Z axis is set to be the vertical direction using the **SET Z UP** command.

Methodology

The design lateral shear force at each floor in each mode is computed by STAAD in accordance with the Indian IS: 1893 (Part 1)-2002 equations 7.8.4.5c and 7.8.4.5d.

$$Q_{ik} = A_k \cdot \phi_{ik} \cdot P_k \cdot W_k$$

and

$$V_{ik} = \sum_{i=i+1}^n Q_{ik}$$

Note: All symbols and notations in the above equation are as per IS: 1893 (Part 1)-2002.

STAAD utilizes the following procedure to generate the lateral seismic loads.

1. You provide the value for $Z/2 \cdot I/R$ as factors for input spectrum
2. The program calculates time periods for first six modes or as specified.
3. The program calculates S_a/g for each mode utilizing time period and damping for each mode.
4. The program calculates design horizontal acceleration spectrum A_k for different modes.
5. The program then calculates mode participation factor for different modes.
6. The peak lateral seismic force at each floor in each mode is calculated.
7. All response quantities for each mode are calculated.
8. The peak response quantities are then combined as per the specified method (**SRSS**, **CQC**, **ABS**, **CSM** or **TEN**) to get the final results.

General Format

The data in the following format can be contained all on a single line or broken into two or three lines, so long as the second and third lines start with the **ACC** and **DOMINANT** or **SIGN** commands.

```
SPECTRUM { SRSS | ABS | CQC | CSM | TEN } 1893 (TORSION)
(COUPLED) (OPPOSITE) (ECCENTRIC f12) *{ X f1 | Y f2 | Z f3
} ACC (SCALE f4) {DAMP f5 | CDAMP | MDAMP } (MIS f6) (ZPA
f7) (IGN f14) ({ DOMINANT f8 | SIGN }) (IMR f9) (STARTCASE
f10)
```

The following command (**SOIL TYPE** parameter) must be in a separate line.

```
{ SOIL TYPE f11 | *{ P1,V1; P2,V2; P3,V3;...PN,VN } }
```

The following command, if present, must be on a separate line. This performs the optional soft story check.

```
( CHECK SOFT STORY )
```

The following command, if present, must be on a separate line. This performs the story drift check.

```
( CHECK STORY DRIFT ) (RF f13)
```

Where:

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Table 5-27: Parameters used for IS: 1893 (Part 1) 2002 response spectrum

Parameter	Default Value	Description
f ₁ , f ₂ , f ₃	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. These must be entered as the product of $Z/2 \cdot I/R$. Any one or all directions can be input. Directions not provided will default to zero.
f ₄	1.0	Linear scale factor by which design horizontal acceleration spectrum will be multiplied. This factor signifies that the structures and foundations, at which level base shear will be calculated, are placed below the ground level. Note: If site specific spectra curve is used then f ₄ value is to be multiplied by the scale factor by which spectra data will be multiplied. Usually to factor g's to length/sec ² units.
f ₅	0.05	Damping ratio for all modes when DAMP is used. Default value is 0.05 (5% damping if 0 or blank entered).

Parameter	Default Value	Description
f6		<p>Use Missing Mass method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the f6 value entered in length/sec² (this value is not multiplied by SCALE).</p> <p>If f6 is zero, then the spectral acceleration at the ZPA f7 frequency is used. If f7 is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results.</p> <p>Note: If the MIS parameter is entered on any spectrum case it will be used for all spectrum cases.</p>
f7	33 [Hz]	<p>For use with MIS option only. Defaults to 33 Hz if not entered. Value is printed but not used if MIS f6 is entered.</p>

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Parameter	Default Value	Description
f8	1 (1st Mode)	<p>Dominant mode method. All results will have the same sign as mode number f8 alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero).</p> <p>Note: Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode.</p>
f9	1	<p>The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through f9 will be used. Missing Mass modes are not output.</p>

Parameter	Default Value	Description
fio	Highest Load Case No. + 1	The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If fio is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case no. plus one.
fi1		The soil type present. Depending upon time period, types of soil and damping, average response acceleration coefficient, S_a/g is calculated. 1 = for rocky or hard soil 2 = medium soil 3 = soft soil sites
P1,V1; P2,V2; P3,V3; ... Pn, Vn		Data is part of input immediately following spectrum command. Period - Value pairs (pairs separated by semicolons) are entered to describe the spectrum curve. Period is in seconds and the corresponding Value is acceleration (current length unit/sec ²). If data is in g acceleration units then the factor by which spectra data will be multiplied is g to the current length unit (9.81, 386.4, etc). Do not enter if a Soil Type value is specified.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Parameter	Default Value	Description
f12	0.05	It is a factor which indicates the extent of accidental eccentricity. For all buildings this factor is to be provided as 0.05. However, for highly irregular buildings this factor may be increased to 0.10. This factor is to be externally provided to calculate design eccentricity.
f13		<p>The response reduction factor. If not specified, the program will look for the factor defined under DEFINE 1893 LOAD (See "Indian IS:1893 (Part 1) 2002 Code - Lateral Seismic Load" on page 449). If none is provided there either, a factor of 1.0 is assumed.</p> <p>The response reduction factor represents ratio of maximum seismic force on a structure during specified ground motion if it were to remain elastic to the design seismic force. Actual seismic force is reduced by a factor RF to obtain design force.)</p>

Parameter	Default Value	Description
f14	0.009	<p>It indicates the mass participation (in percent) of those modes to be excluded while considering torsion provision of IS-1893. Depending upon the model it may be found that there are many local modes and torsional modes whose mass participation is practically negligible. These modes can be excluded without much change in the final analysis result. If not provided all modes will be considered. If none provided the default value of 0.009% will be considered. If IGN is entered on any one spectrum case it will be used for all spectrum cases. (Optional input)</p> <p>Note: If the value of f14 is considerable it may lead to considerable variation of analysis result from the actual one. Hence caution must be taken while using IGN command.</p>

SAVE: This option results in the creation of an Acceleration data file (.ACC file extension) containing the joint accelerations in g's and radians/sec². These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

SRSS , **CQC** , **ABS** , **CSM** and **TEN** are methods of combining the responses from each mode into a total response.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

- SRSS stands for square root of summation of squares,
- CQC for complete quadratic combination
- ABS for Absolute method.
- CSM is a closely-spaced modes grouping method where the peak response quantities for closely spaced modes (considered to be within 10 percent of each other) are combined by Absolute method. This peak response quantity for closely spaced modes is then combined with those of widely spaced modes by SRSS method.
- TEN percent method combines modal responses as published in US Nuclear Regulatory Guide 1.92 Revision 1 of February 1976.

Note: CQC, SRSS and CSM Grouping methods are recommended by IS:1893 (Part 1) -2002.

1893 indicates the analysis as per IS:1893(Part 1)-2002 procedures.

TORSION indicates that the torsional moment (in the horizontal plane) arising due to eccentricity between the center of mass and center of rigidity needs to be considered. See "Torsion" on page 610 for additional information.

Note: If **TOR** is entered on any one spectrum case it will be used for all spectrum cases.

Lateral shears at story levels are calculated in global X and Z directions. For global Y direction the effect of torsion will not be considered.

COUPLED indicates that the dynamic analysis had been performed for torsionally coupled system. Only application of torsion from accidental eccentricity is sufficient (i.e., the program will exclude torsion from dynamic eccentricity). (Optional input)

OPPOSITE indicates to consider horizontal torsional moment arising due to accidental eccentricity in opposite direction. Since accidental eccentricity can be on either side one must consider lateral force acting at a floor level to be accompanied by a clockwise or an anticlockwise accidental torsion moment.

Note: To provide accidental torsion in both clockwise and anticlockwise direction, you must create two separate response spectrum load cases: one with only **TORSION** and the other with **TORSION OPPOSITE**.

ACC indicates Acceleration spectra will be entered.

DAMP, **MDAMP**, and **CDAMP** Select source of damping input:

- **DAMP** indicates to use the f_2 value for all modes.
- **MDAMP** indicates to use the damping entered or computed with the **DEFINE DAMP** command if entered, otherwise default value of 0.05 will be used.
- **CDAMP** indicates to use the composite damping of the structure calculated for each mode. One must specify damping for different materials under the **CONSTANT** specification.

SIGN = This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This option is ignored for **ABS** option for combining responses.

Warning: Do not enter **DOMINANT** parameter with the **SIGN** option.

If the **MODE SELECT** command is provided along with the **IGN** command, the number of modes excluded from the analysis will be those deselected by the **MODE SELECT** command and *also* those deselected by the **IGN** command.

CHECK SOFT STORY indicates that soft story checking will be performed. If omitted from input, there will be no soft story checking. For additional details, See "Indian IS:1893 (Part 1) 2002 Code - Lateral Seismic Load" on page 449 in Section 5.31.2.5.

Individual Modal Response Case Generation

Individual modal response (**IMR**) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the **IMR** parameter is entered, then **STAAD** will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an **IMR** case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with Repeat Load combinations of these scaled modes and the static live and dead loads to form results that are all with

Section 5 Commands and Input Instructions

5.32 Loading Specifications

internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

Note: When the IMR option is entered for a Spectrum case, then a Perform Analysis & Change must be entered after each such Spectrum case.

See "Response Spectrum Specification - Generic Method" on page 587 for additional details on IMR load case generation.

Notes

- a. The design base shear V_B , calculated from the Response Spectrum method, is compared with the base shear V_b , calculated by empirical formula for the fundamental time period. If V_B is less than V_b , all of the response quantities are multiplied by V_b/V_B as per Clause 7.8.2.

For this, the following input is necessary before defining any primary load case.

```
DEFINE 1893 LOAD
ZONEf1 1893-spec
SELFWEIGHT
JOINT WEIGHT
joint-list WEIGHT w
MEMBER WEIGHT

UNI v1 v2 v3
mem-list
CONv4 v5

CHECK SOFT STORY
```

1893-Spec= {RF f2, I f3, SS f4, (ST f5), DM f6, (PX f7), (PZ f8), (DT f9)}

Refer to Section 5.31.2.5 for full details on this command structure.

Note: STAAD does not calculate the fundamental frequency of the structure needed for the empirical base shear V_b calculation; so you must enter either the ST parameter or the PX and PZ parameters in the **DEFINE 1893 LOAD** data.

- b. The following interpolation formula is adopted for interpolation between damping values as given in Table 3.

Interpolation and/or extrapolation of ground response acceleration for a particular mode has been made for determining the spectrum ordinates corresponding to the modal damping value for use in Response Spectrum analysis. The relationship that shall be used for this purpose is defined by:

$$S_a = A e^{-\xi} + B/\xi$$

Where:

S_a = Spectrum ordinate

ξ = damping ratio

Constants A and B are determined using two known spectrum ordinates S_{a_1} and S_{a_2} corresponding to damping ratios ξ_1 and ξ_2 , respectively, for a particular time period and are as follows:

$$A = \frac{S_{a_1}\xi_1 - S_{a_2}\xi_2}{\xi_1 e^{-\xi_1} - \xi_2 e^{-\xi_2}}$$

$$B = \frac{\xi_1 \xi_2 (S_{a_2} e^{-\xi_1} - S_{a_1} e^{-\xi_2})}{\xi_1 e^{-\xi_1} - \xi_2 e^{-\xi_2}}$$

Where:

$$\xi_1 < \xi < \xi_2$$

- c. The story drift in any story shall not exceed 0.004 times the story height as per Clause 7.11.1. To check this, the following command should be given after the analysis command.

PRINT STOREY DRIFT

A warning message will be printed if story drift exceeds this limitation.

- d. If any soft story (as per definition in Table 5 of IS:1893-2002) is detected, a warning message will be printed in the output.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Torsion

The torsion arising due to dynamic eccentricity (i.e., static eccentricity multiplied by dynamic amplification factor) between center of mass and center of rigidity is applied along with accidental torsion, as per the recommendations of Cl. 7.9.2 of the IS 1893 code. The dynamic eccentricity is automatically calculated by the program (in both cases of **TOR** and **TOR OPP** options), while the amount of accidental eccentricity can be specified through the **ECC** option (if not specified, default of 5% of lateral dimension of the floor in the direction of the earthquake will be considered).

Non-symmetric or torsionally unbalanced buildings are prone to earthquake damage due to coupled lateral and torsional movements (i.e., the translational vibration of the building couples with its torsional vibrations within elastic range). The level of coupling between lateral and torsional vibrations of the building can be larger, thus leading to significantly higher lateral-torsional coupling than that predicted by elastic analysis.

- Cl. 7.8.4.5 of IS 1893 (Part 1) : 2002 is valid for buildings with *regular* or *nominally irregular* plan configurations. For buildings which are *irregular* in plan, it is better to consider torsion from dynamic eccentricity into analysis; even if torsionally coupled vibration is considered during response spectrum analysis.
- Cl. 7.9.2 Note 2 of Amendment No. 1 January 2005 states that, in the case that a 3D dynamic analysis is carried out, the dynamic amplification factor 1.5—as given by Cl. 7.9.2—can be replaced by 1.0. This implies that the code also recommends to use Cl. 7.9.2 for all types of buildings by including torsion from both dynamic and accidental eccentricity in the response spectrum analysis.

Methodology

As per IS1893-2002 code, provision shall be made in all buildings for increase in shear forces on the lateral force resisting elements resulting from the horizontal torsional moment arising due to eccentricity between the center of mass and the center of rigidity.

In response spectrum analysis all the response quantities (i.e. joint displacements, member forces, support reactions, plate stresses, etc) are calculated for each mode of vibration considered in the analysis. These response quantities from each mode are combined using a modal combination method (either by CQC,

SRSS, ABS, TEN PERCENT, etc) to produce a single positive result for the given direction of acceleration. This computed result represents a maximum magnitude of the response quantity that is likely to occur during seismic loading. The actual response is expected to vary from a range of negative to positive value of this maximum computed quantity.

No information is available from response spectrum analysis as to when this maximum value occurs during the seismic loading and what will be the value of other response quantities at that time. As for example, consider two joints J_2 and J_3 whose maximum joint displacement in global X direction come out to be X_1 and X_2 respectively. This implies that during seismic loading joint J_1 will have X direction displacement that is expected to vary from $-X_1$ to $+X_1$ and that for joint J_2 from $-X_2$ to $+X_2$. However, this does not necessarily mean that the point of time at which the X displacement of joint J_1 is X_1 , the X displacement of joint J_2 will also be X_2 .

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

Steps

For each mode following steps are performed to include Torsion provision.

1. Lateral storey force at each floor is calculated. Refer Cl. 7.8.4.5c of IS 1893 (Part 1) : 2002. (Q_{ik} at floor i for mode k)
2. At each floor design eccentricity is calculated. Refer Cl. 7.9.2 and Cl. 7.9.2 Note 2 of Amendment No. 1 January 2005 of IS 1893 (Part 1) : 2002.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Com- mand Option	Design Eccen- tricity, e_{di}
TOR	$e_{di} = e_{si} + e_{ac} \cdot b_i$
TOR OPP	$e_{di} = e_{si} - e_{ac} \cdot b_i$
TOR COU	$e_{di} = e_{ac} \cdot b_i$
TOR COU OPP	$e_{di} = -e_{ac} \cdot b_i$

Where:

e_{si} = dynamic eccentricity arising due to center of mass and center of rigidity at floor i (static eccentricity multiplied by dynamic amplification factor 1.0 for response spectrum analysis),

e_{ac} = extent of accidental eccentricity (0.05 by default, unless not specified)

b_i = floor plan dimension in the direction of earthquake loading.

3. Torsional moment is calculated at each floor. ($M_{ik} = Q_{ik} \cdot e_{di}$ at floor i for mode k)
4. The lateral nodal forces corresponding to torsional moment are calculated at each floor. These forces represent the additional story shear due to torsion.
5. Static analysis of the structure is performed with these nodal forces.
6. The analysis results (i.e., member force, joint displacement, support reaction, etc) from torsion are algebraically added to the corresponding modal response quantities from response spectrum analysis.

Modal Combination

Steps 1 to 6 are performed for all modes considered and missing mass correction (if any). Finally the peak response quantities from different modal response are combined as per **CQC** or **SRSS** or **TEN PERCENT** or **CSM** method.

Notes

After the analysis is complete following files are generated.

- a. Story shear for each mode for each load case is given in the file <FILENAME>_RESP1893.TXT.
- b. Rotational stiffness of each floor is given in the file <FILENAME>_ROT1893.TXT.
- c. Center of mass, center of rigidity, design eccentricity at each floor level and additional shear due to torsion at each floor level for each mode for each load case is given in the file <FILENAME>_TOR1893.TXT.

5.32.10.1.3 Response Spectrum Specification per Eurocode 8 1996

This command may be used to specify and apply the **RESPONSE SPECTRUM** loading as per the 1996 edition of Eurocode 8 (EC8) for dynamic analysis.

General Format

```
SPECTRUM { SRSS | ABS | CQC | ASCE | TEN } EURO {ELASTIC
| DESIGN} *{ X f1 | Y f2 | Z f3 } ACC
{DAMP f4 | CDAMP | MDAMP } ( {LIN | LOG} ) (MIS f5) (ZPA f6)
({ DOMINANT f7 | SIGN }) (SAVE) (IMR f8) (STARTCASE f9)
```

Note: The data from **SPECTRUM** through **ACC** must be on the first line of the command, the remaining data can be on the first or subsequent lines with all but last ending with a hyphen (limit of four lines per spectrum).

Starting on the next line, the response spectra is input using following standard input parameters:

```
SOIL TYPE { A | B | C} ALPHA f10 Q f11
```

Where:

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Table 5-28: Parameters used for Eurocode 8 1996 response spectrum

Parameter	Default Value	Description
f ₁ , f ₂ , f ₃	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.
f ₄	0.05	Damping ratio for all modes when DAMP is used. Default value is 0.05 (5% damping if 0 or blank entered).
f ₅		<p>Use Missing Mass method to include the static effect of the masses not represented in the modes. The spectral acceleration for this missing mass mode is the f₅ value entered in length per second squared units (this value is not multiplied by SCALE).</p> <p>If f₅ is zero, then the spectral acceleration at the ZPA f₆ frequency is used. If f₆ is zero or not entered, then the spectral acceleration at 33Hz is used. The results of this calculation are SRSSed with the modal combination results.</p> <p>Note: If the MIS parameter is entered on any spectrum case it will be used for all spectrum cases.</p>

Parameter	Default Value	Description
f6	33 [Hz]	For use with MIS option only. Defaults to 33 Hz if not entered. Value is printed but not used if MIS f5 is entered.
f7	1 (1st Mode)	<p>Dominant mode method. All results will have the same sign as mode number f7 alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero).</p> <p>Note: Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode.</p>
f8	1	The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through f8 will be used. Missing Mass modes are not output.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Parameter	Default Value	Description
f9	Highest Load Case No. + 1	The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If f12 is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case no. plus one.
f10		Alpha is defined to be design ground acceleration expressed in terms of acceleration due to gravity(g). For most of the application of Eurocode 8, the hazard is described in terms of a single parameter (i.e., the value of effective peak ground acceleration in rock or firm soil). This acceleration is termed as the design ground acceleration.
f11		Q is the behavior factor used to reduce the elastic response spectra to the design response spectra. The behavior factor is an approximation of the ratio of the seismic forces, that the structure would experience, if its response was completely elastic with 5% viscous damping, to the minimum seismic forces that may be used in design- with a conventional linear model still ensuring a satisfactory response of the structure.

Unlike the custom defined response spectra, the EC 8 response spectra is not input using frequency-acceleration pairs. Based on the type of Response Spectra (Elastic/Design), Soil Type, Alpha and Q the software generates the applicable response spectra curve using the guidelines of section 4.2.2 or 4.2.4 of Eurocode 8 as applicable.

SRSS, ABS, CQC, CSM, & TEN Percent are methods of combining the responses from each mode into a total response. CQC and TEN include the effect of response magnification due to closely spaced modal frequencies. CQC is a more sophisticated and realistic method and is recommended.

- **SRSS**- Square Root of Summation of Squares method.
- **CQC** - Complete Quadratic Combination method. (Default).
- **ABS** - Absolute sum. (Very conservative worst case)
- **TEN** - Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (1976).
- **CSM** - Closely-Spaced Modes grouping method where the peak response quantities for closely spaced modes (considered to be within 10 percent of each other) are combined using the Absolute method. This peak response quantity for closely spaced modes is then combined with those of widely spaced modes by SRSS method.

The specifier **EURO** is mandatory to denote that the applied loading is as per the guidelines of Eurocode 8.

The response spectrum loading can be based on either **ELASTIC** or **DESIGN** response spectra.

Within the scope of Eurocode 8 the earthquake motion at a given point of the surface is generally represented by an elastic ground acceleration response spectrum termed as “Elastic Response Spectrum”. STAAD generates the Elastic Response Spectra using the guidelines of section 4.2.2 and Table 4.1 of Eurocode 8.

The capacity of structural systems to resist seismic actions in the nonlinear range generally permits their design for forces smaller than those corresponding to a linear elastic response. To avoid explicit nonlinear structural analysis in design, the energy dissipation capacity of the structure through mainly ductile behavior of its elements and/or other mechanisms, is taken into account by performing a linear analysis based on a response spectrum which is a reduced form of the corresponding elastic response spectrum. This reduction is accomplished by introducing the behavior factor Q and the reduced response spectrum is termed

Section 5 Commands and Input Instructions

5.32 Loading Specifications

as “Design Response Spectrum”. STAAD generates the Elastic Response Spectra using the guidelines of section 4.2.4 and Table 4.2 of Eurocode 8.

So, if the structure is supposed to resist seismic actions in the nonlinear range the Design Response Spectra is to be used.

ACC indicates that the Acceleration spectra will be entered. Eurocode 8 does not provide displacement response spectra.

DAMP, CDAMP, MDAMP: Select source of damping input.

- **DAMP** indicates to use the f_4 value for all modes.
- **CDAMP** indicates to use Composite modal damping if entered, otherwise same as MDAMP.
- **MDAMP** indicates to use the damping entered or computed with the **DEFINE DAMP** command if entered, otherwise default value of 0.05 will be used.

LIN or **LOG**. Select linear or logarithmic interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. **LIN** is default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve. For **LIN** or **LOG**, when **FILE fn** is entered, the interpolation along the damping axis will be linear.

Note: The last LIN-LOG parameter entered on all of the spectrum cases will be used for all spectrum cases.

SAVE = The save option results in the creation of a file (file name with "Acc" extension) containing the joint accelerations in g's and radians/sec²

SIGN = This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. Do not enter **DOMINANT** parameter with the **SIGN** option. **SIGN** option is ignored for **ABS** option.

SOIL TYPE parameter is used to define the subsoil conditions based on which the response spectra will be generated. Based on the subsoil conditions the soil types may be of three kinds

- Type A :for Rock or stiff deposits of sand
- Type B :for deep deposits of medium dense sand,gravel or medium stiff clays.
- Type C:Loose cohesionless soil deposits or deposits with soft to medium stiff cohesive soil.

Please refer section 3.2 of Eurocode 8 for detailed guidelines regarding the choice of soil type.

Individual Modal Response Case Generation

Individual modal response (**IMR**) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with Repeat Load combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be ω^2 times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

Note: When the IMR option is entered for a Spectrum case, then a Perform Analysis & Change must be entered after each such Spectrum case.

See "Response Spectrum Specification - Generic Method" on page 587 for additional details on IMR load case generation.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Description

This command should appear as part of a loading specification. If it is the first occurrence, it should be accompanied by the load data to be used for frequency and mode shape calculations. Additional occurrences need no additional information. Maximum response spectrum load cases allowed in one run is 4.

Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement. For dynamic mass modeling, see sections 5.32 and 1.18.3 of STAAD Technical Reference Manual. An illustration of mass modeling is available, with explanatory comments, in Example Problem No.11_EC8.

Example

```
LOAD 2 SPECTRUM X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
MEMBER LOADS
5 CON GX 5.0 6.0
5 CON GY 5.0 6.0
5 CON GX 7.5 10.0
5 CON GY 7.5 10.0
5 CON GX 5.0 14.0
5 CON GY 5.0 14.0
SPECTRUM SRSS EURO ELASTIC X 1 ACC DAMP 0.05 -
LIN MIS 0 ZPA 40
SOIL TYPE A ALPHA 2 Q 1.5
```

Multiple Response Spectra

For special conditions more than one spectrum may be needed to adequately represent the seismic hazard over an area. This happens when the earthquake affecting the area are generated by sources varying widely in location and other parameters. In those cases different values of **ALPHA** as well as **Q** may be required to indicate the different shapes of response spectrum for each type of earthquake.

5.32.10.1.4 Response Spectrum Specification per Eurocode 8 2004

This command may be used to specify and apply the **RESPONSE SPECTRUM** loading as per the 2004 edition of Eurocode 8, 'General Rules, seismic actions and rules for buildings', BS EN 1998-1:2004. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

General Format

```
SPECTRUM { SRSS | ABS | CQC | ASCE | TEN } EURO 2004
{ELASTIC | DESIGN} {RS1 | RS2} *{ X f1 | Y f2 | Z f3 } ACC
{DAMP f4 | CDAMP | MDAMP } ( {LIN | LOG} ) (MIS f5) (ZPA f6)
({ DOMINANT f7 | SIGN }) (SAVE) (IMR f8) (STARTCASE f9)
```

Note: The data **SPECTRUM** through **ACC** must be on the first line of the command, the remaining data can be on the first or subsequent lines with all but last ending with a hyphen (limit of four lines per spectrum).

Starting on the next line, the response spectra is input using following standard input parameters:

```
SOIL TYPE { A | B | C | D | E } ALPHA f10 Q f11
```

Where:

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Table 5-29: Parameters used for Eurocode 8 2004 response spectrum

Parameter	Default Value	Description
f ₁ , f ₂ , f ₃	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.
f ₄	0.05	Damping ratio for all modes when DAMP is used. Default value is 0.05 (5% damping if 0 or blank entered).
f ₅		<p>Use Missing Mass method to include the static effect of the masses not represented in the modes. The spectral acceleration for this missing mass mode is the f₅ value entered in length per second squared units (this value is not multiplied by SCALE).</p> <p>If f₅ is zero, then the spectral acceleration at the ZPA f₆ frequency is used. If f₆ is zero or not entered, then the spectral acceleration at 33Hz is used. The results of this calculation are SRSSed with the modal combination results.</p> <p>Note: If the MIS parameter is entered on any spectrum case it will be used for all spectrum cases.</p>

Parameter	Default Value	Description
f6	33 [Hz]	For use with MIS option only. Defaults to 33 Hz if not entered. Value is printed but not used if MIS f5 is entered.
f7	1 (1st Mode)	<p>Dominant mode method. All results will have the same sign as mode number f7 alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero).</p> <p>Note: Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode.</p>
f8	1	The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through f8 will be used. Missing Mass modes are not output.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Parameter	Default Value	Description
f9	Highest Load Case No. + 1	The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If f9 is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case no. plus one.
f10		Alpha is the 'Ground Acceleration On Type A Ground' and defined in Eurocode 8 as a_g , as used in equations 3.2 – 3.5. Refer to the Eurocode for further information.
f11		Q is the 'Behaviour Factor' and is an approximation of the ratio of the seismic forces that the structure would experience if its response was completely elastic with 5% viscous damping, to the seismic forces that may be used in design, with a conventional elastic analysis model, still ensuring a satisfactory response of the structure.

Unlike the custom defined response spectra the EC 8 response spectra is not input using frequency-acceleration pairs. Based on the type of Response Spectra (Elastic/Design), Soil Type, Alpha and Q the software generates the applicable response spectra curve using the guidelines of section 3.2.2.2 or 3.2.2.3 or 3.2.2.5 of Eurocode 8: 2004 as applicable.

SRSS, ABS, CQC, CSM, & TEN Percent are methods of combining the responses from each mode into a total response. CQC and TEN include the effect of response magnification due to closely spaced modal frequencies. CQC is a more sophisticated and realistic method and is recommended.

- **SRSS** - Square Root of Summation of Squares method.
- **CQC** - Complete Quadratic Combination method. (Default).
- **ABS** - Absolute sum. (Very conservative worst case)
- **TEN** - Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (1976).
- **CSM** - Closely-Spaced Modes grouping method where the peak response quantities for closely spaced modes (considered to be within 10 percent of each other) are combined using the Absolute method. This peak response quantity for closely spaced modes is then combined with those of widely spaced modes by SRSS method.

See Sections 1.18.3, 5.30, and 5.34 of STAAD Technical Reference Manual

The keywords **EURO 2004** is mandatory to denote that the applied loading is as per the guidelines of Eurocode 8.

The response spectrum loading can be based on either **ELASTIC** or **DESIGN** response spectra.

Two types of response spectra curve can be generated based on either **RS₁** (for response spectra type 1 curve) or **RS₂** (for response spectra type 2 curve) for each **ELASTIC** and **DESIGN** option.

Within the scope of Eurocode 8 the earthquake motion at a given point of the surface is generally represented by an elastic ground acceleration response spectrum termed as “Elastic Response Spectrum”. STAAD generates the Elastic Response Spectra using the guidelines of section 4.2.2 and Table 4.1 of Eurocode 8.

The capacity of structural systems to resist seismic actions in the nonlinear range generally permits their design for forces smaller than those corresponding to a linear elastic response. To avoid explicit nonlinear structural analysis in design, the energy dissipation capacity of the structure through mainly ductile behavior of its elements and/or other mechanisms, is taken into account by performing a linear analysis based on a response spectrum which is a reduced form of the corresponding elastic response spectrum. This reduction is accomplished by introducing the behavior factor Q and the reduced response spectrum is termed as “Design Response Spectrum”. STAAD generates the Elastic Response Spectra using the guidelines of section 4.2.4 and Table 4.2 of Eurocode 8. Therefore, if the structure is supposed to resist seismic actions in the nonlinear range the Design Response Spectra should be specified.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

ACC indicates that the Acceleration spectra will be entered. Eurocode 8 does not provide displacement response spectra.

DAMP , **CDAMP** , **MDAMP**. Defines the method of damping to be employed.

- **DAMP** signifies use of single damping factor f_4 for all modes.
- **CDAMP** signifies use of Composite Modal Damping. This evaluates the damping from that defined in the material or constant definitions. If there is no damping information entered in the material or constant definitions, the behavior is the same as MDAMP.
- **MDAMP** signifies the use of Modal Damping. If this option is selected, then a **DEFINE DAMP** command is required. If this has not been defined, then a default value of 0.05 will be used. See Technical Reference Manual section 5.26.4 Modal Damping Information for more information on this command.

LIN or **LOG**. Select linear or logarithmic interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. **LIN** is default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

SIGN = This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign.

Warning: Do not enter **DOMINANT** parameter with the **SIGN** option. **SIGN** option is ignored for **ABS** option.

SAVE = The save option results in the creation of a file (file name with an .ACC extension) containing the joint accelerations in g's and radians/sec²)

SOIL TYPE parameter is used to define the subsoil conditions based on which the response spectra will be generated as defined in Table 3.1 Ground Types.

Based on the subsoil conditions the soil types may be of five kinds:

- Type **A**: rock or other rock-like geographical formation.
- Type **B**: very dense sand, gravel or very stiff clay.
- Type **C**: Deep deposits of dense or medium dense sand, gravel or stiff clay.
- Type **D**: Deposits of loose-to-medium cohesionless soil or of predominantly soft to firm cohesive soil.
- Type **E**: Surface alluvium layer

Please refer section 3.2 of Eurocode8 for detailed guidelines regarding the choice of soil type.

Individual Modal Response Case Generation

Individual modal response (**IMR**) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with Repeat Load combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

Note: When the IMR option is entered for a Spectrum case, then a Perform Analysis & Change must be entered after each such Spectrum case.

See "Response Spectrum Specification - Generic Method" on page 587 for additional details on IMR load case generation.

Description

This command should appear as part of a loading specification. If it is the first occurrence, it should be accompanied by the load data to be used for frequency and mode shape calculations. Additional occurrences need no additional information.

Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement. For

Section 5 Commands and Input Instructions

5.32 Loading Specifications

dynamic mass modeling, see sections 5.32 and 1.18.3 of STAAD Technical Reference Manual.

Example

```
LOAD 2 SPECTRUMIN X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
MEMBER LOADS
5 CON GX 5.0 6.0
5 CON GY 5.0 6.0
5 CON GX 7.5 10.0
5 CON GY 7.5 10.0
5 CON GX 5.0 14.0
5 CON GY 5.0 14.0
SPECTRUM SRSS EURO ELASTIC X 1 ACC DAMP 0.05 -
LIN MIS 0 ZPA 40
SOIL TYPE A ALPHA 2 Q 1.5
```

Multiple Response Spectra

For special conditions more than one spectrum may be needed to adequately represent the seismic hazard over an area. This happens when the earthquake affecting the area is generated by sources varying widely in location and other parameters. In such cases different values of **ALPHA** as well as **Q** may be required to indicate the different shapes of response spectrum for each type of earthquake.

5.32.10.1.5 Response Spectrum Specification per IBC 2006

This command may be used to specify and apply the **RESPONSE SPECTRUM** loading as per the 2006 edition of the ICC specification *International Building Code* (IBC), for dynamic analysis. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

Methodology

The methodology for calculating the response spectra is defined in ASCE-7-2005, section 11.4. The following is a quick summary:

- a. Input S_s and S_1 (this could have been searched from database or entered explicitly)
- b. Calculate

$$S_{ms} = F_a \times S_s$$

and

$$S_{m1} = F_v \times S_1$$

Where:

F_a and F_v are determined from the specified site classes A – E and using tables 11.4-1 and 11.4-2. For site class F, the values must be supplied. These are required to be provided by the user. You may also specify values for F_a and F_v in lieu of table values.

- c. Calculate

$$S_{ds} = (2/3) S_{ms}$$

and

$$S_{d1} = (2/3) S_{m1}$$

The spectrum is generated as per section 11.4.5.

General Format

```
SPECTRUM { SRSS | ABS | CQC | ASCE | TEN } IBC 2006 *{ X f1
| Y f2 | Z f3 } ACC
{DAMP f5 | CDAMP | MDAMP } ( {LIN | LOG} ) (MIS f6) (ZPA f7)
({ DOMINANT f8 | SIGN }) (SAVE) (IMR f9) (STARTCASE f10)
```

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Note: The data from **SPECTRUM** through **ACC** must be on the first line of the command. The data shown on the second line above can be continued on the first line or one or more new lines with all but last ending with a hyphen (limit of four lines per spectrum).

The command is completed with the following data which must be started on a new line:

```
{ZIP f11 | LAT f12 LONG f13 | SS f14 S1 f15 } SITE CLASS  
(f16) (FA f17 FV f18) TL f19
```

Where:

Table 5-30: Parameters used for IBC 2006 response spectrum

Parameter	Default Value	Description
f1, f2, f3	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.
f5	0.05	Damping ratio for all modes when DAMP is used. Default value is 0.05 (5% damping if 0 or blank entered).

Parameter	Default Value	Description
f6		<p>Optional parameter to use 'Missing Mass' method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the f6 value entered in length/sec² (this value is not multiplied by SCALE).</p> <p>If f6 is zero, then the spectral acceleration at the ZPA f7 frequency is used. If f7 is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results.</p> <p>Note: If the MIS parameter is entered on any spectrum case it will be used for all spectrum cases.</p>
f7	33 [Hz]	<p>For use with MIS option only. Defaults to 33 Hz if not entered. Value is printed but not used if MIS f6 is entered.</p>

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Parameter	Default Value	Description
f8	1 (1st Mode)	<p>Dominant mode method. All results will have the same sign as mode number f8 alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value is entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only 1 direction factor may be nonzero).</p> <p>Note: Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode.</p>
f9	33 [Hz]	<p>The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through f9 will be used. Missing Mass modes are not output.</p>

Parameter	Default Value	Description
f10	Highest Load Case No. + 1	The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If f10 is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case no. plus one.
f11		The zip code of the site location to determine the latitude and longitude and consequently the S_s and S_1 factors. (IBC 2006, ASCE 7-02 Chapter 22)
f12		The latitude of the site used with the longitude to determine the S_s and S_1 factors. (IBC 2006, ASCE 7-02 Chapter 22)
f13		The longitude of the site used with the latitude to determine the S_s and S_1 factors. (IBC 2006, ASCE 7-02 Chapter 22)
f14		Mapped MCE for 0.2s spectral response acceleration. (IBC 2006, ASCE 7-02 Chapter 22)
f15		Mapped spectral acceleration for a 1-second period. (IBC 2000, equation 16-17. IBC 2003, ASCE 7-02 section 9.4.1.2.4-2. IBC 2006, ASCE 7-05 Section 11.4.1)

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Parameter	Default Value	Description
fi6		Enter A through F for the Site Class as defined in the IBC code. (IBC 2000, Section 1615.1.1 page 350. IBC 2003, Section 1615.1.1 page 322. IBC 2006 ASCE 7-05 Section 20.3)
fi7		Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006, ASCE 7-05 Section 11.4.3)
fi8		Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006, ASCE 7-05 Section 11.4.3)
fi9		Long-Period transition period in seconds. (IBC 2006, ASCE 7-02 Chapter 22)

SRSS, ABS, CQC, CSM, & TEN Percent are methods of combining the responses from each mode into a total response. CQC and TEN include the effect of response magnification due to closely spaced modal frequencies. CQC is a more sophisticated and realistic method and are recommended.

- **SRSS**- Square Root of Summation of Squares method.
- **CQC** - Complete Quadratic Combination method. (Default).
- **ABS** - Absolute sum. (Very conservative worst case)
- **TEN** - Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (1976).
- **CSM** - Closely-Spaced Modes grouping method where the peak response quantities for closely spaced modes (considered to be within 10 percent of each other) are combined using the Absolute method. This peak response

quantity for closely spaced modes is then combined with those of widely spaced modes by SRSS method.

Note: If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

IBC2006 indicates that the spectrum should be calculated as defined in the IBC 2006 specification.

ACC indicates that an acceleration spectrum is defined.

Source of damping input:

- **MDAMP** indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used.
- **CDAMP** is the composite damping of the structure calculated for each mode. One must specify damping for different materials under **CONSTANT** specification.

LIN, LOG The interpolation method of the spectrum curves for determining the spectrum value for modes intermediate points. LIN is default option if not specified.

SIGN = This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. Do not enter DOMINANT parameter with the SIGN option. SIGN option is ignored for ABS option.

SAVE = Optional parameter to save the results in a file (file name with .ACC extension) containing the joint accelerations in g's and radians/sec²)

Individual Modal Response Case Generation

Individual modal response (**IMR**) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with Repeat Load combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

Note: When the IMR option is entered for a Spectrum case, then a Perform Analysis & Change must be entered after each such Spectrum case.

See "Response Spectrum Specification - Generic Method" on page 587 for additional details on IMR load case generation.

5.32.10.1.6 Response Spectrum Specification per SNiP II-7-81

This command may be used to specify and apply the **RESPONSE SPECTRUM** loading as per SNiP II-7-81 for dynamic analysis.

Note: This feature requires STAAD.Pro 2007.03 or greater.

General Format

```
SPECTRUM { SRSS | CQC } SNIP A f1 *{ { X f2 | KWX f3 KX1 f4 } |  
{ Y f5 | KWY f6 KY1 f7 } | { Z f8 | KWZ f9 KZ1 f10 } } ACC  
(SCALE f11)  
{DAMP f12 | CDAMP | MDAMP } ( {LIN | LOG} ) (MIS f13) (ZPA  
f14) ( { DOMINANT f15 | SIGN } ) SOIL { 1 | 2 | 3 } (SAVE)
```

The data from **SPECTRUM** through **SCALE** above must be on the first line of the command, the remaining data can be on the first or subsequent lines with all but last ending with a hyphen (limit of four lines per spectrum).

Where:

Table 5-31: Parameters for SNiP II-7-81 response spectrum

Parameter	Default Value	Description
f_1		Zoning factor, A, which is based on maximum acceleration factor for the seismic zone. This factor must be modified for SOIL types other than 2. The exact zone factor value used for a specific location requires engineering judgement. The following table serves as a guide for accelerations and corresponding zone factors which would be used.
$f_2, f_5, \text{ and } f_8$	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero. Alternatively, you may input individual parameters such as KWX, KX1 the product of which would be used as the factor along that direction.
f_{11}		A linear scaling factor by which the spectrum data will be multiplied. This can be used to factor
f_{12}	0.05	Damping ratio for all modes when DAMP is used. Default value is 0.05 (5% damping if 0 or blank entered).

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Parameter	Default Value	Description
f ₁₃		<p>Use Missing Mass method to include the static effect of the masses not represented in the modes. The spectral acceleration length/sec² for this missing mass mode is the f₁₃ value entered in length per second squared units (this value is not multiplied by SCALE). If f₁₃ is zero, then the spectral acceleration at the ZPA f₁₄ frequency is used. If f₁₄ is zero or not entered, then the spectral acceleration at 33Hz is used. The results of this calculation are SRSSed with the modal combination results.</p> <p>For SRSS and CQC, the results of this calculation are SRSSed with the modal combination results. If either f₁₃ or f₁₄ are not entered, the defaults will be used. Missing mass does not include the effect of masses lumped at the supports unless the support is a stiff spring or an Enforced support.</p> <p>Note: If the MIS parameter is entered on any spectrum case it will be used for all spectrum cases.</p>

Parameter	Default Value	Description
f_{14}		For use with MIS option only. Defaults to 33 Hz if not entered. Value is printed but not used if MIS f_{13} is entered.
f_{15}		Dominant mode method. All results will have the same sign as mode number f_{15} alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value is entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only 1 direction factor may be nonzero). Do not enter SIGN parameter with this option. Ignored for ABS option.

SRSS & CQC are methods of combining the responses from each mode into a total response. The CQC method require damping, SRSS does not use damping unless Spectra-Period curves are made a function of damping (see File option below). CQC includes the effect of response magnification due to closely spaced modal frequencies. CQC is more sophisticated and realistic method and is recommended.

- **SRSS** - Square Root of Summation of Squares method as prescribed by the SNiP II-7-81 code.
- **CQC** - Complete Quadratic Combination method.

The specifier **SNIP** is mandatory to denote that the applied loading is as per the guidelines of SNiP II-7-81.

ACC indicates that the definition is that of an acceleration spectrum.

DAMP, **MDAMP**, and **CDAMP** Select source of damping input:

Section 5 Commands and Input Instructions

5.32 Loading Specifications

- **DAMP** indicates to use the f_2 value for all modes.
- **MDAMP** indicates to use the damping entered or computed with the **DEFINE DAMP** command if entered, otherwise default value of 0.05 will be used.
- **CDAMP** indicates to use the composite damping of the structure calculated for each mode. One must specify damping for different materials under the **CONSTANT** specification.

LIN or **LOG**: Select linear or logarithmic interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. **LIN** is default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve. The last **LIN-LOG** parameter entered on the spectrum cases will be used for all spectrum cases.

SIGN = This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. Do not enter **DOMINANT** parameter with the **SIGN** option.

SOIL = Defines the subsoil conditions on which the response spectrum will be generated.

Note: The Zoning Factor, $A(f_1)$, must be adjusted for soil types other than type 2.

1. Non-weathered rock and rocklike geological formation or permafrost subsoil.
2. Weathered rock or deep deposits of medium dense sand, gravel or medium stiff clays.
3. Loose cohesion less soil deposits or deposits with soft to medium stiff cohesive soil.

SAVE = This option results in the creation of a file (file name with .ACC extension) containing the joint accelerations in g's and radians/sec²

Description

Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement. For dynamic

mass modeling, see sections 5.32 and 1.18.3 of STAAD Technical Reference Manual. An illustration of mass modeling is available, with explanatory comments, in the sample file SEISMIC_RUSS.STD, which can be found in the STAAD.Pro installation folder under SPROV8I/STAAD/EXAMP/RUS/.

Example

The definition of a SNiP response spectrum in the X direction on a structure built on weather rock and where the Zoning Factor is 0.7071. As this is the first load case with a response spectrum, then the masses are modeled as loads.

```
LOAD 2 LOADTYPE SEISMIC TITLE SPECTRUM IN X-DIRECTION
*MASSES
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
MEMBER LOADS
5 CON GX 5.0 6.0
5 CON GY 5.0 6.0
5 CON GX 7.5 10.0
5 CON GY 7.5 10.0
5 CON GX 5.0 14.0
5 CON GY 5.0 14.0
*SNIP SPECTRUM DEFINITION
SPECTRUM SRSS SNIP A 0.7071 X 1.0 ACC DAMP 0.05 SCALE 1.0
LIN MIS 0 ZPA 40 SOIL 2
...
```

Note: The maximum response spectrum load cases allowed in one run is 50.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

For full details on Response Spectrum refer to section 5.32.10.1 Response Spectrum Specification.

5.32.10.2 Application of Time Varying Load for Response History Analysis

Used to apply loads which are defined with a changing magnitude of acceleration, force, or moment (See "Definition of Time History Load" on page 507). These can either be assigned to specific nodes and/or globally to the all the supports on the model as a ground motion. In addition to the data produced from a general modal analysis, a load case which includes time history loading will produce a set of graphs in the post processing mode to indicate how the displacement, velocity or acceleration of a selected node changes in each of the three global directions over the time period of the applied dynamic loading. set of commands may be used to model Time History loading on the structure for Response Time History analysis. Nodal time histories and ground motion time histories may both be provided under one load case.

Notes

- a. A Time History analysis requires the mode shapes. These are calculated using the mass matrix determined from the loading specified in the first dynamic load case. (See "Modal Calculation Command" on page 671)
- b. The Node Displacement table reports the maximum displacement that occurs at each node over the entire time range.
- c. The displacement of the model at a specific time instance can be displayed by using the time slider bar on the Results toolbar when displaying a load case with time history loading.
- d. If any Node Groups are defined, the time history graphs can be set to display the average results for the group. The name of the selected node or group being displayed is given in the graph title bar.
- e. A model can include only one load case with time history loads.

The following set of commands may be used to model Time History loading on the structure for Response Time History analysis. Nodal time histories and ground motion time histories may both be provided within one load case.

General Format

TIME LOAD

joint-list *{ FX | FY | FZ | MX | MY | MZ } I_t I_a f_2

GROUND MOTION { **ABS** | **(REL)** } { **X** | **Y** | **Z** } I_t I_a f_2

Where:

ABS = nodal results are absolute (elastic response + motion of ground). If entered on any ground command, all results will be absolute.

REL = nodal results are relative (elastic response). (Default)

I_t = sequential position in the input data of type number of time varying load. To refer to first type number entered, use a 1 here regardless of actual type number entered. Ground Motion must have an Acceleration Type; Time Load forces must have a Force type; and Time Load moments must have a Moment Type (see Section 5.31.4).

I_a = arrival time number (see Section 5.31.4) (integer). This is the sequential number of the arrival time in the list explained in section 5.31.4. Thus the arrival time number of a_3 is 3 and of a_n is n.

f_2 = The Force, Moment, or Acceleration Amplitude at this joint and direction will be multiplied by this factor (default = 1.0). For accelerations, if the amplitude-time curve was in g's, please use the Scale Factor in the Define Time History command to convert g's to the acceleration units used in that command. This is recommended due to possible unit changes between that command and this command.

Note: Multiple loads at a joint-direction pair for a particular (I_t I_a) pair will be summed. However there can be no more than four (I_t I_a) pairs associated with a particular joint-direction pair, the first four such entries will be used. Loads at slave joint directions will be ignored.

Either **TIME LOAD** or **GROUND MOTION** or both may be specified under one load case. More than one load case for time history analysis is not permitted.

For **TIME LOAD** data, multiple direction specifiers can be in one entry as follows (the direction specifiers must be on one line and missing values are assumed to be 1):

TIME LOAD

```
2 3 FX 1 FZ 1 4 -2.1 MX 2 2
6 7 FX FY FZ
```

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Example

```
LOAD 1
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
MEMBER LOADS
5 CON GX 7.5 10.0
5 CON GY 7.5 10.0
5 CON GZ 7.5 10.0
TIME LOAD
2 3 FX 1 3
5 7 FX 1 6
GROUND MOTION REL X 2 1
```

In the above example, the permanent masses in the structure are provided in the form of selfweight and member loads (see sections 5.32 and 1.18.3) for obtaining the mode shapes and frequencies. The mass model can also be created using a Reference Load case with the Loadtype of Mass (See "Mass Modeling Using Reference Loads" on page 533). The rest of the data is the input for application of the time varying loads on the structure. Forcing function type 1 is applied at joints 2 and 3 starting at arrival time number 3. (Arrival time number 3 is 1.8 seconds in example shown in section 5.31.4). Similarly, forcing function type 1 is applied at joints 5 and 7 starting at arrival time number 6 (4.4 seconds). A ground motion (type 2) acts on the structure in the x-direction starting at arrival time number 1 (0.0 seconds).

5.32.10.3 Floor Spectrum Response

The following commands have been added in order to allow the response spectrum of floors to be extracted from a time history analysis. The data generated consists of frequency versus acceleration pairs.

This command is used to specify the calculation of floor and/or joint spectra from time history results. The Floor Response Spectrum command must immediately follow an analysis command. That analysis can only contain a single time history load case.

General Format

The format of the Floor Spectrum command is such that 3 sets of data are required thus:

1. Initiate command

GENERATE FLOOR SPECTRUM

2. Specify Floor Groups

Next, identify the floors which will have spectrum curves generated either by referencing a NODE GROUP (see section 5.16) or explicitly listing the list of nodes that constitute the floor and the direction.

**BEGIN FLOOR DIRECTION {GX | GY | GZ} (*title*)
{*joint-group* | *joint-list*}**

Where:

GX, GY and GZ specify up to three global directions for which *frequency versus acceleration* spectrums will be generated for this floor.

title is an optional description of up to 50 characters, for this floor that will be displayed on the graphs in post processing.

The above can be repeated as many lines as necessary to specify all of the groups and directions that are needed.

3. Specify Options

Next enter additional parameters used in the Floor Spectrum calculations.

**OPTIONS { FLOW f1 | FHIGH f2 | FDELTA f3 | DAMP f4 |
(RELATIVE) } (THPRINT i1) (SPRINT)**

This command may be continued over a number of lines by ending each line except the last with a hyphen.

Required parameters:

- f₁ = Lowest frequency to be in the calculated spectrum. **FLOW** should be greater than 0.01 Hz.
- f₂ = Highest frequency to be in the calculated spectrum.
- f₃ = The spectrum will be calculated at **FDELTA** intervals from **FLOW** to **FHIGH**

Section 5 Commands and Input Instructions

5.32 Loading Specifications

- f_4 = Up to 10 damping values may be entered. One spectrum will be generated for each damping value for each global direction requested for each floor defined. The spectrum will be based on these modal damping ratios. 3% damping should be entered as 0.03. The default is 0.05.
- **RELATIVE**: If there is ground motion defined and you want the spectrums to be based on the relative acceleration of the floor to the ground acceleration, then include the **RELATIVE** parameter value.

Optional parameters:

- $i_1 = 0$, no print; $=2$, Print the time history acceleration being used in each spectrum calculation.
 - **SPRINT**: Print the calculated spectrum.
4. End the command

Finally, the command is completed with the line:

END FLOOR SPECTRUM

A successful analysis will result in the output file containing the base shear in the global X, Y and Z directions.

Example

```
DEFINE TIME HISTORY
TYPE 1 FORCE
0 -20 0.5 100 1 200 1.5 500 2 800 2.5 500 3 70 16 0
ARRIVAL TIME
0
DAMPING 0.075
*
LOAD 1 LOADTYPE SEISMIC TITLE TIME HISTORY LOAD CASE
* MASS MODEL REQUIRED
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
JOINT LOAD
```

```

1 TO 6 FX 62.223 FY 62.223 FZ 62.223
* TIME LOADS
TIME LOAD
2 FX 1 1
PERFORM ANALYSIS
*
GENERATE FLOOR SPECTRUM
BEGIN FLOOR DIRECTION GX GZ GROUND MOTION
_FL1
_FL17
BEGIN FLOOR DIRECTION GX GZ FLOOR 18 A/C UNIT 36
_FL18
OPTIONS FLO 0.5 FHI 35.0 FDEL 0.1-
DAMP 0.03 0.05 0.07 -
RELATIVE
END FLOOR SPECTRUM

```

5.32.11 Repeat Load Specification

This command is used to create a primary load case using combinations of previously defined primary load cases.

General Format

```

REPEAT LOAD
i1, f1, i2, f2 ... in, fn

```

Where:

$i_1, i_2 \dots i_n$ = primary load case numbers
 $f_1, f_2 \dots f_n$ = corresponding factors

This command can be continued to additional lines by ending all but last with a hyphen. Limit of 550 prior cases may be factored. Prior cases to be factored may also contain the **REPEAT LOAD** command.

Description

This command may be used to create a primary load case using combinations of previously defined primary load case(s). The **REPEAT LOAD** differs from the load **COMBINATION** command (See "Load Combination Specification" on page 672) in two ways:

1. A **REPEAT LOAD** is treated as a new primary load. Therefore, a P-Delta analysis will reflect correct secondary effects. (**LOAD COMBINATIONS**, on the other hand, algebraically combine the results such as displacements, member forces, reactions and stresses of previously defined primary loadings evaluated independently).
2. In addition to previously defined primary loads, you can also add new loading conditions within a load case in which the **REPEAT LOAD** is used.
3. The **REPEAT LOAD** option is available with load cases with **JOINT LOADS** and **MEMBER LOADS**. It can also be used on load cases with **ELEMENT PRESSURE** loads and **FIXED END LOADS**.

Modal dynamic analysis load cases (Response Spectrum, Time History, Steady State) should not be used in **REPEAT LOAD**. It is also not available for loads generated using some of the program's load generation facilities such as **MOVING LOAD** Generation. However load cases with **WIND LOAD** may be used in Repeat Load.

UBC cases may only be used in **REPEAT LOAD** if there is a **PERFORM ANALYSIS** and **CHANGE** command after each UBC case. See notes with UBC **LOAD** command.

Prestress on a given member from 2 or more load cases cannot be combined.

Example:

```
LOAD 1 DL + LL
SELFWEIGHT Y -1.4
MEMBER LOAD
1 TO 7 UNIFORM Y -3.5
LOAD 2 DL + LL + WL
REPEAT LOAD
1 1.10
```

4. For a load case that is defined using the **REPEAT LOAD** attribute, the constituent load cases themselves can also be **REPEAT LOAD** cases. See load case 4 below.

```

LOAD 1
SELFWEIGHT Y -1.0
LOAD 2
MEMBER LOAD
2 UNI GY -1.5
LOAD 3
REPEAT LOAD
1 1.5
LOAD 4
REPEAT LOAD
2 1.2 3 1.25

```

Notional Loads and Repeat Loads

Notional Loads (See "Notional Loads" on page 665) *cannot* be included in a **REPEAT LOAD** case. For example if the **LOAD 2** is a notional load, you cannot use it in **LOAD 3** using a **REPEAT LOAD**. The notional load will simply not be repeated in such case.

Incorrect syntax:

```

LOAD 2 LOADTYPE NONE TITLE TEST-NOTIONAL
NOTIONAL LOAD
LOAD 3 LOADTYPE NONE TITLE TEST-COMBO
REPEAT LOAD
2 1.0

```

Or

```

LOAD 1 : DEAD
JOINT LOAD

```

Section 5 Commands and Input Instructions

5.32 Loading Specifications

```
13 TO 16 29 TO 32 45 TO 48 61 TO 64 FY -100
NOTIONAL LOAD
1 X 0.002

LOAD 2 : IMPOSED
JOINT LOAD
13 TO 16 29 TO 32 45 TO 48 61 TO 64 FY -50
NOTIONAL LOAD
2 X 0.002

LOAD 3 : LOAD 1 + LOAD 2 + NOTIONAL LOADS
REPEAT LOAD
1 1.0 2 1.0
```

The correct syntax is:

```
LOAD 1 : DEAD
JOINT LOAD
13 TO 16 29 TO 32 45 TO 48 61 TO 64 FY -100
NOTIONAL LOAD
1 X 0.002

LOAD 2 : IMPOSED
JOINT LOAD
13 TO 16 29 TO 32 45 TO 48 61 TO 64 FY -50
NOTIONAL LOAD
2 X 0.002

LOAD 3 : LOAD 1 + LOAD 2 + NOTIONAL LOADS
REPEAT LOAD
1 1.0 2 1.0
NOTIONAL LOAD
```

```
1 X 0.002 2 X 0.002
```

5.32.12 Generation of Loads

This command is used to generate Moving Loads, UBC Seismic loads and Wind Loads using previously specified load definitions.

Primary load cases may be generated using previously defined load systems. The following sections describe generation of moving loads, UBC seismic loads and Wind Loads.

5.32.12.1 Generation of Moving Loads

This command is used to generate Moving Loads using previously specified load definitions.

Also see Sections 1.17 and 5.31.1

Pre-defined moving load system types may be used to generate the desired number of primary load cases, each representing a particular position of the moving load system on the structure. This procedure will simulate the movement of a vehicle in a specified direction on a specified plane on the structure.

General Format

```
LOAD GENERATION n (ADD LOAD i )
TYPE j x1 y1 z1 *{ XINC f1 | YINC f2 | ZINC f3 } ( { YRANGE
| ZRANGE } r)
```

Where:

n = total no. of primary load cases to be generated.

i = load case no. for the previously defined load case to be added to the generated loads.

j = type no. of previously defined load system.

x_1, y_1, z_1 = x, y and z coordinates (global) of the initial position of the reference load.

f_1, f_2, f_3 = x, y or z (global) increments of position of load system to be used for generation of subsequent load cases. Use only XINC & ZINC if Y up; Use only XINC & YINC if Z up.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

r = (Optional) defines section of the structure along global vertical direction to carry moving load. This r value is added and subtracted to the reference vertical coordinate (y_1 or z_1) in the global vertical direction to form a range. The moving load will be externally distributed among all members within the vertical range thus generated. r always should be a positive number. In other words, the program always looks for members lying in the range Y_1 and $Y_1+ABS(r)$ or Z_1 and $Z_1+ABS(r)$. The default r value is very small, so entering r is recommended.

The **ADD LOAD** specification may be used to add a previously defined load case to all the load cases generated by the **LOAD GENERATION** command. In the example below, the **SELFWEIGHT** specified in load case 1 is added to all the generated load cases.

Sequential load case numbers will be assigned to the series of generated primary load cases. Numbering will begin at one plus the immediate previous load case number. Allow for these when specifying load cases after load case generation.

Notes

- a. Primary load cases can be generated from Moving Load systems for frame members only. This feature does not work on finite elements.
- b. This facility works best when the roadway, as well as the movement of the vehicle are along one of the global horizontal (X or Z) or (X or Y) directions. For bridge decks which are skewed with respect to the global axes, the load generation may not yield the most satisfactory results. In such cases, the STAAD.Beava program, which is an add-on module to STAAD.Pro, is recommended. That program works on the influence line/influence surface method, and is considerably superior to the moving load generator described in this section. It also has the advantage of being able to calculate the critical load positions on decks modeled using plate elements, something which this facility cannot at present..
- c. The x_1 , y_1 , z_1 values of the starting position of the reference wheel must be provided bearing in mind that the reference wheel has to be at the elevation of the deck. An improper set of values of these parameters may result in the wheels being positioned incorrectly, and consequently, no load may be generated at all.

Example

```

LOAD 1 DL ONLY
SELFWEIGHT
LOAD GENERATION 20 ADD LOAD 1
TYPE 1 0. 5. 10. XI 10.
TYPE 2 0. 10. 10. ZI 15.
LOAD 22 LIVE LOAD ON PAVEMENT
MEMB LOAD
10 TO 20 30 TO 40 UNI GY -5.0
LOAD COMBINATION 31 10 0.75 22 0.75
PERFORM ANALYSIS

```

5.32.12.1.1 Generation of Seismic Loads

This command is used to generate UBC Seismic loads using previously specified load definitions.

5.32.12.1.2 Generation of UBC or IBC Seismic Loads

Built-in algorithms will automatically distribute the base shear among appropriate levels and the roof per the relevant code specifications. The following general format should be used to generate loads in a particular direction.

General Format

```

LOAD i
code LOAD { X | Y | Z } (f1) (DEC f2) (ACC f3)
code = { UBC | IBC | 1893 | AIJ | COL | CFE | NTC | RPA }

```

Where:

i = load case number

*f*₁ = factor to be used to multiply the UBC Load (default = 1.0). May be negative.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

f_2 = multiplying factor for Natural Torsion, arising due to static eccentricity which is the difference between center of mass and center of rigidity of a rigid floor diaphragm, to be used to multiply the UBC, IBC, 1893, etc. horizontal torsion load (default = 0.0). Must be a positive or zero.

f_3 = multiplying factor for Accidental Torsion, to be used to multiply the UBC, IBC, 1893, etc. accidental torsion load (default = 1.0). May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

Use only horizontal directions.

To include horizontal torsional moment arising due to static eccentricity for a rigid floor diaphragm following conditions must be satisfied.

- The floor must be modeled as a rigid diaphragm.
- A positive value for **DEC** must be provided. If it is 0.0 (zero), then the only accidental torsion will be considered for that particular seismic load case.
- The **ACC** command must not be present in seismic definition (i.e., in the **DEFINE code LOAD** command). If present, the natural torsion factor will be ignored and only the accidental torsion for all seismic loads will be considered.

The design eccentricity for calculating horizontal torsion is the **DEC + ACC** values. When ACC is negative, it becomes **DEC - ACC** (i.e., the torsion magnitudes are always additive).

Example

```
DEFINE UBC LOAD
ZONE 0.2 K 1.0 I 1.5 TS 0.5
SELFWEIGHT
JOINT WEIGHT
1 TO 100 WEIGHT 5.0
101 TO 200 WEIGHT 7.5
LOAD 1 UBC IN X-DIRECTION
UBC LOAD X DEC 1.0 ACC 0.05
JOINT LOAD
```

```

5 25 30 FY -17.5
PERFORM ANALYSIS
CHANGE
LOAD 2 UBC IN X-DIRECTION
UBC LOAD X DEC 1.0 ACC -0.05
JOINT LOAD
5 25 30 FY -17.5
PERFORM ANALYSIS
CHANGE
LOAD 3 UBC IN Z-DIRECTION
UBC LOAD Z DEC 0.0 ACC 0.05
PERFORM ANALYSIS
CHANGE
LOAD 3 DEAD LOAD
SELFWEIGHT
LOAD COMBINATION 4
1 0.75 2 0.75 3 1.0

```

In the above example, notice that the first three load cases are UBC load cases. They are specified before any other load cases.

Notes

- a. The UBC load cases should be provided as the first set of load cases. Non-UBC primary load case specified before a UBC load case is not acceptable. Additional loads such as MEMBER LOADS and JOINT LOADS may be specified along with the UBC load under the same load case.

Example of *Incorrect* Usage: The error here is that the UBC cases appear as the 3rd and 4th cases, when they should be the 1st and 2nd cases.

```

LOAD 1
SELFWEIGHT Y -1
LOAD 2

```

Section 5 Commands and Input Instructions

5.32 Loading Specifications

```
JOINT LOAD
3 FX 45
LOAD 3
UBC LOAD X 1.2
JOINT LOAD
3 FY -4.5
LOAD 4
UBC LOAD Z 1.2
MEMBER LOAD
3 UNI GY -4.5
PERFORM ANALYSIS
```

Example of Correct Usage

```
SET NL 10
LOAD 1
UBC LOAD X 1.2
JOINT LOAD
3 FY -4.5
PERFORM ANALYSIS
CHANGE
LOAD 2
UBC LOAD Z 1.2
MEMBER LOAD
3 UNI GY -4.5
PERFORM ANALYSIS
CHANGE
LOAD 3
SELFWEIGHT Y -1
LOAD 4
JOINT LOAD
```

```

3 FX 45
PERFORM ANALYSIS
LOAD LIST ALL

```

- b. If the UBC cases are to be factored later in a Repeat Load command; or if the UBC case is to be used in a tension/compression analysis; or if Re-analysis (two analysis commands without a CHANGE or new load case in between); then each UBC case should be followed by PERFORM ANALYSIS then CHANGE commands as shown in the example above. Otherwise the PERFORM ANALYSIS then CHANGE can be omitted. Using the CHANGE command will require the SET NL command to define the maximum number of load cases being entered in the analysis. Also LOAD LIST ALL should be entered after the last PERFORM ANALYSIS command.

Example of *Incorrect* Usage: The error here is that the **CHANGE** command is missing before Load Case 2.

```

LOAD 1
UBC LOAD X 1.2
SELFWEIGHT Y -1
JOINT LOAD
3 FY -4.5
PDELTA ANALYSIS
LOAD 2
UBC LOAD Z 1.2
SELFWEIGHT Y -1
JOINT LOAD
3 FY -4.5
PDELTA ANALYSIS

```

Example of Correct Usage

```

LOAD 1
UBC LOAD X 1.2
SELFWEIGHT Y -1

```

Section 5 Commands and Input Instructions

5.32 Loading Specifications

```
JOINT LOAD
3 FY -4.5
PDELTA ANALYSIS
CHANGE
LOAD 2
UBC LOAD Z 1.2
SELFWEIGHT Y -1
JOINT LOAD
3 FY -4.5
PDELTA ANALYSIS
CHANGE
```

- c. Up to 8 UBC cases may be entered.
- d. The **REPEAT LOAD** specification cannot be used for load cases involving UBC load generation unless each UBC case is followed by an analysis command then **CHANGE**.

Example

```
LOAD 1
UBC LOAD X 1.0
PDELTA ANALYSIS
CHANGE
LOAD 2
SELFWEIGHT Y -1
PDELTA ANALYSIS
CHANGE
LOAD 3
REPEAT LOAD
1 1.4 2 1.2
PDELTA ANALYSIS
```

- e. If UBC load generation is performed for the X and the Z (or Y if Z up) directions, the command for the X direction must precede the command for the Z (or Y if Z up) direction.

5.32.12.1.3 Generation of IS:1893 Seismic Load

The following general format should be used to generate the IS 1893 load in a particular direction.

General Format

LOAD *i*

1893 LOAD { X | Y | Z } (*f*₁) (ACC *f*₂)

where

i = load case number

*f*₁ = factor to be used to multiply the 1893 Load (default = 1.0)

*f*₂ = multiplying factor for Accidental Torsion, to be used to multiply the UBC, IBC, 1893, etc. accidental torsion load (default = 1.0). May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

Use only horizontal directions.

Example

In the above example, the first two load cases are the 1893 load cases. They are specified before any other load case.

```

DEFINE 1893 LOAD
ZONE 0.05 RF 1.0 I 1.5 SS 1.0
SELFWEIGHT
JOINT WEIGHT
7 TO 12 WEIGHT 17.5
13 TO 20 WEIGHT 18.0
MEMBER WEIGHT
1 TO 20 UNI 2.0

```

Section 5 Commands and Input Instructions

5.32 Loading Specifications

```
LOAD 1 1893 LOAD IN X-DIRECTION
1893 LOAD X
JOINT LOAD
5 25 30 FY -17.5
LOAD 2 1893 LOAD IN Z-DIRECTION
1893 LOAD Z
LOAD 3 DEAD LOAD
SELFWEIGHT
LOAD COMBINATION 4
1 0.75 2 0.75 3 10
```

5.32.12.2 Generation of Wind Loads

This command is used to generate Wind Loads using previously specified load definitions.

The built-in wind load generation facility can be used to calculate the wind loads based on the parameters defined in Section 5.31.3. The following general format should be used to perform the wind load generation. See Section 1.17.3 for the two types of structures on which the load can be generated. For closed type structures, the vertical panel areas bounded by beam members only (and ground), and exposed to the wind, are used to define loaded areas (plates and solids are ignored). The loads generated are applied only at the joints at vertices of the bounded areas. For open type structures also, generation is done considering only the members in the model.

The automated load generator should only be used for vertical panels. Panels not parallel to the global Y axis (for Y UP) should be loaded separately.

General Format

```
LOAD i
WIND LOAD (-){ X | Y | Z } (f) TYPE j (OPEN) { XR f1 f2 | YR
f1 f2 | ZR f1 f2 | LIST memb-list | ALL }
```

Where:

i = Load case number

X, -X, Z or -Z = Direction of wind in global axis system. Use horizontal directions only.

j = Type number of previously defined systems;

f = The factor to be used to multiply the wind loads. Negative signs may be used to indicate opposite direction of resulting load (default=1.0)

f_1, f_2 = global coordinate values to specify X or Y or Z range for member selection.

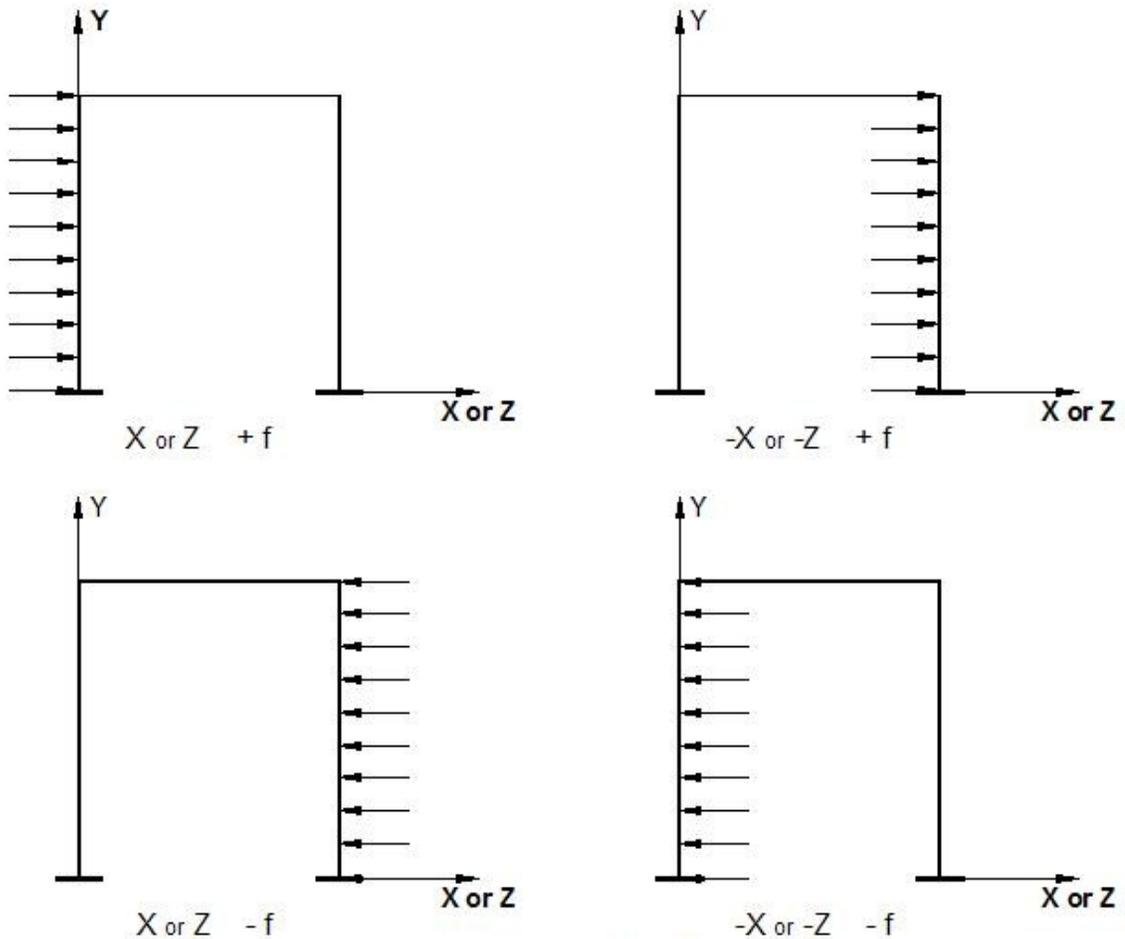
OPEN optional word to be used if loading is to be generated on open-type of structures. If this not specified, load will be generated assuming the panels are “closed”.

Using X, -X, Z or -Z and the f factor. With respect to the axis, a minus sign indicates that suction occurs on the other side of the selected structure. If all of the members are selected and X (or Z) is used and the factor is positive, then the exposed surfaces facing in the -x (or -z) direction will be loaded in the positive x (or z) direction (normal wind in positive direction). See diagrams that follow. If X and a negative factor is used, then the exposed surfaces facing in the +x direction will be loaded in the negative x direction (normal wind in negative direction). [If -X is entered and a negative factor, then the exposed surfaces facing in the -x direction will be loaded in the negative x direction (suction). If -X is entered and a positive factor, then the exposed surfaces facing in the +x direction will be loaded in the positive x direction (suction).]

Figure 5-53: Sign convention for internal and external pressure

Section 5 Commands and Input Instructions

5.32 Loading Specifications



A member list or a range of coordinate values (in global system) may be used. All members which have both end coordinates within the range are assumed to be candidates (for closed type structures) for defining a surface which may be loaded if the surface is exposed to the wind. The loading will be in the form of joint loads (not member loads). 1, 2 or 3 ranges can be entered to form a “layer”, “tube” or “box” for selecting members in the combined ranges. Use ranges to speed up the calculations on larger models.

Example

```
DEFINE WIND LOAD
TYPE 1
INTENSITY 0.1 0.12 HEIGHT 100 200
EXP 0.6 JOI 1 TO 25 BY 7 29 TO 37 BY 4 22 23
TYPE 2
```

```

INT 0.1 0.12 HEIGHT 100 900
EXP 0.3 YR 0 500
LOAD 1
SELF Y -1.0
LOAD 2
WIND LOAD Z 1.2 TYPE 2 ZR 10 11
LOAD 3
WIND LOAD X TYPE 1 XR 7 8 ZR 14 16
LOAD 4 SUCTION ON LEEWARD SIDE
WIND LOAD -X 1.2 LIST 21 22 42

```

Example

For open structures

```

LOAD 1 WIND LOAD IN Z DIRECTION
WIND LOAD 2 -1.2 TYPE 1 OPEN

```

Notes

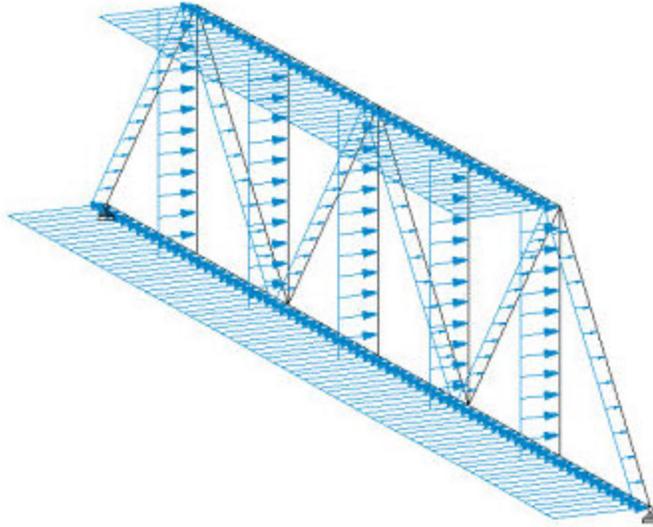
- a. For closed type structures, panels or closed surfaces are generated by the program based on the members in the ranges specified and their end joints. The area within each closed surface is determined and the share of this area (influence area) for each node in the list is then calculated. The individual bounded areas must be planar surfaces, to a close tolerance, or they will not be loaded.

Hence, one should make sure that the members/joints that are exposed to the wind make up a closed surface (ground may form an edge of the closed surface). Without a proper closed surface, the area calculated for the region may be indeterminate and the joint force values may be erroneous. Consequently, the number of exposed joints should be at least three.
- b. Plates and solids are not considered for wind load generation. On such entities, wind must be applied using pressure loading facilities for plates and solids.

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Figure 5-54: Load diagram for wind on open structures



5.32.13 Generation of Snow Loads

This command is used to generate Snow Loads using previously specified Snow load definitions. This input should be a part of a load case.

General Format

SNOW LOAD

```
_flr_group TYPE j CS f1 { BALA | UNBA } { OBST | UNOB }  
{ MONO | HIP | GABLE }
```

Where:

_flr_group = The members that form the roof and that are to be loaded by snow load must be listed in a floor group (See section 5.16).

j = type no. of previously defined snow load system.

f1 = Roof Slope factor (CS). Default = 0.0. For sloped roofs, the roof slope factor is described in section 7.4 of the SEI/ASCE-7-02. A value of 0 indicates that the roof is horizontal.

BALA or UNBA = Balanced or unbalanced snow load. Default is balanced. These terms are described in section 7.6, and figures 7.3 and 7.5 of ASCE 7-02.

OBST or UNOB = Obstructed or unobstructed. Default is unobstructed.

MONO or HIP or GABLE= Roof type. Default is MONO.

Use as many floor groups and types as necessary in each load case.

5.32.14 Notional Loads

A notional load is a lateral load (horizontal load) which is derived from an existing vertical load case. This load type has been introduced to accommodate a requirement in design codes. The AISC 360-05 specification for example defines notional loads as lateral loads that are applied at each framing level and are specified in terms of gravity loads.

Description

Both Principal and Reference load cases can be selected and moved into the Notional Load Definition where the required factor and direction can be specified. The notional loads are calculated and applied as joint loads.

Note: The actual values of the applied loading are not displayed in the User Interface until after the analysis has been performed.

General Format

LOAD nLoad LOADTYPE type TITLE title

...

Load items

...

NOTIONAL LOAD

n { X | Z } (f_1)

Where:

n = Load case number of the primary load case or reference load case which contains the vertical load items.

f_1 = factor by which the contents of LN are to be multiplied. Typically, codes recommend 0.2 to 0.3 % (0.002 to 0.003). The default is 0.002 or the factor specified in **DIRECT ANALYSIS** definition block. .

Section 5 Commands and Input Instructions

5.32 Loading Specifications

Multiple load items can be specified under any **NOTIONAL LOAD** and there can be one or more blocks of **NOTIONAL LOAD** command(s) in any load case.

Note: Notional Loads can be derived from gravity load cases specified under the **DEFINE REFERENCE LOAD** table which are described in section 5.31.6

Example 1

```
LOAD 1 : DEAD
JOINT LOAD
13 TO 16 29 TO 32 45 TO 48 61 TO 64 FY -100
LOAD 2: DEAD NOTIONAL LOAD
NOTIONAL LOAD
1 X 0.002
...
LOAD 10: IMPOSED
JOINT LOAD
13 TO 16 29 TO 32 45 TO 48 61 TO 64 FY -50
LOAD 11:IMPOSED NOTIONAL LOAD
NOTIONAL LOAD
10 X 0.002
```

Example 2

If we want to combine the load cases in the example shown above, the correct syntax would be

```
LOAD 3 : LOAD 1 + LOAD 2 + NOTIONAL LOADS
REPEAT LOAD
1 1.0 2 1.0
NOTIONAL LOAD
1 X 0.002 2 X 0.002
```

The following syntax would not include the contributions of Notional loads for each component cases as Notional Loads are not going to be repeated.

```
LOAD 3 : LOAD 1 + LOAD 2 + NOTIONAL LOADS
REPEAT LOAD
1 1.0 2 1.0
```

Example 3

Similarly,

```
LOAD 1 LOADTYPE NONE TITLE TEST
SELFWEIGHT Y -1.15
MEMBER LOAD
143 145 CON GY -16
502 515 CON GY -95
LOAD 2 LOADTYPE NONE TITLE TEST-NOTIONAL
NOTIONAL LOAD
1 X 1.0
LOAD 3 LOADTYPE NONE TITLE TEST-COMBO
REPEAT LOAD
2 1.0
```

Load case 3 in the above example would produce no loads.

Example 4

Using Notional loads with **REFERENCE** loads

```
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE DEAD TITLE REF DEAD LOAD
SELFWEIGHT Y -1 LIST 1 TO 120
FLOOR LOAD
YRANGE 19 21 FLOAD -0.05 GY
YRANGE 39 41 FLOAD -0.05 GY
```

Section 5 Commands and Input Instructions

5.32 Loading Specifications

```
LOAD R2 LOADTYPE LIVE TITLE REF LIVE LOAD
FLOOR LOAD
YRANGE 19 21 FLOAD -0.03 GY
YRANGE 39 41 FLOAD -0.03 GY
END DEFINE REFERENCE LOADS
*****
* LOAD COMBINATIONS INCLUDING NOTIONAL LOADS *
*****
*
*DEAD + LIVE +/- NOTIONAL LOAD
*
LOAD 1
REFERENCE LOAD
R1 1.2 R2 1.6
NOTIONAL LOAD
R1 X 0.0024 R2 X 0.0032
LOAD 2
REFERENCE LOAD
R1 1.2 R2 1.6
NOTIONAL LOAD
R1 X -0.0024 R2 X -0.0032
LOAD 3
REFERENCE LOAD
R1 1.2 R2 1.6
NOTIONAL LOAD
R1 Z 0.0024 R2 Z 0.0032
LOAD 4
REFERENCE LOAD
R1 1.2 R2 1.6
NOTIONAL LOAD
R1 Z -0.0024 R2 Z -0.0032
```

If a **PRINT STATIC CHECK** or **PRINT LOAD DATA** command is added with any analysis specification in the input file, the generated NOTIONAL LOADS would be printed as in the following:

```

NOTIONAL LOAD - (POUN,FEET)
LOADING      4  DL+NDLX
JOINT        DIRECTION          LOAD
...
6117         X   -0.00200 X      74124.266 =    -148.249
6118         X   -0.00200 X     115504.569 =    -231.009
6119         X   -0.00200 X      38397.157 =     -76.794
6120         X   -0.00200 X      15551.155 =     -31.102
6121         X   -0.00200 X      17347.718 =     -34.695
6122         X   -0.00200 X      29341.723 =     -58.683
6123         X   -0.00200 X      39771.147 =     -79.542
6124         X   -0.00200 X      39712.750 =     -79.426
6125         X   -0.00200 X      34006.254 =     -68.013
=====
48554642.872   -97109.290

```

5.33 Reference Load Cases - Application

Describes how you can call the data specified under those types in actual load cases. Reference Load types are described in section 5.31.6 of this manual.

General Format

The format of a reference to a Reference Load in a primary load (j) case is thus:

```

LOAD j LOADTYPE (type) load_title
REFERENCE LOAD
R(i) 1.0

```

...

Example

```

LOAD 1 LOADTYPE NONE TITLE D+L
REFERENCE LOAD
R1 1.0 R2 1.0
LOAD 2 LOADTYPE NONE TITLE DEAD+SNOW
REFERENCE LOAD

```

Section 5 Commands and Input Instructions

5.34 Frequency Calculation

```
R1 1.0 R3 1.0
LOAD 3 LOADTYPE NONE TITLE D+H
REFERENCE LOAD
R1 1.0 R4 1.0
ELEMENT LOAD
1212 1267 TRAP GY JT -0.54 -0.44 -0.44 -0.54.
```

5.34 Frequency Calculation

There are two methods available in STAAD for calculating the frequencies of a structure – 1) an approximate method called the Rayleigh method and 2) a more exact method which involves the solution of an eigenvalue problem.

Both methods are explained in the following sections.

5.34.1 Rayleigh Frequency Calculation

This command may be used to calculate the Rayleigh method approximate frequency of the structure for vibration corresponding to the general direction of deflection generated by the load case that precedes this command. Thus, this command typically follows a load case.

General Format

CALCULATE RAYLEIGH (FREQUENCY)

Description

This command is specified after all other load specifications of any primary load case for which the Rayleigh frequency is calculated. This Rayleigh frequency calculation is based on the Rayleigh iteration method using 1 iteration. If a more accurate, full-scale eigensolution is required, the **MODAL CALCULATION** command (see next section) may be used. A full eigensolution is automatically performed if a **RESPONSE SPECTRUM** or **TIME HISTORY** is specified in any load case.

Example

```
LOADING 1
SELFWEIGHT X 1.0
```

```

CALCULATE RAYLEIGH FREQUENCY
LOADING 2
SELFWEIGHT Z 1.0
CALCULATE RAYLEIGH FREQUENCY
LOADING 3 WIND LOAD

```

In this example, the Rayleigh frequency is calculated for the X direction mode of vibration in load case 1, and the Z direction in case 2.

In case 1, the structure is being displaced by **SELFWEIGHT** applied statically along global X. For most frames which are supported only at their base, this produces a deflected shape resembling the lowest mode shape along global X. Hence, this frequency will resemble that for the lowest X direction mode.

Similarly, the frequency calculated for load case 2 ought to be similar to that for the lowest Z direction mode because the selfweight is applied statically along global Z.

Hint: If you want a mode or frequency to be for lateral motion (X or Z), then enter the loads in the X or Z directions. Entering loads in the vertical direction when the lowest modes are in the lateral direction is a common mistake.

The output will consist of the value of the frequency in cycles per second (cps), the maximum deflection along with that global direction and the joint number where that maximum occurs.

Notes

This command is based on the Rayleigh method of iteration using 1 iteration. The frequency calculated estimates the frequency as if the structure were constrained to vibrate in the static deflected shape generated by the loads in the load case.

In many instances, the forces should be in one global direction to get the mode and frequency associated with that direction.

5.34.2 Modal Calculation Command

This command may be used to obtain a full scale eigensolution to calculate relevant frequencies and mode shapes. It should not be entered if this case or any other case is a **TIME LOAD** or **RESPONSE SPECTRUM** case. For Steady

Section 5 Commands and Input Instructions

5.35 Load Combination Specification

State/Harmonic analysis this command must be included in the load case that defines the weights and weight moment of inertias for eigensolutions (see sections 1.18.3 and 5.32).

General Format

MODAL (CALCULATION REQUESTED)

This command is typically used in a load case after all loads are specified. The loads will be treated as weights and weight moment of inertias for eigensolutions (see sections 1.18.3 and 5.32). You are advised to specify the loads keeping this in mind.

This case will be independently solved statically and dynamically. Static results using the loads will include joint displacements, member forces, support reactions, and other outputs computed from a normal static analysis without any dynamic effects included.

STAAD.Pro can include the stress stiffening effect (geometric stiffness) based on the axial member forces/plate in-plane stresses from a selected load case when calculating the modes & frequencies of a structure.

In addition, the Dynamic results (using loads as masses) will include Mode Shapes and Frequencies.

Note: The **MODAL CALCULATION** command can be included in any of the primary load cases, but only in one of them.

5.35 Load Combination Specification

This command may be used to combine the results of the analysis. The structure will *not* be analyzed for the combined loading. The combination of results may be algebraic, SRSS, a combination of both algebraic and SRSS, or Absolute.

Note: The **LOAD COMBINATION** specification is not appropriate for obtaining secondary effects for combined loads. Refer to notes below.

General Format

LOAD COMBINATION ({ **SRSS** | **ABS** }) **i** **a**₁
i₁ **f**₁ **i**₂ **f**₂ ... **f**_{srss}

Where:

i = Load combination number. This can be any integer smaller than 100,000 that is not the same as any previously defined primary load case or load combination number.

a_i = Any title for the load combination.

i_1, i_2, \dots = represents the load case or load combination numbers which are to be combined.

f_1, f_2, \dots = represents corresponding factors to be applied to loadings. A value of 0.0 is also allowed.

f_{SRSS} = optional factor to be applied as a multiplying factor on the combined result of the SRSS load combination (see examples below).

A hyphen may be used at the end of a line to continue this command onto the next line.

Description

If the load combination option is left out (i.e., neither **ABS** or **SRSS** is included), then the results from analyses will be combined algebraically.

```
LOAD COMBINATION 6 DL+LL+WL
```

```
1 0.75 2 0.75 3 1.33
```

LOAD COMBINATION ABS

If the **ABS** load combination method is included, then the absolute value of results from the analyses will be combined.

```
LOAD COMBINATION ABS 7 DL+LL+WL
```

```
1 0.85 2 0.65 3 2.12
```

If the **SRSS** load combination method is included, then the results from analyses may be combined both algebraically and using the SRSS (Square Root of Summation of Squares) method. The combination scheme may be mixed if required. For example, in the same load combination case, results from load cases may be combined in the SRSS manner and then combined algebraically with other load cases.

Note: The case factor is not squared.

Section 5 Commands and Input Instructions

5.35 Load Combination Specification

If some of the factors are negative, then the sum of the factored squares may become negative. If negative, the SQRT will be of the absolute value and the result of the SQRT will be set negative.

Example of Simple SRSS Combination

Several combination examples are provided to illustrate the possible combination schemes

```
LOAD COMBINATION SRSS 8 DL+SEISMIC
```

```
1 1.0 2 0.4 3 0.4
```

This (LOAD COMBINATION SRSS 8) illustrates a pure SRSS load combination with a default SRSS factor of one. The following combination scheme will be used:

$$v = 1.0 \cdot (1 \cdot L1^2 + 0.4 \cdot L2^2 + 0.4 \cdot L3^2)^{1/2}$$

where

v = the combined value

L1, L2, and L3 = values from load cases 1, 2 and 3.

Since an SRSS factor is not provided, the default value of 1.0 is being used.

Notes

- In the **LOAD COMBINATION SRSS** option, if the minus sign precedes any load case no., then that load case will be combined algebraically with the SRSS combination of the rest.
- If the secondary effects of combined load cases are to be obtained through a P-Delta, Member/Spring Tension/Compression, Multilinear Spring, or Nonlinear Analysis, then the **LOAD COMBINATION** command is inappropriate for the purpose. See the **REPEAT LOAD** command (section 5.32.11) for details.
- In a load combination specification, a value of 0 (zero) as a load factor is permitted. In other words, a specification such as

```
LOAD COMB 7
```

```
1 1.35 2 0.0 3 1.2 4 0.0 5 1.7
```

is permitted. This is the same as

```
LOAD COMB 7
```

1 1.35 3 1.2 5 1.7

- d. All combination load cases must be provided immediately after the last primary load case.
- e. The total number of primary and combination load cases combined cannot exceed the limit described in section 5.2 of this manual.
- f. A maximum of 550 load cases can be combined using a **LOAD COMBINATION** command.
- g. Load combinations can refer to previously defined load combination numbers.

Note: This feature requires build 20.07.07.30 or higher.

Examples of Algebraic & SRSS Combination in the Same Load Combination Case

Example 1

```
LOAD COMBINATION SRSS 9
-1 0.75 2 1.3 3 2.42 0.75
```

The combination formula will be as follows:

$$v = 0.75 \cdot L1 + 0.75 \cdot (1.3 \cdot L2^2 + 2.42 \cdot L3^2)^{1/2}$$

where

v = combined value

L2 & L3 = values from load cases 2 & 3.

In the above specification, a minus sign precedes load case 1. Thus, Load 1 is combined algebraically with the result obtained from combining load cases 2 and 3 in the SRSS manner. The SRSS factor of 0.75 is applied on the SRSS combination of 2 and 3.

Example 2

```
LOAD COMBINATION SRSS 10
-1 0.75 -2 0.572 3 1.2 4 1.7 0.63
```

Section 5 Commands and Input Instructions

5.36 Calculation of Problem Statistics

Here, both load cases 1 and 2 are combined algebraically with the SRSS combination of load cases 3 and 4. Note the SRSS factor of 0.63. The combination formula will be as follows:

$$v = 0.75 \cdot L1 + 0.572 \cdot L2 + 0.63 \cdot (1.2 \cdot L3^2 + 1.7 \cdot L4^2)^{1/2}$$

5.36 Calculation of Problem Statistics

This item has been removed. Please contact the Bentley Technical Support Group for further details.

5.37 Analysis Specification

STAAD analysis options include linear static analysis, P-Delta (or second order analysis), Nonlinear analysis, and several types of Dynamic analysis.

This command is used to specify the analysis request. In addition, this command may be used to request that various analysis related data, like load info, statics check info, etc. be printed.

5.37.1 Linear Elastic Analysis

Used to perform a static, linear elastic analysis on the structure.

General Format

```
PERFORM ANALYSIS (PRINT { LOAD DATA | STATICS CHECK  
| STATICS LOAD | BOTH | ALL } )
```

Without one of these analysis commands, no analysis will be performed. These **ANALYSIS** commands can be repeated if multiple analyses are needed at different phases.

The following **PRINT** options are available:

- If the **PRINT LOAD DATA** command is specified, the program will print an interpretation of all the load data.
- **PRINT STATICS CHECK** will provide a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.
- **PRINT STATICS LOAD** prints everything that **PRINT STATICS CHECK** does, plus it prints a summation of all internal and external forces at each joint (generates voluminous output).

- **PRINT BOTH** is equivalent to **PRINT LOAD DATA** plus **PRINT STATICS CHECK**.
- **PRINT ALL** is equivalent to **PRINT LOAD DATA** plus **PRINT STATICS LOAD**.
- The **PRINT MODE SHAPES** command may be added separately after the Analysis command if mode shapes are desired. Refer to section 5.42 for using additional Print specifications.

Notes

This command directs the program to perform the analysis that includes:

- a. Checking whether all information is provided for the analysis;
- b. Forming the joint stiffness matrix;
- c. Checking the stability of the structure;
- d. Solving simultaneous equations, and
- e. Computing the member forces and displacements.
- f. If a **RESPONSE SPECTRUM**, **TIME LOAD**, or **GROUND MOTION** is specified within a load case or the **MODAL CALCULATION** command is used, a dynamic analysis is performed.

General Analysis Comments

STAAD allows multiple analyses in the same run. Multiple analyses may be used for the following purposes:

- a. Successive analysis and design cycles in the same run result in optimized design. STAAD automatically updates changes in member cross-sectional sizes. Thus the entire process is automated.

Refer to Example 1 in the Getting Started & Examples manual for detailed illustration.

- b. Multiple analyses may be used for load-dependent structures. For example, structures with bracing members are analyzed in several steps. The bracing members are assumed to take Tension load only. Thus, they need to be activated and inactivated based on the direction of lateral loading.

The entire process can be modeled in one STAAD run using multiple **PERFORM ANALYSIS** commands. STAAD is capable of performing a design based on the load combinations provided.

Section 5 Commands and Input Instructions

5.37 Analysis Specification

Refer to Example 4 in the Getting Started & Examples manual for detailed illustration.

- c. You may also use Multiple Analyses to model change in other characteristics like supports, releases, section properties, etc.
- d. Multiple Analyses may require use of additional commands like the **SET NL** command (section 5.5) and the **CHANGE** command.
- e. Analysis and **CHANGE** are required after UBC cases if the case is subsequently referred to in a Repeat Load command or if the UBC case will be re-solved after a Select command or after a Multiple analysis.

5.37.2 P-Delta Analysis Options

Used to perform a second-order analysis, considering either large- or small-delta effects—or both—on the structure.

General Format

The following options are available in STAAD.Pro when performing a P-Delta analysis:

Note: Options 1 and 2 in the present form are effective from STAAD.Pro 2007 Build 06.

1. P-Delta analysis with Small & Large Delta effects or Large Delta effects only. With this option the global forces are adjusted with every iteration

```
PDELTA (n) ANALYSIS (CONVERGE (m)) ( { LARGEDELTA |  
SMALLDELTA } ) (PRINT print-options)
```

Where:

n = no. of iterations desired (default value of n = 1).

m = the maximum number of iterations used to check convergence, even if convergence has not been achieved. Do *not* use **CONVERGE** when n is specified.

SMALLDELTA is the default

2. P-Delta analysis including stress stiffening effect of the KG matrix. With this option the global stiffness is adjusted with every iteration

```
PDELTA KG ANALYSIS (PRINT print-options)
```

print-options = { LOAD DATA | STATICS CHECK | STATICS LOAD
| BOTH | ALL }

See section 5.37.1 for details.

Without one of these P-Delta analysis commands, no P-Delta analysis will be performed.

These **ANALYSIS** commands can be repeated if multiple analyses are needed at different phases.

A **PDELTA ANALYSIS** will correctly reflect the secondary effects of a combination of load cases only if they are defined using the **REPEAT LOAD** specification (section 5.32.11). Secondary effects will not be evaluated correctly for load combinations.

P-Delta effects are computed for frame members and plate elements only. They are not calculated for solid elements or curved beams.

Notes for Small or Large Delta (Option 1)

- a. This command directs the program to perform the analysis that includes:
 1. Checking whether all information is provided for the analysis;
 2. Forming the joint stiffness matrix
 3. Checking the stability of the structure;
 4. Solving simultaneous equations, and
 5. Computing the member forces and displacements.
 6. For P-Delta analysis, forces and displacements are recalculated, taking into consideration the chosen P-Delta effect.
 7. If a **RESPONSE SPECTRUM**, **TIME LOAD**, or **GROUND MOTION** is specified within a load case or the **MODAL CALCULATION** command is used, a dynamic analysis is performed.

Note: Computing P-Delta effects for dynamic load cases is not recommended since such effects are not considered.

8. In each of the iterations of the **PDELTA ANALYSIS**, the load vector will be modified to include the secondary effect generated by the displacements caused by the previous iterations.
- b. The default procedure of option 1 is based on “P-small & large Delta” effects. (sometimes referred to as P- δ & P- Δ). Enter the LargeDelta parameter to

Section 5 Commands and Input Instructions

5.37 Analysis Specification

only include the “PlargeDelta” effects (P- Δ only). SmallDelta is recommended.

- c. This **PDELTA (n) ANALYSIS** command (option 1) should specify 3 to 30 iterations to properly incorporate the P-Delta effect. With this many iterations, the **PDELTA (n) ANALYSIS SMALLDELTA** command results are as good as or better than the **PDELTA KG** (option 2) command results for static analysis. The advantage of this **PDELTA (n) ANALYSIS** command comes from not having to re-form and then triangular factorize the stiffness matrix for every iteration within every case. Also this command allows tension/compression.
- d. Be aware that global buckling can occur in P-Delta analysis, resulting in large or infinite or NaN values for displacement. Do not use the results of such a case. Sometimes the loads from Repeat Load combination cases are too large; sometimes partial moment releases rather than the full release is needed, sometimes connectivity needs to be corrected. Always check the maximum displacements for P-Delta analyses.
- e. When the **CONVERGE** command is not specified, the member end forces are evaluated by iterating **n** times (once, if **n** is not specified).

Warning: Do *not* enter **n** when **CONVERGE** is provided.

- f. When the **CONVERGE** command is included, the member end forces are evaluated by performing a convergence check on the joint displacements. In each step, the displacements are compared with those of the previous iteration in order to check whether convergence is attained based on the convergence displacement tolerance. In case **m** is specified, the analysis will stop after that iteration even if convergence has not been achieved. If convergence is achieved in less than **m** iterations, the analysis is terminated.

Note: The convergence option is *not* recommended.

To set the convergence displacement tolerance, enter one of the following:

- **SET DISPLACEMENT i_2** command, where i_2 = a target displacement value in current units. The default value is the maximum span of the structure divided by 120.
- **SET PDELTA TOL i_9** command, where i_9 = a displacement threshold for convergence in current units of length. The default value is 0.01

inch. If the maximum change in displacement from two consecutive iterations is less than $ftol$, then that load case is converged.

Refer to section 5.5 for additional information.

Example

Following are some examples on use of the command for P-Delta analysis as described in option 1.

```
PDELTA ANALYSIS
PDELTA 5 ANALYSIS
PDELTA ANALYSIS CONVERGE
PDELTA ANALYSIS CONVERGE 5
PDELTA 20 ANALYSIS SMALLDELTA PRINT STATICS CHECK
```

STAAD allows multiple P-Delta analyses in the same run (see the General Comments section of 5.37.1 for details).

Notes for Stress Stiffening Matrix (Option 2)

The P-Delta analysis also provides the option of including the stress stiffening effect of the K_g matrix into the member / plate stiffness.

- a. A regular STAAD P-Delta Analysis (option 1) performs a first order linear analysis and obtains a set of joint forces from member/plates based on both large and small P-Delta effects. In contrast, the P-Delta KG Analysis (that is, with the K_g option selected) includes the effect of the axial stress after the first analysis is used to modify the stiffness of the member/plates. A second analysis is then performed using the original load vector. Large & small P-Delta effects are always included in this KG option.
- b. This command directs the program to perform the analysis that includes:
 1. Solving the static case.
 2. Re-forming the global joint stiffness matrix to include the K_g matrix terms which are based on the computed tensile/compressive axial member forces.
 3. Solving simultaneous equations for displacements;
 4. If a **RESPONSE SPECTRUM**, **TIME LOAD**, or **GROUND MOTION** is specified

Section 5 Commands and Input Instructions

5.37 Analysis Specification

within a load case or the **MODAL CALCULATION** command is used, a dynamic analysis is performed. The static cases solved for a **PDELTA KG** analysis command will be solved first then the dynamic analysis cases.

Note: The stiffness matrix used in the dynamic analysis will be the $K+K_g$ matrix used in the last iteration for the last static case. This is a stress stiffened dynamic analysis, sometimes known as a PDELTA Dynamic analysis .

- c. A **PDELTA KG ANALYSIS** will correctly reflect the secondary effects of a combination of load cases only if they are defined using the **REPEAT LOAD** specification (section 5.32.11). Secondary effects will not be evaluated correctly for load combinations since only final results are combined.
- d. P-Delta KG effects are computed for frame members and plate elements only. They are not calculated for solid elements. The results are based on “P-large & small Delta” effects.
- e. For static analysis, the other P-Delta command [**PDELTA (n) ANALYSIS (SMALL)**] with 20 or more iterations (option 1) is preferred.
- f. Tension/compression only members are *not* allowed with this **PDELTA KG** command. You must use the **PDELTA (n) ANALYSIS (SMALL)** command instead.
- g. Be aware that global buckling can occur in a **PDELTA KG ANALYSIS**. This condition is usually detected by STAAD. A message is issued (in the basic solver negative L-matrix diagonals may be reported) and the results for that case are set to zero. STAAD will continue with the next load case.
- h. Global buckling may not be detected which could result in a solution with large or infinite or NaN values for displacement or negative L-matrix diagonals or stability errors. Do not use the results of such cases. This condition may require a nonlinear analysis. Sometimes the loads from Repeat Load combination cases are too large; sometimes partial moment releases rather than the full release is needed, sometimes connectivity needs to be corrected. Always check the maximum displacements for P-Delta analyses.

Example

```
PDELTA KG ANALYSIS PRINT BOTH
```

PDELTA KG 2 ANALYSIS

5.37.3 Nonlinear Cable Analysis

General Format

```
{ PERFORM CABLE } ANALYSIS ( { STEPS f1 | EQITERATIONS f2 |
EQTOLERANCE f3 | SAGMINIMUM f4 | STABILITY f5 f6 | KSMALL f7
} ) (print-options)
```

```
print-options = { LOAD DATA | STATICS CHECK | STATICS LOAD
| BOTH | ALL }
```

See section 5.37.1 for details.

This command may be continued to the next line by ending with a hyphen.

Where:

f1 = Number of load steps. The applied loads will be applied gradually in this many steps. Each step will be iterated to convergence. Default is 145. The f1 value, if entered, should be in the range 5 to 145.

f2 = Maximum number of iterations permitted in each load step. Default is 300. Should be in the range of 10 to 500.

f3 = The convergence tolerance for the above iterations. Default is 0.0001.

f4 = Cables (not trusses) may sag when tension is low. This is accounted for by reducing the E value. Sag minimum may be between 1.0 (no sag E reduction) and 0.0 (full sag E reduction). Default is 0.0. As soon as SAGMIN becomes less than 0.95 the possibility exists that a converged solution will not be achieved without increasing the steps to 145 or the pretension loads. The Eq iterations may need to be 300 or more. The Eq tolerance may need to be greater or smaller.

f5 = A stiffness matrix value, f5, that is added to the global matrix at each translational direction for joints connected to cables and nonlinear trusses for the first f6 load steps. The amount added linearly decreases with each of the f6 load steps. If f5 entered, use 0.0 to 1000.0. Default is 1.0. This parameter alters the stiffness of the structure.

Section 5 Commands and Input Instructions

5.37 Analysis Specification

f_6 = The number of load steps over which f_5 is gradually applied.
Default is 1 (one) step.

f_7 = A stiffness matrix value, f_7 , that is added to the global matrix at each translational direction for joints connected to cables and nonlinear trusses for every load step. The range for f_7 is between 0.0 and 1.0. Default is 0.0 . This parameter alters the stiffness of the structure.

Notes

- STAAD allows multiple analyses in the same run (see the General Comments in section 5.37.1).
- Multiple Analyses may require use of additional commands like the **SET NL** command and the **CHANGE** command.
- Analysis and **CHANGE** are required between primary cases for **PERFORM CABLE ANALYSIS**.

5.37.4 Buckling Analysis

Note: This feature is available in STAAD.Pro 2007 Build 01 and higher.

General Format

PERFORM BUCKLING ANALYSIS (MAXSTEPS f_1) (PRINT *print-options*)

print-options = { LOAD DATA | STATICS CHECK | STATICS LOAD
| BOTH | ALL }

See section 5.37.1 for details.

Without this command (or any of the analysis commands described earlier), no analysis will be performed. These ANALYSIS commands can be repeated if multiple analyses are needed at different phases.

Where:

f_1 = maximum no. of iterations desired (default value of $n = 10$). 15 is recommended. Used for Basic Solver, only.

There are two procedures available. One comes with the **basic solver** and the other with the **advanced solver**.

Basic Solver

This command directs the program to perform an analysis that includes:

- a. Solving the static case.
- b. Re-forming the global joint stiffness matrix to include the Kg matrix terms which are based on the computed tensile/compressive axial member forces and inplane plate stresses..
- c. Solving simultaneous equations for displacements;
- d. Repeat b) and c) for the number of required additional iterations; either until convergence or until MAXSTEPS is reached.

If the loads must be in the opposite direction, STAAD will stop solving that case at 1 iteration. The results for the case will be outputted; then STAAD will continue with the next case.

Convergence occurs when two consecutive buckling factors in the iteration are within 0.1% of each other.

Results are based on the highest successful Buckling Factor estimate that was calculated; as if the original applied loads times the buckling factor had been entered.

Example with Basic Solver:

```
PERFORM BUCKLING ANALYSIS MAXSTEPS 15 -  
PRINT LOAD DATA
```

Advanced Solver

Note: MAXSTEPS input is ignored when using the advanced solver.

This command directs the program to perform the analysis that includes:

- a. Solving the static case.
- b. Re-forming the global joint stiffness matrix to include the Kg matrix terms which are based on the computed tensile/compressive axial member forces & inplane plate stresses..
- c. Solving an eigenvalue problem for up to 4 buckling factors and buckling shapes

Section 5 Commands and Input Instructions

5.37 Analysis Specification

A Buckling Analysis will correctly reflect the secondary effects of a combination of load cases only if they are defined using the REPEAT LOAD specification (Section 5.32.11). Buckling will not be performed for LOAD COMBINATIONS cases.

Buckling Kg matrices are computed for frame members and plate elements only. They are not calculated for solid elements or curved members. The results are based on “P-large & small Delta” effects (refer to Option 1 in section 5.37.2).

Buckling Analysis solves for Buckling Factors. These are the amounts by which the load case must be factored for the buckling shape to occur. Only the last buckling case will be presented in the post processing. However, the buckling factors of all buckling cases will be written to the output.

If the loads must be in the opposite direction, STAAD will compute negative buckling factors.

Results are for the normalized buckling shape not as if the original applied loads times the buckling factor had been entered.

Example with Advanced Solver:

```
PERFORM BUCKLING ANALYSIS
```

5.37.5 Direct Analysis

The Direct Analysis is available effective STAAD.Pro 2007 Build 03

General Format

```
PERFORM DIRECT ANALYSIS ( {LRFD or ASD} TAUTOL f1 DISPtol f2  
ITERDIRECT i3 (REDUCEDEI i4) (PDiter i5) PRINT print-  
options )
```

```
print-options = { LOAD DATA | STATICS CHECK | STATICS LOAD  
| BOTH | ALL }
```

See "Linear Elastic Analysis" on page 676 for details.

This command directs the program to perform the analysis that includes:

- Reduce Axial & Flexure stiffness to 80% for members selected in the Define Direct input. The 80% applies only to analysis.
- Solving the static case which has notional loads included.
- Perform iterations of the iterative PDelta with SmallDelta analysis

- procedure (default 15 iterations).
- d. Solving simultaneous equations for displacements;
 - e. Compute Tau-b of AISC 05 Direct Analysis Appendix 7 based on required strength versus yield strength.
 - f. Re-forming the global joint stiffness matrix.
 - g. Solving simultaneous equations for displacements;
 - h. Repeat steps c) through g) until converged or **ITERDIRECT** iterations are reached.

A Direct Analysis will correctly reflect the secondary effects of a combination of load cases only if they are defined using the **REPEAT LOAD** specification (Section 5.32.11) and/or **REFERENCE LOAD** specification (Section 5.33). Direct analysis will not be performed for **LOAD COMBINATIONS** cases.

Notional loads must be defined using a **DEFINE NOTIONAL** table.

A list of members which will have their initial Tau-b value set and/or have their Axial stiffness reduced and/or their flexural stiffness reduced must be entered using a **DEFINE DIRECT** table.

For information on **NOTIONAL LOADS**, see sections 5.31.7 and 5.32.14.

PDELTA iterative load adjustments are computed for frame members only. They are not calculated for plate or solid elements. The results are based on “P-large&small Delta” effects (refer to Option 1 in section 5.37.2).

Convergence occurs when 2 consecutive iterations have all member tau-b values the same within a tolerance, **TAUTOL**, and displacements & rotations the same within a tolerance, **DISPTOL**.

LRFD is the default (all generated loads are factored by 1.0). If ASD entered then loads are factored by 1.6 for the Pdelta and Tau-b calculations. ASD final results are based on the final displacements divided by 1.6

If resulting displacements are diverging, then the P-Delta iterations will be terminated and the current iteration results will be used as the final results for that load case.

Section 5 Commands and Input Instructions

5.37 Analysis Specification

Table 5-32: Direct analysis parameters

Parameter Name	Default Value	Description
TAUTOL	0.01	Tau-b tolerance f_1 is normally 0.001 to 1.0.
DISPTOL	0.01 inch (displacement) 0.01 radians (rotation)	Displacement tolerance f_2 should not be too tight. The value is in current length units.
ITERDIRECT	1	Limits the number of iterations. A value for i_3 between 1 to 10 is typically sufficient.
REDUCEDEI	1	Integer, i_4 , specifies whether to use the reduced EI ($\text{Tau-b} * 0.8 * EI$) for member section moment & section displacement. = 1 - uses the reduced EI ($\text{Tau-b} * 0.8 * EI$) for member section moment & section displacement calculations = 0 - uses the full EI for member section moment & section displacement calculations.
PDiter	15	The number of iterations, i_5 , used in the iterative PDelta with SmallDelta analysis procedure within Direct Analysis; 5 to 25 iterations is the normal range. The default is recommended..

Note: You can use the **SET NOPRINT DIRECT** command to turn off the tau-b details in the output file when running a Direct Analysis. This can greatly reduce the volume of output content for models with many load cases.

Example

```
PERFORM DIRECT ANALYSIS LRFD TAUTOL 0.01 -  
DISPTOL 0.01 ITERDIRECT 2 -  
PRINT LOAD DATA
```

5.37.6 Steady State and Harmonic Analysis

The options available under steady state analysis in STAAD are described in the next few sections.

5.37.6.1 Purpose

This analysis type is used to model steady, harmonically varying load on a structure to solve for the steady harmonic response after the initial transient response has damped out to zero. STAAD Steady State analysis options include results for one forcing frequency or for a set of frequencies. You may specify ground motion or a distributed joint loading in one load case. Damping is required either in this input or from the Modal Damp input or from the Composite Damping input.

This command is used to specify the analysis request, specify that the load case with the **MODAL CALCULATION** command (which must be prior to this analysis command) be used as the definition of the mass distribution, and to begin a block of data input describing the steady state forcing functions, the output frequencies, and the printing of the joint responses.

All of the input and output frequencies are in Hertz (Hz or CPS).

Related topics can be found in the following sections:

Section 5 Commands and Input Instructions

5.37 Analysis Specification

Misc. Settings for Dynamics, Cut off values, and mode selection	-	5.30
Frequency Calculation	-	5.34
Modal & Composite Damping	-	5.26.4, 5.26.5
Analysis Specification	-	5.37
Dynamic Analysis overview	-	1.18.3

The Modal Calculation command is required in the weight/mass definition load case. See section 5.34.

General Format

PERFORM STEADY STATE ANALYSIS

This command directs the program to perform the analysis that includes:

- a. Checking whether all information is provided for the analysis;
- b. Forming the joint stiffness matrix;
- c. Solving simultaneous equations;
- d. Solving for modes and frequencies;
- e. Computing for the steady state joint displacements, velocities & accelerations and phase angles;
- f. Computing the above quantities versus frequency and displaying the results graphically.
- g. Member & element forces & stresses and support reactions are not currently computed.

The first input after the Perform Steady State Analysis command is:

BEGIN ({ STEADY | HARMONIC }) ({ FORCE | GROUND })

Steady or Harmonic

Steady = the analysis is at one forcing frequency.

Harmonic = the analysis is at several frequencies.

Force or Ground

Choose whether the loading is a distributed joint force load or a ground motion.

This command selects which of the 4 load/analysis types, that are available, will be used in this analysis. These four are described in sections 5.37.6.4 through 5.37.6.7.

This block of data should be terminated with the **END STEADY** command as mentioned in Section 5.37.6.9.

The steady state/harmonic analysis will calculate the maximum displacement and the associated phase angle for each of 6 joint directions, relative to the ground motion, for each frequency defined in the next section.

In **PRINT JOINT DISP** and in Post processor displayed results, the load case displacement for a given joint and direction will be the maximum value over all of the frequencies (without the phase angles) for a Steady State load case.

In post-processing for harmonic analysis, Log-Log graphs of any joint's relative translational displacement or velocity or acceleration versus frequency may be selected.

See section 5.37.6.8 for printing displacements with phase angles by frequency.

5.37.6.2 Define Harmonic Output Frequencies

If Harmonic is requested above, then optionally include the next input.

FREQUENCY (FLO f_1 FHI f_2 NPTS f_3 (MODAL) FLIST freqs)

Where:

f_1 = Lowest frequency to be included in Harmonic output. Default to half the first natural frequency.

f_2 = Highest frequency to be included in Harmonic output. Default to highest frequency plus largest difference between two consecutive natural frequencies.

f_3 = Number of plot frequencies to be included between natural frequencies. Defaults to 5 (7 if fewer than 10 modes) (3 if more than 50 modes). These points are added to improve the graphic display of responses versus frequency.

The natural and forcing frequencies are automatically included in the plot frequencies.

Section 5 Commands and Input Instructions

5.37 Analysis Specification

MODAL

This option causes the natural frequencies between **FLO** and **FHI** to be added to the list of forcing frequencies.

FLIST

freqs = List of forcing frequencies to be included in the Harmonic analysis. Continue freqs input to additional lines by ending each line except the last with a hyphen.

Only forcing frequencies will be used to create load case results and print results.

5.37.6.3 Define Load Case Number

The load case number is automatically the case with the **MODAL CALC** command (see Section 5.34.2)

5.37.6.4 Steady Ground Motion Loading

This set of commands may be used to specify steady ground motion loading on the structure, the ground motion frequency, the modal damping, and the phase relationship of ground motions in each of the global directions.

General Format

This command specifies the ground motion frequency and damping.

STEADY GROUND FREQ f_1 { **DAMP** f_2 | **CDAMP** | **MDAMP** } { **ABSOLUTE** | **RELATIVE** }

Where:

f_1 = value is the steady state frequency (Hz) at which the ground will oscillate.

f_2 = Damping ratio for all modes when **DAMP** is selected. Default value is 0.05 (5% damping if 0 or blank entered).

DAMP, **MDAMP**, and **CDAMP** Select source of damping input:

- **DAMP** indicates to use the f_2 value for all modes.
- **MDAMP** indicates to use the damping entered or computed with the **DEFINE DAMP** command if entered, otherwise default value of 0.05 will be used.

- CDAMP indicates to use the composite damping of the structure calculated for each mode. One must specify damping for different materials under the CONSTANT specification.

ABSOLUTE or **RELATIVE**. Ground motion results in output file will be relative to the ground unless ABSOLUTE is specified. Graphical results are relative. This option has no effect for the force loading cases.

General Format

Enter the direction of the ground motion, the acceleration magnitude, and the phase angle by which the motion in this direction lags (in degrees). One Ground Motion command can be entered for each global direction.

GROUND MOTION { X | Y | Z } { ACCEL | DISP } f_3 PHASE f_4

Where:

f_3 = Ground acceleration in g's or displacement in length units.
 f_4 = Phase angle in degrees

5.37.6.5 Steady Force Loading

This set of commands may be used to specify JOINT loads on the structure, the forcing frequency, the modal damping, and the phase relationship of loads in each of the global directions.

General Format

This command specifies the forcing frequency and damping for a case of steady forces.

STEADY FORCE FREQ f_1 { DAMP f_2 | CDAMP | MDAMP }

Where:

f_1 = value is the steady state frequency at which the joint loads below will oscillate.

f_2 = Damping ratio for all modes when **DAMP** is selected. Default value is 0.05 (5% damping if 0 or blank entered).

DAMP, **MDAMP**, and **CDAMP** Select source of damping input:

Section 5 Commands and Input Instructions

5.37 Analysis Specification

- DAMP indicates to use the f_2 value for all modes.
- MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used.
- CDAMP indicates to use the composite damping of the structure calculated for each mode. One must specify damping for different materials under the CONSTANT specification.

Joint Loads

Refer to Section 5.32.1 for additional information on Joint Loads

```
JOINT LOAD ( [ PHASE *{ X | Y | Z } f7] )
```

Where:

f_7 = Phase angle in degrees. One phase angle per global direction.

Bracketed data may be entered for each global direction on the same line. All moments specified below will be applied with a phase angle of 0.0. All forces specified below will be applied with the phase angle specified above, if any. Default is 0.0.

Next are the joint forces, if any. Repeat as many lines of joint force data as needed.

```
joint-list *{ FX f1 | FY f2 | FZ f3 | MX f4 | MY f5 | MZ f6 }
```

Where:

f_1, f_2, f_3 = specify a force in the corresponding global direction.

f_4, f_5, f_6 = specify a moment in the corresponding global direction.

Notes

- a. Joint numbers may be repeated where loads are meant to be additive in the joint.
- b. **UNIT** command may be on lines in between joint-list lines.
- c. Forces applied at a slave DOF will be ignored.

Copy Loads

The next command, Copy Load, may optionally be placed here to use the equivalent joint loads from prior cases. This feature enables using the more complex loading commands like selfweight, floor load, wind load, etc. that are not directly available here.

COPY LOAD

$i_1, f_1, i_2, f_2 \dots i_n, f_n$

Where:

$i_1, i_2 \dots i_n$ = prior primary load case numbers that are in this analysis set.

$f_1, f_2 \dots f_n$ = corresponding factors

This command can be continued to additional lines by ending all but last with a hyphen. These cases must have been between the Perform Steady State Analysis command and the prior Analysis command (if any).

5.37.6.6 Harmonic Ground Motion Loading

This set of commands may be used to specify harmonic ground motion loading on the structure, the modal damping, and the phase relationship of ground motions in each of the global directions. Response at all of the frequencies defined in section 5.37.6.2 will be calculated.

General Format

HARMONIC GROUND { DAMP f_1 | CDAMP | MDAMP } { ABSOLUTE
| RELATIVE }

Where:

f_1 = Damping ratio for all modes when **DAMP** is selected. Default value is 0.05 (5% damping if 0 or blank entered).

This command specifies the damping. The steady state response will be calculated for each specified output frequency entered or generated, see section 5.37.6.2.

DAMP, **MDAMP**, and **CDAMP** Select source of damping input:

Section 5 Commands and Input Instructions

5.37 Analysis Specification

- DAMP indicates to use the f_2 value for all modes.
- MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used.
- CDAMP indicates to use the composite damping of the structure calculated for each mode. One must specify damping for different materials under the CONSTANT specification.

ABSOLUTE or **RELATIVE**. Ground motion results in output file will be relative to the ground unless ABSOLUTE is specified. Graphical results are relative. This option has no effect for the force loading cases.

General Format

GROUND MOTION { X | Y | Z } { ACCEL | DISP } f_3 PHASE f_4

Enter the direction of the ground motion, the acceleration and the phase angle by which the motion in this direction lags (in degrees). One Ground Motion command can be entered for each global direction.

Where:

f_3 = Ground acceleration in g's or displacement in length units.

f_4 = Phase angle in degrees

Next is an optional amplitude versus frequency specification to be used when the ground motion acceleration is a function of frequency. For any forcing frequency an amplitude can be determined, from the data below, which will multiply the acceleration f_3 entered above. If no amplitude data is entered for a direction then the acceleration is f_3 for that direction.

AMPLITUDE (A a B b C c)

Where:

Amplitude = $a \cdot \omega^2 + b \cdot \omega + c$

ω = forcing frequency in rad/sec.

a, b, c = Constants. a and b default to 0.0 and c defaults to 1.0.

Or

AMPLITUDE

(f_1 a_1 f_2 a_2 ... f_n a_n)

$f_1 a_1 f_2 a_2 \dots f_n a_n$ = Frequency - Amplitude pairs are entered to describe the variation of acceleration with frequency. Continue this data onto as many lines as needed by ending each line except the last with a hyphen (-). These pairs must be in ascending order of frequency. Use up to 199 pairs. Linear interpolation is used.

One Ground Motion and Amplitude command set can be entered for each global direction.

5.37.6.7 Harmonic Force Loading

This set of commands may be used to specify JOINT loads on the structure, the modal damping, and the phase relationship of loads in each of the global directions. Response at all of the frequencies defined in section 5.37.6.2 will be calculated.

General Format

HARMONIC FORCE { DAMP f_1 | CDAMP | MDAMP }

Where:

f_1 = Damping ratio for all modes. Default value is 0.05 (5% damping if 0 or blank entered).

This command specifies the damping for a case of harmonic forces.

DAMP , MDAMP and CDAMP. Select source of damping input.

- DAMP indicates to use the f_1 value for all modes.
- MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used.
- CDAMP indicates to use the composite damping of the structure calculated for each mode. One must specify damping for different materials under the CONSTANT specification

Joint Loads

Refer to Section 5.32.1 for additional information on Joint Loads

JOINT LOAD ([PHASE *{ X | Y | Z } f_7])

Where:

Section 5 Commands and Input Instructions

5.37 Analysis Specification

f_7 = Phase angle in degrees. One phase angle per global direction.

Bracketed data may be entered for each global direction on the same line. All moments specified below will be applied with a phase angle of o.o. All forces specified below will be applied with the phase angle specified above, if any. Default is o.o.

Next are the joint forces, if any. Repeat as many lines of joint force data as needed.

```
joint-list *{ FX f1 | FY f2 | FZ f3 | MX f4 | MY f5 | MZ f6 }
```

Where:

f_1, f_2, f_3 = specify a force in the corresponding global direction.

f_4, f_5, f_6 = specify a moment in the corresponding global direction.

Notes

- Joint numbers may be repeated where loads are meant to be additive in the joint.
- UNIT** command may be on lines in between joint-list lines.
- Forces applied at a slave DOF will be ignored.

Copy Loads

The next command, Copy Load, may optionally be placed here to use the equivalent joint loads from prior cases. This feature enables using the more complex loading commands like selfweight, floor load, wind load, etc. that are not directly available here.

COPY LOAD

```
i1, f1, i2, f2 ... in, fn
```

Where:

$i_1, i_2 \dots i_n$ = prior primary load case numbers that are in this analysis set.

$f_1, f_2 \dots f_n$ = corresponding factors

This command can be continued to additional lines by ending all but last with a hyphen. These cases must have been between the Perform Steady State Analysis command and the prior Analysis command (if any).

Next is an optional force multiplier (amplitude) versus frequency specification to be used when the force loading is a function of frequency. For any forcing frequency an amplitude can be determined, from the data below, which will multiply the forcing loads entered above. If no amplitude data is entered then the forcing loads are as entered above.

AMPLITUDE (A a B b C c)

Where:

$$\text{Amplitude} = a*\omega^2 + b*\omega + c$$

ω = forcing frequency in rad/sec.

a, b, c = Constants. a and b default to 0.0 and c defaults to 1.0.

Or

AMPLITUDE

(f₁ a₁ f₂ a₂ ... f_n a_n)

f₁ a₁ f₂ a₂ ... f_n a_n = Frequency - Amplitude pairs are entered to describe the variation of forces with frequency. Continue this data onto as many lines as needed by ending each line except the last with a hyphen (-). These pairs must be in ascending order of frequency. Use up to 199 pairs. Linear interpolation is used.

Leaving the direction field blank or inserting ALL will use the same Frequency versus Amplitude for all 6 force directions.

Enter amplitudes for up to three directions. For directions without amplitude input, including moment directions, the amplitude will be set to 1.0.

5.37.6.8 Print Steady State/Harmonic Results

General Format

PRINT HARMONIC DISPLACEMENTS *list-spec*

***list-spec* = { (ALL) | LIST *list of items-joints* }**

This command must be after all steady state/harmonic loadings and before the END STEADY command. For each harmonic frequency of section 5.37.6.2 the following will be printed:

1. Modal responses.
2. Phase angles with 1 line per selected joint containing the phase angle for

Section 5 Commands and Input Instructions

5.37 Analysis Specification

- each of the 6 directions of motion.
3. Displacements table with 1 line per selected joint containing the maximum displacements for each of the 6 directions of motion.
 4. Velocities.
 5. Accelerations.

Steady State Examples

```
BEGIN STEADY GROUND
STEADY GROUND FREQ 22.4 DAMP .033 ABS
GROUND MOTION X ACC .11 PHASE 0.0
GROUND MOTION Y ACC .21 PHASE 10.0
GROUND MOTION Z ACC .15 PHASE 20.0
PRINT HARMONIC DISP ALL
END
```

```
BEGIN HARMONIC GROUND
FREQ FLO 3.5 FHI 33 NPTS 5 MODAL FLIST 4 5 10 -
17 21 30
HARMONIC GROUND DAMP .033 REL
GROUND MOTION X ACC .11 PHASE 0.0
AMPLIT A 0.10 B .21 C 0.03
GROUND MOTION Y DIS .21 PHASE 10.0
AMPLITUDE
3 5 5 4 10 6 -
35 3
GROUND MOTION Z ACC .15 PHASE 20.0
AMPLIT A 0.10 B .21 C 0.03
PRINT HARMONIC DISP ALL
END
```

```
BEGIN STEADY FORCE
```

```
STEADY FORCE FREQ 11.2 DAMP .033
JOINT LOAD PHASE X 0.0 PHASE Y 10.0 PHASE Z 15.0
UNIT KIP
10 5 TO 7 BY 2 88 FX 10.0 FY 5.0
UNIT POUND
10 5 TO 7 BY 2 -
88 FX 10.0 FY 5.0
COPY LOAD
1 1.5 2 0.8 -
3 1.0
PRINT HARMONIC DISP ALL
END
```

```
BEGIN HARMONIC FORCE
FREQ FLO 3.5 FHI 33 NPTS 5 MODAL FLIST 4 5 10 -
17 21 30
HARMONIC FORCE DAMP .033
JOINT LOAD PHASE X 0.0 PHASE Y 10.0 PHASE Z 15.0
UNIT KIP
10 5 TO 7 BY 2 88 FX 10.0 FY 5.0
UNIT POUND
10 5 TO 7 BY 2 -
88 FX 10.0 FY 5.0
COPY LOAD
1 1.5 2 0.8 -
3 1.0
AMPLIT X A 0.10 B .21
AMPLITUDE Y
3 5 5 4 10 6 -
35 3
AMPLIT Z A 0.10 C 0.03
```

Section 5 Commands and Input Instructions

5.37 Analysis Specification

```
PRINT HARMONIC DISP ALL  
END
```

```
BEGIN HARMONIC FORCE  
FREQ FLO 3.5 FHI 33 NPTS 5 MODAL FLIST 4 5 10 -  
17 21 30  
HARMONIC FORCE DAMP .033  
JOINT LOAD PHASE X 0.0 PHASE Y 10.0 PHASE Z 15.0  
UNIT KIP  
10 5 TO 7 BY 2 88 FX 10.0 FY 5.0  
UNIT POUND  
10 5 TO 7 BY 2 -  
88 FX 10.0 FY 5.0  
COPY LOAD  
1 1.5 2 0.8 -  
3 1.0  
AMPLIT ALL A 0.10 B .21  
PRINT HARMONIC DISP ALL  
END
```

Note: For full examples, refer to the files SVM33.STD, SVM32.STD, SS-BEAM2.STD, SS-BEAM3.STD, EXAM07.STD, and EXAM14B.STD in the folder ...\\SPROV8I\\STAAD\\EXAMP\\STEADYSTATE\\, which is found in the program folder where STAAD.Pro V8i was installed.

Stead State Notes

For members, the final results at the joints will be complete but the section results will be as if the member loads were applied as statically equivalent loads at the member ends.

5.37.6.9 Last Line of this Steady State/Harmonic Analysis

Used to end a Steady State or Harmonic Analysis.

General Format**END STEADY****5.37.7 Pushover Analysis**

Refer to the STAAD.Pro Pushover Analysis Help for details of the implementation of pushover analysis.

Note: An advanced analysis module license is required to access this feature.

5.37.8 Geometric Nonlinear Analysis

Note: Effective from STAAD.Pro V8i (release 20.07.05).

General Format

**PERFORM NONLINEAR ANALYSIS (ARC f_1) (ITERATION i_1)
(TOLERANCE f_2) (STEPS i_2) (REBUILD i_3) (KG (i_4)) (JOINT_
TARGET i_5 (i_6)) (DISPL_TARGET f_3) (PRINT *print-specs*)**

The first analysis step must be stable, otherwise use ARC control to prevent instability. The procedure does not use follower loads. Loads are evaluated at the joints before the first step; then those loads translate with the joint but do not rotate with the joint. Equilibrium is computed in the displaced position.

Note: The nonlinear analysis command requires the Advanced Analysis Engine package.

The following table describes the parameters available for nonlinear analysis:

Section 5 Commands and Input Instructions

5.37 Analysis Specification

Table 5-33: Geometric Nonlinear Analysis parameters

Parameter Name	Default Value	Description
f_1	0.0	Displacement control. Value is the absolute displacement limit for the first analysis step. If max. displacement is greater than this limit, ARC will calculate a new step size for the first step and a new value for STEPS. Value should be in current length units. ARC = 0 indicates no displacement control.
i_1	100	Max. Number of iterations to achieve equilibrium in the deformed position to the tolerance specified.
f_2	0.0001 inch	For convergence, two successive iteration results must have all displacements the same within this tolerance. Value entered is in current units.
i_2	1	Number of load steps. Load is applied in stages if entered. One means that all of the load is applied in the first step.

Parameter Name	Default Value	Description
i_3	1	Frequency of rebuilds of the tangent K matrix per load step & iteration. 0. = once per load step 1. = every load step & iteration
i_4	KG	This parameter controls whether the geometric stiffness, KG, is added to the stiffness matrix, K. <ul style="list-style-type: none"> • KG or KG 1 Use K+KG for the stiffness matrix. (Default) • KG 0 Do not use KG.
i_5	none	Joint being monitored in a displacement target analysis.
i_6	1	Local DOF (1 through 6)
f_3	none	Displacement target value in current length units.
<i>print-spec</i>	none	Standard STAAD analysis print options. See section 5.37.1 for details.

If Joint target and Displacement target are entered and the **STEPS** parameter is greater than two; then the analysis will proceed step by step until the targeted joint degree of freedom has displaced **DISPL_TARGET** length units or more.

Nonlinear entities such as tension/compression members, multilinear springs, gaps, etc. are not supported when using a nonlinear analysis. Additionally, nonlinear analysis does not account for post-buckling stiffness of members.

Section 5 Commands and Input Instructions

5.37 Analysis Specification

Warning: The deprecated **NONLINEAR nn ANALYSIS** command will adopt the new procedure (unless a SET command is entered to invoke the old procedure for backward compatibility). The Simple parameter with iterations equal to nn will be used.

5.37.9 Imperfection Analysis

This performs a modified linear elastic analysis using Member Imperfection Specifications (see section 5.26.6) defined on beam and column members.

An Imperfection analysis will reflect the secondary effects only if the camber and/or drift is specified in a **DEFINE IMPERFECTIONS** specification (Section 5.26.6). For combination of load cases, with imperfection, use the Repeat Load specification rather than the Load Combination.

General Format

```
PERFORM IMPERFECTION ANALYSIS (PRINT { LOAD DATA | STATICS  
CHECK | STATICS LOAD | BOTH | ALL } )
```

See "Linear Elastic Analysis" on page 676 for details.

Without one of these analysis commands, no analysis will be performed. These **ANALYSIS** commands can be repeated if multiple analyses are needed at different phases.

5.37.10 Spectrum Command

This command is used to specify the calculation of floor spectra from time history acceleration results. The Floor Response Spectrum command must immediately follow an analysis command associated with the time history load case. The acceleration used may be either the absolute acceleration or the relative to ground acceleration.

Note: This data should only be entered if there is a time history case being solved.

General Format

First line of this Floor Spectrum Data.

```
GENERATE FLOOR SPECTRUM
```

Specify Floor Groups

Each new floor definition starts with the next command:

BEGIN FLOOR DIRECTION { GX | GY | GZ } { TITLE }

GX, GY and GZ specify up to 3 global directions for which acceleration vs frequency spectrums will be generated for this floor. Optionally enter a Title/Description (up to 50 characters) for this floor that will be displayed on the graphs in post processing.

The next one or more lines will identify the floors which will have spectrum curves generated either by referencing a NODE GROUP (see section 5.16) or explicitly listing the joints that constitute the floor. For multiple groups, enter each on a separate line.

{ *_jointgroup* | *jointlist* }

Enter as many lines as necessary to specify all of the groups needed to define this floor.

If more floors are to be defined, then repeat the Begin Floor -direction command followed by the joint-*_Group_Name* data. Enter as many floors as desired.

Specify Options

After the last floor definition, enter the following parameters that will be used in all of the Floor Spectrum calculations.

**OPTIONS ({ FLOW f_1 | FHIGH f_2 | FDELTA f_3 | DAMP f_4 (... f_n)
| RELATIVE }) *print-options***

This command may be continued to the next line by ending the line with a hyphen.

Where:

f_1 = Lowest frequency to be in the calculated spectrum. The value for FLOW should be at least 0.01 Hz.

f_2 = Highest frequency to be in the calculated spectrum.

f_3 = The spectrum will be calculated at FDELTA intervals from FLOW to FHIGH.

f_4 ... f_n = Up to 10 damping values may be entered. One spectrum will be generated for each damping value for each global direction requested for each floor defined. The spectrum will be based on these modal damping ratios.

3% damping should be entered as 0.03. The default is 0.05.

Section 5 Commands and Input Instructions

5.37 Analysis Specification

RELATIVE= If there is ground motion defined and you want the spectrums to be based on the relative acceleration of the floor to the ground acceleration, then enter the Relative parameter. Default is absolute.

Optional print parameters.

Hint: Omitting these options is recommended.

THPRINT = 0, no print; =2, Print the time history acceleration being used in each spectrum calculation.

SPRINT = Print the calculated spectrum.

Last line of the floor spectrum data:

END FLOOR SPECTRUM

Example

```
DEFINE TIME HISTORY
TYPE 1 FORCE
0 -20 0.5 100 1 200 1.5 500 2 800 2.5 500 3 70 16 0
ARRIVAL TIME
0
DAMPING 0.075
*
LOAD 1 LOADTYPE SEISMIC TITLE TIME HISTORY CASE
* MASS MODEL REQUIRED
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
JOINT LOAD
1 TO 6 FX 62.223 FY 62.223 FZ 62.223
* TIME LOADS
TIME LOAD
2 FX 1 1
```

```

PERFORM ANALYSIS
GENERATE FLOOR SPECTRUM
BEGIN FLOOR DIRECTION GX GZ GROUND MOTION
_FL1
_FL17
BEGIN FLOOR DIRECTION GX GZ FLOOR 18 A/C UNIT 36
_FL18
OPTIONS FLO 0.5 FHI 35.0 FDEL 0.1 -
DAMP 0.03 0.05 0.07
END FLOOR SPECTRUM

```

5.38 Change Specification

This command is used to reset the stiffness matrix. Typically, this command is used when multiple analyses are required in the same run.

General Format

CHANGE

This command indicates that input, which will change the stiffness matrix, will follow. This command should only be used when an analysis has already been performed. The CHANGE command does or requires the following:

- sets the stiffness matrix to zero
- makes members active if they had been made inactive by a previous INACTIVE command
- allows the re-specification of the supports with another SUPPORT command that causes the old supports to be ignored. The SUPPORT specification must be such that the number of joint directions that are free to move (DOF or "releases") before the CHANGE must be greater than or equal to the number of "releases" after the CHANGE.
- the supports must be specified in the same order before and after the CHANGE command. To accomplish this when some cases have more supports than others do, you can enter unrestrained joints into the SUPPORT command list using FIXED BUT FX FY FZ MX MY MZ. It is best to put every joint that will be supported in any case into every SUPPORT

Section 5 Commands and Input Instructions

5.38 Change Specification

list.

- CHANGE, if used, should be after PERFORM ANALYSIS and before the next set of SUPPORT, LOADS.
- Only active cases are solved after the CHANGE command.
- Analysis and CHANGE are required between primary cases for PERFORM CABLE ANALYSIS.
- Analysis and CHANGE are required after UBC cases if the case is subsequently referred to in a Repeat Load command or if the UBC case will be re-solved after a Select command or after a Multiple analysis.

Example

Before **CHANGE** is specified:

```
1 PINNED
2 FIXED BUT FX MY MZ
3 FIXED BUT FX MX MY MZ
```

After **CHANGE** is specified:

```
1 PINNED
2 FIXED
3 FIXED BUT FX MZ
```

The **CHANGE** command is not necessary when only member properties are revised to perform a new analysis. This is typically the case in which the user has asked for a member selection and then uses the PERFORM ANALYSIS command to reanalyze the structure based on the new member properties.

Notes

- a. If new load cases are specified after the CHANGE command such as in a structure where the INACTIVE MEMBER command is used, the user needs to define the total number of primary load cases using the SET NL option (see Section 5.5 and Example 4).
- b. Multiple Analyses using the CHANGE command should not be performed if the input file contains load cases involving dynamic analysis or Moving

Load Generation.

- c. Section forces and moments, stress and other results for postprocessing will use the last entered data for supports and member properties regardless of what was used to compute the displacements, end forces and reactions. So beware of changing member properties and releases after a CHANGE command.

5.39 Load List Specification

This command allows specification of a set of active load cases. All load cases made active by this command remain active until a new load list is specified.

This command is used to activate the load cases listed in this command and, in a sense, deactivate all other load cases not listed in this command. In other words, the loads listed are used for printing output and in design for performing the specified calculations. When the PERFORM ANALYSIS command is used, the program internally uses all load cases, regardless of LOAD LIST command, except after a CHANGE command. In these two cases, the LOAD LIST command allows the program to perform analysis only on those loads in the list. If the LOAD LIST command is never used, the program will assume all load cases to be active.

General Format

```
LOAD LIST { load-list | ALL }
```

Example

```
LOAD LIST ALL  
PRINT MEMBER FORCES  
LOAD LIST 1 3  
PRINT SUPPORT REACTIONS  
CHECK CODE ALL
```

In this example, member forces will be printed for all load cases, whereas loading 1 and 3 will be used for printing support reactions and code-checking of all members.

Section 5 Commands and Input Instructions

5.40 Load Envelope

Notes

- a. The **LOAD LIST** command may be used for multiple analyses situations when an analysis needs to be performed with a selected set of load cases only. All load cases are automatically active before a first **CHANGE** command is used.
- b. After a **CHANGE** command has been used anywhere in the data, it is good practice to specify the **LOAD LIST** command after an **ANALYSIS** command and before the next command; otherwise only the last case analyzed may be used in the design.
- c. Do not enter this command within the loads data (from the first Load command in an analysis set to the associated Analysis command).

5.40 Load Envelope

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

It is an alternative to the **LOAD LIST** command described in Section 5.39.

The envelope can be tagged with optional key words to specify the qualitative nature of the load or load combination cases included in the envelop definition.

Example

In the example below, the keyword **SERVICEABILITY** is associated with envelop 2. Keywords can be any single word (with no blank spaces) of the your choice.

```
DEFINE ENVELOPE
1 TO 8 ENVELOPE 1 TYPE CONNECTION
9 TO 15 ENVELOPE 2 TYPE SERVICEABILITY
16 TO 28 ENVELOPE 4 TYPE STRESS
END DEFINE ENVELOPE
```

The first line within the **DEFINE ENVELOPE** command means that load cases numbered 1 to 8 make up the **CONNECTION** type load envelope 1. Similarly load cases 9 to 15 define the **SERVICEABILITY** type load envelope 2.

To print out the support reactions corresponding to load envelope 1, the following commands should be provided in the input file

```
LOAD LIST ENV 1
PRINT SUPPORT REACTIONS
```

5.41 Section Specification

This command is used to specify sections along the length of frame member for which forces and moments are required.

This command specifies the sections, in terms of fractional member lengths, at which the forces and moments are considered for further processing.

General Format

```
SECTION  $f_1$ (  $f_2$ ) (  $f_3$ ) { MEMBER memb-List | (ALL) }
```

Where:

f_1, f_2, f_3 = Section (in terms of the fraction of the member length) provided for the members. Maximum number of sections is 3, and only values between 0.0 and 1.0 will be considered. In other words, no more than three intermediate sections are permissible per **SECTION** command.

Example

```
SECTION 0.17 0.48 0.72 MEMB 1 2
SECTION 0.25 0.75 MEMB 3 TO 7
SECTION 0.6 MEMB 8
```

In the above example, first, section locations of 0.17, 0.48, and 0.72 are set for members 1 and 2. In the next **SECTION** command, sections 0.25 and 0.75 are set for members 3 to 7. In the third **SECTION** command, member 8 has its section specified at 0.6. The remainder of the members will have no sections provided for them. As mentioned earlier, no more than three intermediate sections are allowed per **SECTION** command. However, if more than three intermediate sections are desired, they can be examined by repeating the **SECTION** command after completing the required calculations. The following example will clarify.

Section 5 Commands and Input Instructions

5.42 Print Specifications

Example

```
SECTION 0.2 0.4 0.5 ALL
PRINT SECTION FORCES
SECTION 0.6 0.75 0.9 ALL
PRINT SECTION FORCES
```

In this example, forces at three intermediate sections (namely 0.2, 0.4 and 0.5) are printed. Then forces at an additional 3 sections (namely 0.6, 0.75 and 0.9) are printed. This gives the user the ability to obtain section forces at more than three intermediate sections.

Notes

- The **SECTION** command just specifies the sections. Use the **PRINT SECTION FORCES** command after this command to print out the forces and moments at the specified sections.
- This is a secondary analysis command. The analysis must be performed before this command may be used.
- To obtain values at member ends (**START** and **END**), use the **PRINT MEMBER FORCES** command.
- If this command is specified before a steel design operation, and if the **BEAM** parameter is set to zero, the section locations specified by this command will also be designed for, in addition to the **BEAM** ends.

5.42 Print Specifications

This command is used to direct the program to print various modeling information and analysis results. STAAD offers a number of versatile print commands that can be used to customize the output.

General Format

Data Related Print Commands

```
PRINT { J O I N T C O O R D I N A T E | M E M B E R P R O P E R T I E S | E L E M E N T I N F O R M A T I O N (S O L I D) | M E M B E R P R O P E R T I E S | S U P P O R T I N F O R M A T I O N | A L L } { (A L L) | L I S T i t e m / j o i n t / m e m b e r - L i s t }
```

Print Location of CG

PRINT CG (_group_name)

Print Analysis Results

PRINT { (JOINT) DISPLACEMENTS | MEMBER FORCES (GLOBAL) | ANALYSIS RESULTS | MEMBER SECTION FORCES | MEMBER STRESSES | ELEMENT (JOINT) STRESSES (AT f_1 f_2) | ELEMENT FORCES | ELEMENT (JOINT) STRESSES SOLID | MODE SHAPES } *list-spec*
***list-spec* = { (ALL) | LIST *joint/member/elements-list* }**

Print Support Reactions

PRINT SUPPORT REACTIONS

Print Story Drifts or Stiffness

PRINT STORY DRIFT (f_3)

PRINT STORY STIFFNESS

Print Location of Center of Rigidity at each Floor

PRINT DIAPHRAGM CR

Description

The list of items is not applicable for **PRINT ANALYSIS RESULTS** and **PRINT MODE SHAPES** commands.

The **PRINT JOINT COORDINATES** command prints all interpreted coordinates of joints.

The **PRINT MEMBER INFORMATION** command prints all member information, including member length, member incidences, beta angles, whether or not a member is a truss member and the member release conditions at start and end of the member (1 = released, 0 = not released).

The **PRINT ELEMENT INFORMATION** command prints all incident joints, element thicknesses, and Poisson ratios for Plate/Shell elements. The **PRINT ELEMENT INFORMATION SOLID** command prints similar information for Solid elements.

The **PRINT MEMBER PROPERTIES** command prints all member properties including cross sectional area, moments of inertia, and section moduli in both axes. Units for the properties are always INCH or CM (depending on FPS or METRIC) regardless of the unit specified in UNIT command.

The following designation is used for member property names:

AX

Cross section area

Section 5 Commands and Input Instructions

5.42 Print Specifications

AY

Area used to adjust shear/bending stiffness in local Y axis to account for pure shear in addition to the classical bending stiffness

AZ

Area used to adjust shear/bending stiffness in local Z axis to account for pure shear in addition to the classical bending stiffness.

IZ

Moment of Inertia about the local Z-axis

IY

Moment of Inertia about the local Y-axis

IX

Torsional constant

SY

Smallest section modulus about the local Y-axis

SZ

Smallest section modulus about the local Z-axis

The **PRINT MATERIAL PROPERTIES** command prints all material properties for the members, including E (modulus of elasticity), G (shear modulus), weight density and coefficient of thermal expansion (alpha) for frame members. This command is available for members only. G may be listed as zero if command is before load cases and Poisson ratio was entered but G was not entered.

The **PRINT SUPPORT INFORMATION** command prints all support information regarding their fixity, releases and spring constant values, if any. The **LIST** option is not available for this command.

The **PRINT ALL** command is equivalent to last five print commands combined. This command prints joint coordinates, member information, member properties, material properties and support information, in that order.

The **PRINT CG** command prints out the coordinates of the center of gravity and the total weight of the structure or of a single group of member/elements. If the CG of a portion of the structure is desired, the members and elements of that portion must be assigned using a group name (see section 5.16 for details on using group-names). Only the selfweight of the structure is used to calculate the

C.G. User defined joint loads, member loads etc. are not considered in the calculation of C.G.

The **PRINT (JOINT) DISPLACEMENTS** command prints joint displacements in a tabulated form. The displacements for all six directions will be printed for all specified load cases. The length unit for the displacements is always INCH or CM (depending on FPS or METRIC unit) regardless of the unit specified in **UNIT** command.

The **PRINT (MEMBER) FORCES** command prints member forces (i.e., Axial force (AXIAL), Shear force in local Y and Z axes (SHEAR-Y and SHEAR-Z), Torsional Moment (TORSION), Moments about local Y and Z axes (MOM-Y and MOM-Z)) in a tabulated form for the listed members, for all specified load cases. The **GLOBAL** option will output forces in the global coordinate system rather than the member local coordinate system for each member.

The **PRINT SUPPORT REACTIONS** command prints global support reactions in a tabulated form, by support, for all specified load cases. Use LIST option for selected joints.

The **PRINT ANALYSIS RESULTS** command is equivalent to the above three commands combined. With this command, the joint displacements, support reactions and member forces, in that order, are printed.

The **PRINT (MEMBER) SECTION FORCES** command prints shears & bending moments at the intermediate sections specified with a previously input SECTION command. The printing is done in a tabulated form for all specified cases for the first requested member, then for the next member, etc.

The **PRINT (MEMBER) STRESSES** command tabulates member stresses at the start joint, end joint and all specified intermediate sections. These stresses include axial (i.e., axial force over the area), bending-y (i.e., moment-y over section modulus in local y-axis), bending-z (i.e., moment-z over section modulus in local z-axis), shear stresses in both local y and z directions (shear flow, q, over the shear area), and combined (absolute combination of axial, bending-y and bending-z) stresses.

For **PRISMATIC** sections, if AY and/or AZ is not provided, the full cross-sectional area (AX) will be used.

For **TAPERED** sections, the values of AY and AZ are those for the location where the stress is printed. Hence at the location o.o, the AY and AZ are based on the dimensions of the member at the start node.

The **PRINT ELEMENT STRESSES** command must be used to print plate stresses (SX, SY, SXY, SQX, SQY), moments per unit width (MX, MY, MXY) and principal

Section 5 Commands and Input Instructions

5.42 Print Specifications

stresses (SMAX, SMIN, TMAX) for plate/shell elements. Typically, the stresses and moments per unit width at the centroid will be printed. The Von Mises stresses (VONT, VONB) as well as the angle (ANGLE) defining the orientation of the principal planes are also printed.

The variables that appear in the output are the following. See "Sign Convention of Plate Element Stresses and Moments" on page 28 for more information regarding these variables.

SQX

Shear stress on the local X face in the Z direction

SQY

Shear stress on the local Y face in the Z direction

MX

Moment per unit width about the local X face

MY

Moment per unit width about the local Y face

MXY

Torsional Moment per unit width in the local X-Y plane

SX

Axial stress in the local X direction

SY

Axial stress in the local Y direction

SXY

Shear stress in the local XY plane

VONT

Von Mises stress on the top surface of the element

VONB

Von Mises stress on the bottom surface of the element

TrescaT

Tresca stress on the top surface of the element

TrescaB

Tresca stress on the bottom surface of the element

SMAX

Maximum in-plane Principal stress

SMIN

Minimum in-plane Principal stress

TMAX

Maximum in-plane Shear stress

ANGLE

Angle which determines direction of maximum principal stress with respect to local X axis

If the **JOINT** option is used, forces and moments at the nodal points are also printed out in addition to the centroid of the element.

The **AT** option may be used to print element forces at any specified point within the element. The **AT** option must be accompanied by f_1 and f_2 . f_1 and f_2 are local X and Y coordinates (in current units) of the point where the stresses and moments are required. For detailed description of the local coordinate system of the elements, refer to Section 1.6 of this manual.

The **PRINT ELEMENT FORCES** command enables printing of plate “corner forces” [$F_p = K_p \cdot D_p$] in global axis directions.

The **PRINT ELEMENT (JOINT) STRESS SOLID** command enables printing of stresses at the center of the **SOLID** elements. The variables that appear in the output are the following.

Normal Stresses

SXX, SYX and SZZ

Shear Stresses

SXY, SYZ and SZX

Principal Stresses

S_1 , S_2 and S_3 .

Von Mises Stresses

SE

Direction cosines

Section 5 Commands and Input Instructions

5.42 Print Specifications

Six direction cosines are printed following the expression DC, corresponding to the first two principal stress directions.

The **JOINT** option will print out the stresses at the nodes of the solid elements.

The **PRINT MODE SHAPES** command prints the relative joint motions of each of the modes that were calculated. The maximum motion is arbitrary and has no significance. Dynamic analysis will scale and combine the mode shapes to achieve the final dynamic results.

Example

```
PERFORM ANALYSIS
PRINT ELEMENT JOINT STRESS
PRINT ELEMENT STRESS AT 0.5 0.5 LIST 1 TO 10
PRINT SUPPORT REACTIONS
PRINT JOINT DISPLACEMENTS LIST 1 TO 50
PRINT MEMBER FORCES LIST 101 TO 124
```

Example

Printing the Center of Gravity (CG)

```
PRING CG
PRINT CG _RAFTERBEAMS
PRINT CG _RIDGEBEAMS
```

Notes

1. The output generated by these commands is based on the current unit system. The user may wish to verify the current unit system and change it if necessary.
2. Results may be printed for all joints/members/elements or based on a specified list.

Printing the Story Drift and Stiffness

The **PRINT STORY DRIFT** command may be used to obtain a print-out of the average lateral displacement of all joints at each horizontal level along the height of the structure.

The procedure used in STAAD.Pro for calculating story drift is independent of any code. For example, the story drift determination as explained in section 12.8.6 of the ASCE 7-05 code is *not* implemented in STAAD.

The method implemented in STAAD involves:

- a. Find all the distinct Y coordinates in the model. Those are what STAAD calls as stories.
- b. For each of those distinct stories, find all the nodes at that story elevation.
- c. For each story, find the average displacement along the horizontal directions (X and Z) by adding up corresponding displacement for all the nodes at that storey, and dividing by the number of nodes for that story. Thus, even if there is only a single node representing a story, a drift is calculated for that story too.

In STAAD.Pro if **PRINT STORY DRIFT** command is issued, the program prints the average of horizontal displacements of all the joints present at the particular floor level.

However, to check inter-story drift, the following commands needs to be issued after the **PERFORM ANALYSIS** command.

```
LOAD LIST i1
PRINT STORY DRIFT f3
```

Where:

i_1 = the primary load number for which inter-story drift check is required

f_3 = The allowable drift factor, as per the code provision

The program will calculate the relative horizontal displacement between two adjacent floors. This calculated value is checked against the allowable limit. The result is reported as either "PASS" or "FAIL" in the output.

There is only one exception to this format. For IS 1893: 2002 static seismic load case, even if this factor is not provided the program internally checks if the loading is IS 1893 static seismic loading or not. If yes, it automatically calculates the inter-story drift and checks against the code provisions.

Section 5 Commands and Input Instructions

5.43 Stress/Force output printing for Surface Entities

For dynamic IS 1893: 2002 response spectrum analysis the above format for inter-story drift check does not hold true. The reason is that in response spectrum analysis the joint displacements represents the maximum magnitude of the response quantity that is likely to occur during seismic loading. Any response quantity like story drift should be calculated from actual displacements of each mode considered during analysis. The inter-story drift from each mode is combined using modal combination to get the maximum magnitude of this response quantity. In order to compute story drift for IS 1893 response spectrum the load case command format described in Section 5.32.10.1.2 must be used..

The **PRINT STORY STIFFNESS** command may be used to include the calculated lateral stiffness of each story used in determining the drift. In STAAD.Pro lateral stiffness is calculated only when the floor is modeled as rigid floor diaphragm since it functions as transferring story shears and torsional moments to lateral force-resisting members during earthquake.

Printing the Center of Rigidity & Center of Mass

The **PRINT DIAPHRAGM CR** command may be used to obtain a print-out of the center of rigidity and center of mass at each rigid diaphragm in the model. The lateral force at each floor, as generated by earthquake and wind loading, acts at the center of rigidity of each floor which is modeled as rigid floor diaphragm. The center of mass of each floor is defined as the mean location of the mass system of each floor. The mass of the floor is assumed to be concentrated at this point when the floor is modeled as rigid diaphragm. The distance between these two is the lever arm for the natural torsion moment for seismic loads when that option is used.

5.43 Stress/Force output printing for Surface Entities

Default locations for stress/force output, design, and design output for surface elements are set as follows:

SURFACE DIVISION X xd

SURFACE DIVISION Y yd

Where:

xd = number of divisions along X axis,

yd = number of divisions along Y axis.

xd and yd represent default numbers of divisions for each edge of the surface where output is requested. The output is provided for sections located between

division segments. For example, if the number of divisions = 2, then the output will be produced for only one section (at the center of the edge).

Values of internal forces may be printed out for any user-defined section of a wall modeled using the Surface element.

General Format

**PRINT SURFACE FORCE (ALONG {X | Y}) (AT a)
(BETWEEN d₁ d₂) LIST s₁ s₂ ... s_i**

Where:

a = distance along the local axis from start of the member to the full cross-section of the wall,

d₁ d₂ = coordinates in the direction orthogonal to ξ , delineating a fragment of the full cross-section for which the output is desired.

s₁ s₂ ... s_i = list of surfaces for output generation

ALONG specifies local axis of the surface element

Notes

- If the keyword ALONG is omitted, direction Y (default) is assumed.
- If command AT is omitted, output is provided for all sections along the specified (or default) edge. Number of sections will be determined from the SURFACE DIVISION X or SURFACE DIVISION Y input values.
- If the BETWEEN keyword is omitted, the output is generated based on full cross-section width.

The attributes associated with the surface element, and the sections of this manual where the information may be obtained, are listed below:

Attributes	Related Sections
Surfaces Incidences	5.13.3
Openings in surface	5.13.3

Section 5 Commands and Input Instructions

5.44 Printing Section Displacements for Members

Attributes	Related Sections
Local Coordinates system for surfaces	1.6.3
Specifying sections for stress/force output	5.13.3
Property for surfaces	5.21.2
Material constants	5.26.3
Surface loading	5.32.3.4
Stress/Force output printing	5.42
Shear Wall Design	3.8.2, 5.55

5.44 Printing Section Displacements for Members

This command is used to calculate and print displacements at sections (intermediate points) of frame members. This provides the user with deflection data between the joints.

General Format

```
PRINT SECTION (MAX) DISPLACEMENTS (NSECT i) (SAVE a) {  
  NOPRINT | ALL | LIST memb-list }
```

Where:

i = number of sections to be taken. Defaults to 12 if **NSECT** is not used and also if **SAVE** is used (max=24, min=2).

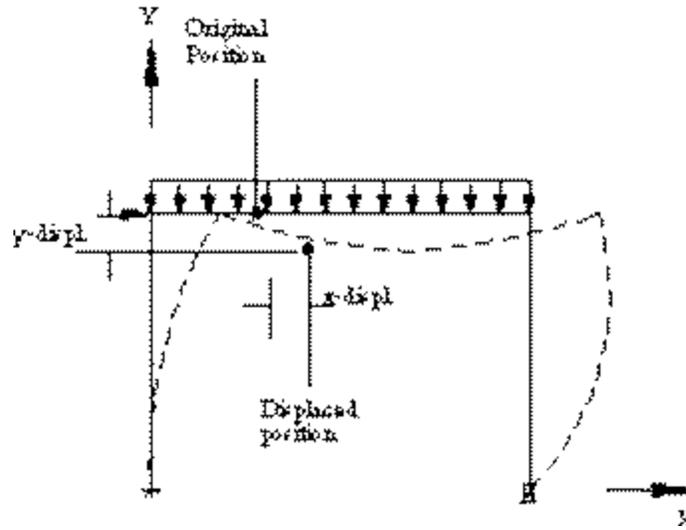
a = File name where displacement values can be stored and used by the STAADPL graphics program. If the **NOPRINT** command is used in conjunction with the **SAVE** command, the program writes the data to file only and does not print them in the output.

Note: This option is not necessary in STAAD.Pro.

Description

This command prints displacements at intermediate points between two joints of a member. These displacements are in global coordinate directions (see figure). If the MAX command is used, the program prints only the maximum local displacements among all load cases.

Figure 5-55: Displacements in the global coordinate directions



Example

```
PRINT SECTION DISPL SAVE
PRINT SECTION MAX DISP
```

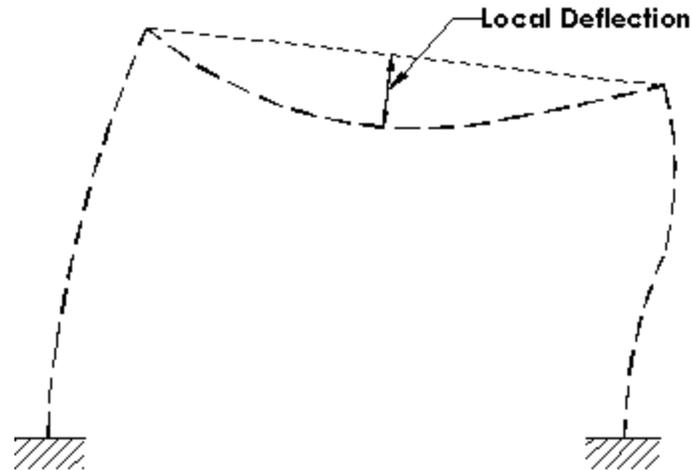
SECTION DISPLACEMENTS are measured in **GLOBAL COORDINATES**. The values are measured from the original (undeflected) position to the deflected position. See figure above.

The maximum local displacement is also printed. First, the location is determined and then the value is measured from this location to the line joining start and end joints of the deflected member.

Figure 5-56: Local deflection

Section 5 Commands and Input Instructions

5.45 Printing the Force Envelope



Notes

- The section displacement values are available in Global Coordinates. The undeflected position is used as the datum for calculating the deflections.
- This is a secondary analysis command. An analysis must be performed before this command may be used.

5.45 Printing the Force Envelope

This command is used to calculate and print force/moment envelopes for frame members. This command is not available for finite elements.

General Format

```
PRINT { FORCE | MAXFORCE } ENVELOPE (NSECTION i) list-spec  
list-spec = { LIST memb-list | (ALL) }
```

Description

Where:

i = the number of equally spaced sections to be considered in printing maximum and minimum force envelopes (where i is in the range 2 through 96). If the NSECTION i command is omitted, i will default to 12.

MAXFORCE command produces maximum/minimum force values only of all sections, whereas the FORCE command prints maximum/minimum force values at every section as well as the max/min force values of all sections. The force

components include FY, MZ, FZ, and MY. The SECTION command (as described in section 5.40) does not define the number of sections for force envelopes. For the sign convention of force values, refer to Section 1.19

Note: This is a secondary analysis command and should be used after analysis specification.

Example

```
PRINT FORCE ENV
PRINT MAXF ENV NS 15
PRINT FORCE ENV NS 4 LIST 3 TO 15
```

5.46 Post Analysis Printer Plot Specifications

This command has been discontinued in STAAD.Pro. Please use the facilities of the Graphical User Interface (GUI) for screen and hard copy graphics.

5.47 Size Specification

This command provides an estimate for required section properties for a frame member based on certain analysis results and user requirements.

General Format

```
SIZE *{ WIDTH f1 | DEFLECTION f2 | LENGTH f3 | BSTRESS f4 |
SSTRESS f5 } { MEMBER memb-list | ALL }
```

Where:

f₁ = allowable width

f₂ = Maximum allowable (Length/Maxm. local deflection) ratio

f₃ = Length for calculating the above ratio. The default value is the actual member length.

f₄ = Maximum allowable bending stress.

f₅ = Maximum allowable shear stress.

The values must be provided in the current unit system.

Section 5 Commands and Input Instructions

5.48 Steel and Aluminum Design Specifications

Description

This command may be used to calculate required section properties for a member based on analysis results and user specified criteria. The user specified criteria may include Member width, Allowable (Length/Maxm. Deflection) Ratio, Maxm. allowable bending stress and Maximum allowable shear stress. Any number of these criteria may be used simultaneously. The output includes required Section Modulus (about major axis), required Shear Area (for shear parallel to minor axis), Maxm. moment capacity (about major axis), Maxm. shear capacity (for shear parallel to minor axis) and Maxm. (Length/local maxm. deflection) ratio.

Example

```
SIZE WID 12 DEFL 300 LEN 240 BSTR 36 ALL
SIZE DEFL 450 BSTR 42 MEMB 16 TO 25
```

Note: It may be noted that sizing will be based on only the criteria specified by the user in the relevant **SIZE** command.

In the first example above, sizing will be based on user specified member width of 12, Length/Deflection ratio of 300 (where Length= 240) and max. allowable bending stress of 36.

In the second example, sizing will be based on Length/Deflection ratio of 450 (where Length= actual member length) and max. allowable bending stress of 42.

Note: This is a post-analysis facility and must be used after the analysis specifications.

5.48 Steel and Aluminum Design Specifications

This section describes the specifications necessary for structural steel & aluminum design.

The specific details of the implementation of these codes may be found in the following places:

- American AISC ASD & AISC LRFD - Section 2 of this manual
- AASHTO - Section 2 of this manual
- AISI (American Cold formed steel code) - Section 2 of this manual

- All other countries - International Codes Manual
- American Aluminum specifications - International Codes Manual
- ASCE 10-97 Transmission Tower code - International Codes Manual

Section 5.49.1 discusses specification of the parameters that may be used to control the design. Sections 5.49.2 and 5.49.3 describe the **CODE CHECKING** and **MEMBER SELECTION** options respectively. Member Selection by optimization is discussed in 5.49.4.

STAAD also provides facilities for Weld Design per the American welding codes. Details may be found in sections 2.12 and 5.49.5 of this manual.

5.48.1 Parameter Specifications

This set of commands may be used to specify the parameters required for steel and aluminum design.

General Format

PARAMETER

CODE *design-code*

{ *parameter-name* f_1 | PROFILE a_1 (a_2 a_3) } { MEMBER *memb-list* | ALL | *member-group-name* | *deck-name* }

Table 5-34:
design-code =

<u>AASHTO</u>	<u>JAPAN</u>	<u>ASCE</u>
<u>AISC</u>	<u>LRFD</u>	<u>DUTCH</u>
<u>AISI</u>	<u>NORWAY</u>	<u>NPD</u>
<u>ALUMINUM</u>	<u>BS5400</u>	<u>DANISH</u>
<u>S136</u>	<u>RUSSIA</u>	<u>BSK94</u>
<u>AUSTRALIAN</u>	<u>LRFD2</u>	<u>FINNISH</u>
<u>BRITISH</u>	<u>API</u>	<u>IS801</u>
<u>CANADIAN</u>	<u>TIMBER</u>	<u>IS802</u>
<u>FRENCH</u>	<u>SP</u>	<u>MEXICAN</u>
<u>GERMAN</u>	<u>ANISH</u>	<u>BAS59520 1990</u>
<u>INDIA</u>	<u>CHINA</u>	
	<u>EN1993</u>	

Section 5 Commands and Input Instructions

5.48 Steel and Aluminum Design Specifications

Please see section 5.16 for the definition of member group names. Deck names are explained in section 5.20.7.

parameter-name = refers to the PARAMETER NAME (s) listed in the parameter table contained in the Steel and Aluminum Design section.

f_1 = Value of the parameter. Not all parameters require values.

Description

The details of the parameters available for specific codes may be found in the following places:

- American AISC ASD & AISC LRFD - Section 2 of this manual
- AASHTO - Section 2 of this manual
- AISI (American Cold formed steel code) - Section 2 of this manual
- All other countries - International Codes Manual
- American Aluminum specifications - International Codes Manual
- ASCE72 Transmission Tower code - International Codes Manual
- API code - International Codes Manual
- ANSI/AISC N690 - International Codes Manual
- ASME NF 3000 - International Codes Manual

You can control the design through specification of proper parameters.

The **PROFILE** parameter is available for only a limited number of codes like the AISC ASD and AISC LRFD. You can specify up to three profiles (a1, a2 and a3). Profile is the first three letters of a section name from its steel table, like, W8X, W12, C10, L20 etc. The PROFILE parameter-name is used only for member selection where members are selected from each of those profile names. The **PROFILE** for a T-section is the corresponding W-shape. Also, the shape specified under **PROFILE** has to be the same as that specified initially under **MEMBER PROPERTIES**.

The **CODE** parameter let you choose the type of steel or aluminum code to be checked for design. The default steel code depends on the country of distribution.

Example

```
PARAMETERS
CODE AISC
```

```

KY 1.5 MEMB 3 7 TO 11
NSF 0.75 ALL
PROFILE W12 W14 MEMB 1 2 23
RATIO 0.9 ALL

```

Notes

- All unit sensitive values should be in the current unit system.
- For default values of the parameters, refer to the appropriate parameter table.

5.49 Code Checking Specification

Used to perform the **CODE CHECKING** operation for steel and aluminum members.

This command checks the specified members against the specification of the desired code. Refer to Section 2 of this manual for detailed information.

General Format

```

CHECK CODE { MEMBER memb-list | ALL | member-group-name
| deck-name | PMEMB pmember-list }

```

Example

```

CODE CHECK MEMB 22 TO 35
CODE CHECK PMEMB 1 3
CODE CHECK _BEAMS
CODE CHECK ALL
CODE CHECK PMEMB ALL

```

Notes

- The output of this command may be controlled using the **TRACK** parameter. Various codes support various levels of details. Refer to the appropriate section of the documentation, as explained in the table in section 5.48.1 for more information on the **TRACK** parameter.

Section 5 Commands and Input Instructions

5.49 Code Checking Specification

- b. Member group names and deck names are explained in section 5.16 and 5.20.7 respectively.
- c. The **PMEMB** list is explained in section 5.16.2.

5.49.1 Member Selection Specification

This command may be used to perform the MEMBER SELECTION operation.

This command instructs STAAD to select specified members based on the parameter value restrictions and specified code. The selection is done using the results from the most recent analysis and iterating on sections until a least weight size is obtained. Refer to Section 2 for more details.

General Format

```
SELECT { MEMBER memb-list | ALL | member-group-name | deck-name | PMEMB pmember-list }
```

It is important that the keywords **MEMBER**, **ALL**, or **PMEMB** be provided. Thus, the keyword **SELECT** by itself is not sufficient.

Example

```
SELECT MEMB 22 TO 35  
SELECT PMEMB 1 3  
SELECT _COLUMNS  
SELECT ALL  
SELECT PMEMB ALL
```

Notes

- a. The output of this command may be controlled using the **TRACK** parameter. Various codes support various levels of details. Refer to the appropriate section of the documentation, as explained in the table in section 5.48.1 for more information on the **TRACK** parameter.
- b. Member selection can be done only after an analysis has been performed. Consequently, the command to perform the analysis has to be specified before the **SELECT MEMBER** command can be specified.
- c. This command does not cause the program to re-analyze for results based

on the selected member sizes. However, to maintain compatibility of analysis results with the final member sizes, you should enter a subsequent PERFORM ANALYSIS command. Otherwise the post processor will display the prior results with the revised member sizes.

- d. Member group names and deck names are explained in section 5.16 and 5.20.7 respectively.
- e. The **PMEMB** list is explained in section 5.16.2.

5.49.2 Member Selection by Optimization

This command performs member selection using an optimization technique based on multiple analysis/design iterations.

The program selects all members based on an optimization technique. This method performs 2 analyses as well as iteration of sizes to reduce the overall structure weight. This command should be used with caution since it will require longer processing time.

General Format

SELECT OPTIMIZED

Notes

- a. The output of this command may be controlled using the TRACK parameter. Three levels of details are available. Refer to the appropriate Steel Design section for more information on the TRACK parameter.
- b. This command will perform 1 additional analysis/design cycles and therefore may be time consuming. Steps taken are: CHECK CODE ALL; then modify ratios; then SELECT ALL; then PERFORM ANALYSIS; then SELECT ALL. See section 5.49 Group Specification for other options used with this command. You may want to repeat this command for further optimization.
- c. See section 5.48.3 note 3.

5.49.3 Weld Selection Specification

This command performs selection of weld sizes for specified members.

By this command, the program selects the weld sizes of the specified members at start and end. The selections are tabulated with all the necessary information. If the TRUSS command is used, the program will design welds for angle and double

Section 5 Commands and Input Instructions

5.50 Group Specification

angle members attached to gusset plates with the weld along the length of members.

General Format

```
SELECT WELD (TRUSS) { MEMBER memb-list | ALL }
```

Notes

Currently, this feature is available only with the American welding specifications. Regardless of the code name specified prior to this command, design will be performed according to US code specifications. Some information on the implementation of these specifications is available in Section 2.12 of this manual. The parameters available for controlling weld sizes are listed in Table 2.1 in Section 2 of this manual.

Note: It is important that the keywords **MEMBER** or **ALL** be provided. Thus, the keyword **SELECT** by itself is not sufficient.

Example

```
SELECT WELD TRUSS MEMB 22 TO 35  
SELECT WELD ALL
```

5.50 Group Specification

This command may be used to group members together for analysis and steel design.

General Format

```
(FIXED GROUP)  
GROUP prop-spec MEMBER memb-list (SAME AS i1)  
prop-spec = { AX | SY | SZ }
```

Where:

*i*₁ = member number used in the SAME AS command. If provided, the program will group the members based on the properties of *i*₁.

Description

This command enables the program to group specified members together for analysis based on their largest property specification.

When **FIXED GROUP** is omitted, the **GROUP** command is usually entered after the member selection command, and the selected members will be grouped immediately and the new member properties will be used in any further operations. After the grouping is completed, the **GROUP** commands are discarded and will not be used again. Further grouping will be done only if a new **GROUP** command is encountered later.

If the **FIXED GROUP** option precedes the group data, the specified grouping will be retained in memory by the program and will be used in subsequent **SELECT** commands. No grouping will occur unless a **SELECT** (MEMBER or ALL or OPTIMIZED) command is performed. However, grouping will be performed with every subsequent **SELECT** command.

Example 1

```
SELECT ALL
GROUP SZ MEMB 1 3 7 TO 12 15
GROUP MEMB 17 TO 23 27 SAME AS 30
```

In this example, the members 1, 3, 7 to 12, and 15 are assigned the same properties based on which of these members has the largest section modulus. Members 17 to 23 and 27 are assigned the same properties as member 30, regardless of whether member 30 has a smaller or larger cross-sectional area. AX is the default property upon which grouping is based.

Example 2

```
FIXED GROUP
GROUP MEMB 1 TO 5
SELECT OPTIMIZED
```

In the above example, the usage of the **FIXED GROUP** command is illustrated. In this example, the **SELECT OPTIMIZED** command involves the six stage process of

Section 5 Commands and Input Instructions

5.51 Steel and Aluminum Take Off Specification

1. CHECK CODE ALL followed by modification of RATIO
2. SELECT ALL
3. GROUPING MEMBERS 1 TO 5
4. PERFORM ANALYSIS
5. SELECT ALL
6. GROUPING MEMBERS 1 TO 5

The **FIXED GROUP** command (and the **GROUP** commands that follow it) is required for execution of steps 3 and 6 in the cycle. You may want to repeat this data for further optimization.

Notes

The **FIXED GROUP + GROUP** commands are typically entered before the member selection for further analysis and design. This facility may be effectively utilized to develop a practically oriented design where several members need to be of the same size.

All the members in a list for a specific **GROUP** command should have the same cross section type. Thus, if the command reads

GROUP MEMB 1 TO 10

and member 3 is a W shape and member 7 is a Channel, grouping will not be done. The 10 members must be either all W shapes or all channels.

Also see section 5.48.3, note 3.

5.51 Steel and Aluminum Take Off Specification

This command may be used to obtain a summary of all steel sections and aluminum sections being used along with their lengths and weights (quantity estimates).

This command provides a listing of the different steel and aluminum table sections used in the members selected. For each section name, the total length and total weight of all members which have been assigned that section will be listed in a tabular form. This can be helpful in estimating steel and aluminum quantities.

General Format

```
{ STEEL | ALUMINUM } (MEMBER) TAKE (OFF) ( { LIST memb-list
| LIST membergroupname | ALL } )
```

If the **MEMBER** option is specified, the length and weight of each member and the section name it is assigned will be reported.

Example

```
STEEL TAKE OFF LIST 71 TO 85
ALUMINUM TAKE OFF LIST_PLGN03
```

5.52 Timber Design Specifications

This section describes the specifications required for timber design. Detailed description of the timber design procedures is available in Section 4.

5.52.1 Timber Design Parameter Specifications

This set of commands may be used for specification of parameters for timber design.

General Format

```
PARAMETER
CODE design-code
parameter-name  $f_1$  { MEMBER member-list | ALL }
```

Where:

f_1 = the value of the parameter.

Table 5-35:

design-code =

TIMBER	AITC 1984	TIMBER CANADIAN
AITC	AITC 1994	TIMBER EC5

parameter-name= refers to the parameters described in Section 4.

Section 5 Commands and Input Instructions

5.52 Timber Design Specifications

Notes

- a. All values must be provided in the current unit system.
- b. For default values of parameters, refer to Section 4.

5.52.2 Code Checking Specification

This command performs code checking operation on specified members based on the American Institute of Timber Construction (AITC), Canadian Standards Agency (CSA), or Eurocode (EC5) codes.

The results of the code checking are summarized in a tabular format. Examples and detailed explanations of the tabular format are available in Section 4.

General Format

```
CHECK CODE { MEMBER member-list | ALL }
```

Note: The output of this command may be controlled by the **TRACK** parameter. Two levels of details are available. Refer to Section 4 for detailed information on the **TRACK** parameter.

5.52.3 Member Selection Specification

This command performs member selection operation on specified members based on the American Institute of Timber Construction (AITC 1984) code.

This command may be used to perform member selection according to the AITC 1984 code. The selection is based on the results of the latest analysis and iterations are performed until the least weight member satisfying all the applicable code requirements is obtained. Parameters may be used to control the design and the results are available in a tabular format. Detailed explanations of the selection process and the output are available in Section 4.

General Format

```
SELECT { MEMBER memb-list | ALL }
```

Note: The output of this command may be controlled by the **TRACK** parameter. Two levels of details are available. Refer to Section 4 for detailed information on the **TRACK** parameter.

5.53 Concrete Design Specifications

This section describes the specifications for concrete design for beams, columns and individual plate elements. The concrete design procedure implemented in STAAD consists of the following steps:

5.53.1 Design Initiation

This command is used to initiate concrete design for beams, columns and individual plate elements.

This command initiates the concrete design specification. With this, the design parameters are automatically set to the default values (as shown on Table 3.1). Without this command, none of the following concrete design commands will be recognized.

Note: This command must be present before any concrete design command is used.

General Format

START CONCRETE DESIGN

5.53.2 Concrete Design-Parameter Specification

This set of commands may be used to specify parameters to control concrete design for beams, columns and individual plate elements.

General Format

CODE *design-code*

parameter-name f_1 { MEMBER *member-list* | ALL }

Section 5 Commands and Input Instructions

5.53 Concrete Design Specifications

Table 5-36:
design-code =

<u>ACI</u>	<u>FRE</u>
<u>AUS</u>	<u>NCH</u>
<u>TRALIAN</u>	<u>GER</u>
<u>BRITISH</u>	<u>MAN</u>
<u>CANADIAN</u>	<u>IND</u>
<u>CHINA</u>	<u>IA</u>
<u>EUROPE</u>	<u>JAP</u>
	<u>AN</u>
	<u>MEX</u>
	<u>ICO</u>
	<u>NOR</u>
	<u>WAY</u>

Where:

f_1 = is the value of the parameter. Wherever applicable, this value is input in the current units. The UNIT command is also accepted during any phase of concrete design.

parameter-name = refers to the concrete parameters described in Table 3.1 for the ACI code and in various corresponding concrete chapters of the International Design Codes Manual for all other codes.

Notes

- All parameter values are provided in the current unit system.
- For default values of parameters, refer to Section 3.4 for the ACI code. For other codes, please see the International Codes manual.

5.53.3 Concrete Design Command

This command may be used to specify the type of design required. Members may be designed as BEAM, COLUMN or ELEMENT.

Members to be designed must be specified as **BEAM**, **COLUMN**, or **ELEMENT**.

Members, once designed as a beam, cannot be redesigned as a column again, or vice versa.

General Format

DESIGN { BEAM | COLUMN | ELEMENT } { *memb-list* | (ALL) }

Notes

- Only plate elements may be designed as **ELEMENT**.
- Enter this command after the parameters needed for this command have been entered.
- The DESIGN ELEMENT command designs individual plate elements using the procedure explained in section 3.8.1 for the ACI code. For theoretical information on designing individual plate elements per other design codes, please see the International Codes manual. For designing shear walls and slabs, see sections 3.8.2, 3.8.3 and 5.55 of this manual.

5.53.4 Concrete Take Off Command

This command may be used to obtain an estimate of the total volume of the concrete, reinforcement bars used and their respective weights.

This command can be issued to print the total volume of concrete and the bar numbers and their respective weight for the members designed.

General Format

CONCRETE TAKE OFF

Sample Output

```

***** CONCRETE TAKE OFF *****
      (FOR BEAMS, COLUMNS AND PLATES DESIGNED ABOVE)
NOTE: CONCRETE QUANTITY REPRESENTS VOLUME OF CONCRETE IN BEAMS, COLUMNS, AND PLATES
DESIGNED ABOVE.
      REINFORCING STEEL QUANTITY REPRESENTS REINFORCING STEEL IN BEAMS AND COLUMNS
DESIGNED ABOVE.
      REINFORCING STEEL IN PLATES IS NOT INCLUDED IN THE REPORTED QUANTITY.
TOTAL VOLUME OF CONCRETE =          4.4 CU. YARD
BAR SIZE          WEIGHT
NUMBER          (in lbs)
-----          -----
      4              261
      5              87
      6              99
      8             161
      9             272
      *** TOTAL=          880

```

Section 5 Commands and Input Instructions

5.54 Footing Design Specifications

Note: This command may be used effectively for quick quantity estimates.

5.53.5 Concrete Design Terminator

This command must be used to terminate the concrete design.

This command terminates the concrete design, after which normal STAAD commands resume.

Note: Without this command, further STAAD commands will not be recognized.

General Format

```
END CONCRETE DESIGN
```

Example

```
START CONCRETE DESIGN  
CODE ACI  
FYMAIN 40.0 ALL  
FC 3.0 ALL  
DESIGN BEAM 1 TO 4 7  
DESIGN COLUMN 9 12 TO 16  
DESIGN ELEMENT 20 TO 30  
END
```

5.54 Footing Design Specifications

This feature has been removed from batch mode design. Please contact the Technical Support department for further information.

Footing design may be performed using STAAD.foundation via the Foundation Design mode in the graphical interface.

5.55 Shear Wall Design

STAAD performs design of reinforced concrete shear walls per the following codes: ACI 318-02 (American), BS 8110 (British), and IS 456:2000 (Indian). In order

to design a shear wall, it must first be modeled using the Surface element.

The attributes associated with the surface element, and the sections of this manual where the information may be obtained, are listed below:

Attributes	Related Sections
Surfaces Incidences	5.13.3
Openings in surface	5.13.3
Local Coordinates system for surfaces	1.6.3
Specifying sections for stress/force output	5.13.3
Property for surfaces	5.21.2
Material constants	5.26.3
Surface loading	5.32.3.4
Stress/Force output printing	5.42
Shear Wall Design	3.8.2, 5.55

Scope of the design is dependent on whether panel definition, as explained in section 5.55.1, precedes the shear wall design input block which is explained in section 5.55.2.

- a. No panel definition.

Design is performed for the specified horizontal full cross-section, located at a distance c from the origin of the local coordinates system.

- b. Panels have been defined.

Design is performed for all panels, for the cross-section located at a distance c from the start of the panel.

Shear Wall design is not available for dynamic load cases.

5.55.1 Definition of Wall Panels for Shear Wall Design

Due to the presence of openings, three types of structural elements may be defined within the boundaries of a shear wall: wall, column, and beam. For each of those entities, a different set of design and detailing rules applies. The types to panels – functionally different parts of the shear wall – are manually assigned. The assignment is based on panel geometry, its position, and overall wall configuration. Wall, column, and beam panels are then designed in accordance with relevant provisions of the code.

General Format

START PANEL DEFINITION

SURFACE *i* **PANEL** *j* **ptype** *x1 y1 z1 x2 y2 z2 ... xn yn zn*

END PANEL DEFINITION

Where:

i

ordinal surface number

j

ordinal panel number

ptype

panel type. For the ACI code, the types available are WALL, COLUMN, and BEAM (for ACI 318 and IS 456). For the British code, only the WALL type is available.

x1 y1 z1 (...)

coordinates of the corners of the panel

5.55.2 Shear Wall Design Initiation

Use to initiate a concrete shearwall design.

General Format

START SHEARWALL DESIGN

CODE { **ACI** | **BRITISH** | **INDIAN** }

parameter_name *f₁*

DESIGN SHEARWALL (AT *f₂*) LIST *wall_list*

```

CREINF  $i_1$ 
TRACK  $i_2$ 
END SHEARWALL DESIGN

```

Codes choices are **ACI** (for ACI 318) or **BRITISH** (for BS 8110).

Where:

parameter-name = refers to the PARAMETER NAME (s) listed in a tabular form in section 3.8.2 of this manual for the ACI code, and in the International Codes manual for BS 8110 and IS 456:2000.

f_1 = Value of the parameter. Not all parameters require values.

f_2 = optional location at which design is to be performed

Note: If the command **AT** is omitted, the design proceeds for all cross sections of the wall or panels, as applicable, defined by the **SURFACE DIVISION X** or **SURFACE DIVISION Y** input values.

i_1 = column reinforcing parameter. Reinforcement distribution within COLUMN panels is controlled by parameter **CREINF**, that may have one of three possible values:

- o. equal number of bars on all four faces of the column (default),
- 1. equal number of bars on two faces perpendicular to the plane of the wall,
- 2. equal number of bars on two faces parallel to the plane of the wall.

i_2 = output parameter TRACK specifies how detailed the design output should be:

- o. indicates a basic set of results data (default),
- 1. full design output will be generated.

5.56 End Run Specification

This command must be used to terminate the STAAD run.

This command should be provided as the last input command. This terminates a STAAD run.

Section 5 Commands and Input Instructions

5.56 End Run Specification

General Format

FINISH

Index

A

Accidental Torsion 606
ACI Beam Design 230
Advanced Solver 274
AIJ (Japanese) 446
AISC-ASD
 Composite Beam Design 133
AISC 360
 ASD 104
 LRFD 103
 Parameters 106
Algerian 469
Allowables per AISC Code 112
Aluminum
 Take Off ReportSee Take Off
 Specification
American LRFD 3rd edition code
 Composite Beam Design
 per 146

Analysis

Buckling 684
Compression Only 77
Dynamic 80
Response Spectrum 82, 586
Secondary 92
Tension Only 77
Time History 84
Analysis Consideration 224
Analysis Facilities 63
Analysis Requirements 138
Anchorage 226
 Design 226
Applying 363
 Fireproofing 363
Area Load 55, 555
Assign 41, 350
 Profile Specification 350
Assigning Material
 Constants 385

Automatic Spring Support Generator

 Foundations 409

Axial Compression 139

Axial Tension 139

B

Beam Design 225, 227

Bending 112-113, 140

 Stress 112

Boundary Conditions

 Torsional 115

Buckling Analysis 73, 684

C

Cable Member

 Linearized 47

Cable Members

 Nonlinear 49

Cable Specification See Member,
 Cable Specification

Castellated Beams 152

Center of Gravity See also Print
 CG

 Print 95

Center of Rigidity 606

Center of Stiffness See Center of
 Rigidity

CFE 462

Change Specification 709

Code Checking 131, 145, 262

 Specification 731

Colombian Seismic Load 444

Combined Axial Force 140

Combined Compression 113

Comision Federal De Elec-
 tricidad 462

Command Formats 276

Command Language

 Separator 294

Command Language Con-
 ventions 275

Composite Beam Design

 AISC-ASD 133

Composite Beam Design
 per 146

Composite Beams 45

Composite Damping 82, 401

 Springs 401

Composite Decks 45, 351

Compression-Only

 Analysis See Analysis, Com-
 pression Only

Compression-Only

 Members See Member,
 Compression

Concrete Design 218

 Section Types 218

Concrete Design Command 740

Concrete Design

 Specifications 739

Concrete Design

 Terminator 742

Concrete Take Off

 Command 741

Cracked Moment 230
 Inertia 230
 Cracked Section Properties 367
 Curved Members 46, 355
 Cut-Off Frequency 427

D

Damping
 Composite 82
 Modeling 82
 Springs 401
 Define Harmonic Output
 Frequencies 691
 Definition 386, 429, 433, 503,
 530
 Load Systems 429
 Material Constants 386
 Moving Load System 429
 Snow Load 530
 Static Force Procedures 433
 Time History Load 507
 Wind Load 503
 Deflection Check With Steel
 Design 133
 Delete Member 324
 Deprecated Commands
 Area Load 555
 Design 116, 140, 225-226, 236,
 244
 Anchorage 226
 elements 236
 Flexure 225

I-shaped beams per ACI-
 318 244
 Shear 140, 225
 Web Tapered Sections 116
 Design Initiation 739
 Design Operations 97, 217, 250
 Design Parameters 117, 140, 220
 Design per American Cold
 Formed Steel Code 209
 Design Specifications 296
 Direct Analysis 686
 Member Definition 534
 Discontinued Commands
 Draw 427
 Printer Plot 727
 Problem Statistics 676
 Displacements 92
 Draw 427
 Drift 721
 Dynamic Analysis 80, 427
 Miscellaneous Settings 427
 Dynamic Loading
 Specification 586

E

Eccentricity 606
 Eigenproblem 80
 Solution 80
 Element Design 236
 Element Ignore Stiffness 374
 Element Loads 59, 544
 Joints 549

Plates 544
 Solids 547
 Element Mesh Generation 310
 Element Plane Stress 382
 Element Property
 Specification 369
 Element Release
 Specification 373
 Element Releases 46, 370
 Element/Surface Property Spec-
 ification 368
 Elements 236, 275, 303
 Designing 236
 End Run 745
 Examples 351

F

Finish 745
 Finite Elements 21, 33
 Fireproofing 363
 Applying 363
 Fixed-End Load
 Specification 579
 Fixed End Member Load 57
 Flexural Design Strength 140
 Flexure 225
 Design 225
 Floor Load 562
 Floor Spectrum Response 644
 Force Envelopes 92
 Foundations 409

Frequencies
 Max. Number 282

G

Gamma Angle 357
 General Comments 136
 Generation 651
 Loads 651
 Geometric Nonlinear
 Analysis 75, 703
 Global Coordinate System 10
 Global Support
 Specification 405
 Groups 318, 734
 Specification 318
 GUI 95

H

Harmonic Force Loading 697
 Harmonic Ground Motion Load-
 ing 695
 Harmonic Loading Function 84
 Harmonic Response 85, 689

I

I-shaped beams per ACI-318 244
 Design 244
 Ignore Inplane Rotation Spec-
 ification 382
 Ignore Specifications 296
 Imperfection Analysis 76, 706
 Imperfection Information 401
 Inactive Member 324

Inactive Members 93

Inclined

- Joint Load Specification 540

Inclined Support

- Specification 408

Inertia 230

- Cracked Moment 230

Input Generation 7

Input Specification 252

Input Width 284

Inter-Story Drift See Drift

Intermediate Sections 92

Introduction 6, 273

IS 1893

- Floor Level Definitions 454
- Response Spectrum 598
- Seismic Load 449
- Soft Story 423, 452
- Story Drift 599

J

Japanese Seismic Load 446

Joint

- Element Load 549
- Redefinition 317

Joint Coordinates

- Specification 296

Joint Load 53

Joint Load Specification 540

Joint Numbers

- Renumbering See Redefinition

Joints

- Max Number 282

Joist Girders 42

L

Lamination

- Orientation 263

Last Line 702

Limits 282

Linearized Cable Members 47

Listing

- Members/Joints 318

Lists

- Groups 318

Load

- Selfweight 582, 584-585
- Temperature 578

Load Cases

- Max. Number 282

Load Combinations

- Max. Number 282

Load Generator 60-61

- Moving 61

Load List Specification 711

Load Systems 429

- Definition 429

Loads 53, 651

- Generation 651
- Specifications 539

Local Coordinate System 11

Local Coordinates 15

LRFD Fundamentals 137

M

Mass Modeling 80

 Reference Loads 533

Master-Slave Joints 53

Master/Slave Specification 420

Material Constants 52, 386

 Definition 386

Member

 Cable Specification 376

 Camber Imperfection 402

 Compression 47, 377

 DeleteSee Delete Member

 Dimensions 219

 Displacements 92

 Drift Imperfection 403

 End ForcesSee Member End
 Forces

 ForcesSee Member Forces

 GroupsSee Groups

 Imperfections 401

 InactiveSee Inactive Member

 Redefinition 317

 RenumberingSee Member,
 Redefinition

 Tension 47, 377

Member End Forces 87

Member Forces 92

Member Incidences Spec-
 ification 300

Member Load 54

Member Load Specification 541

Member Offset
 Specification 383

Member Offsets 50

Member Properties 98

Member Property
 Specification 339

 Examples 351

Member Release
 Specification 370

Member Releases 46, 370

Member Selection 132, 145, 262,
 732-733

Member Stresses 92

Member Truss Specification 375

Member Truss/Cable/Ten-
 sion/Compression Spec-
 ification 375

Members 363

 Max. Number 282

Miscellaneous Facilities 94

Miscellaneous Settings 427
 Dynamic Analysis 427

Modal Calculation
 Command 672

Modal Damping
 Information 396

Mode Selection 428

Mode Shapes 427

Modeling
 Damping 82

Modes

Max. Number 282

Moving 61

Load Generator 61

Moving Load System 429

Definition 429

Multi-linear Spring Support 77

Multilinear Spring Support 414

Multiple Analyses 93

N

Natural Torsion 722

Node

Element Load 549

Non Linear Cable Members 49

Non Linear Cable/Truss

Analysis 78

Nonlinear Analyses

Geometric 75, 703

Notional Loads 665

O

Oneway Load 556

Optimization 132, 733

Orientation

Lamination 263

Output

Description 227

Output Width 284

P

P-Delta

Dynamic Analysis 71

KG 681

Large Delta 678

Small Delta 678

Stress Stiffening 70, 681

P-Delta Analysis 67-68, 678

Page Length/Eject

Command 295

Page New 295

Parameter Specifications 729

Partial Moment Release 372

Plate 303

Element Loads 544

Plate Element 21

Plate Girders 134

Plates

Max. Number 282

Plotting

Facilities 94

Post Analysis Plot 727

Post Processing 95

Poststress Member Load 57

Prestress 57

Prestress Load Specification 573

Print CG 714

Print Force Envelope Spec-
ification 726

Print Section Displacement 724

Print Specifications 714
 Print Steady State/Harmonic
 Results 699
 Printer Plot 727
 Printing
 Facilities 94
 Prismatic Properties 38
 Prismatic Property
 Specification 346
 Prismatic Tapered Tube 347
 Property Specification 350
 Purpose 689

R

Rayleigh Frequency
 Calculation 670
 RC Designer 244
 Redefinition
 Joint 317
 Member 317
 Reduced Section Properties 367
 Reference Loads 531
 Relationship
 Global and Local
 Coordinates 15
 Repeat Load Specification 647
 Response History Analysis 642
 Time Varying Load 642
 Response Spectrum 82, 586
 Eurocode 8-1996 613
 Eurocode 8-2004 621
 Generic 587

IBC 2006 629
 IS 1893-2002 (Indian) 598
 SNiP (Russian) 636
 Response Time History 83
 Rigidity
 Center ofSee Center of Rigid-
 ity
 Rotation
 Structure GeometrySee Struc-
 ture Geometry, Rota-
 tion
 RPA (Algerian) 469

S

Second Order Analysis 67
 Secondary AnalysisSee Redef-
 inition
 Section Classification 138
 Section Properties
 Cracked 367
 Reduced 367
 Section Specification 713
 Section Types 218
 Concrete Design 218
 Seismic Analysis 433
 Seismic Load 462, 469
 AIJ 446
 Columbian 444
 Japanese 446
 UBC 1985 438
 UBC 1994 438
 UBC 1997 434

Select Member 732
 Selfweight 582, 584-585
 Separator See Command Language, Separator
 Set 284
 Shear 140, 225
 Design 140, 225
 Shear Stress 112
 Shear Wall Design 238, 742
 Initiation 744
 Wall Panels 744
 Shear walls 236
 Shell Element 21
 Shell Element Incidence Specification 303
 Singly Symmetric Sections 114
 Size Specification 727
 Slabs 236, 244
 Slender Compression Elements 116
 Slenderness Effects 224
 SNiP II-7-81 636
 Snow Load 530
 Definition 530
 Soft Story
 ASCE 7-95 423
 Check 423
 IS 1893-2002 423, 452
 Soil
 Modeled as Springs 77
 Solid Element 33

Solid Element Incidences Specification 305
 Solids
 Element Loads 547
 Max. Number 282
 Solution
 Eigenproblem 80
 Specification 539
 Loading 539
 Specified Sections 92
 Spring 401, 416
 Composite Damping 401
 Multilinear 414
 Tension/Compression Specification 416
 Spring Support 77
 Springs
 Compression Only 47
 Tension Only 47
 Static Force Procedures 433
 Definitions 433
 Steady Force Loading 693
 Steady Ground Motion Loading 692
 Steady State 85, 689
 Steady State/Harmonic Analysis 702
 Steel
 Take Off Report See Take Off Specification

Steel Design 146
 Tabulated Results 146
Steel Design per AISC/LRFD
 Specification 136
Steel Design Specifications 728
Steel Joist 42
Steel Section Library 40, 98, 341
Steel Sections
 Built-In Library 40, 98, 341
Stiffness
 Center ofSee Center of Rigid-
 ity
Stiffness Analysis 63
Story Drift
 IS 1893 599
Strain Load 59, 578
Stress 112
 Bending 112
Stress Due To Compression 112
Stress Stiffening Matrix 681
Stress/Force output 722
Structure Geometry
 Rotation 324
Structures
 Types 7
Substitute 95
Support Displacement Load 59
Support Joint Displacement
 Specification 580
Support Specifications 405
Supports 52

Surface Constants
 Specification 394
Surface Elements 36
 as Shear Walls 238
 Constants 394
 Loads 551
Surface Entities
 Specification 306
Surface Loads Specification 551
Surface Property
 Specification 369
Surfaces 303

 T
Tabulated Results 146
 Steel Design 146
Take Off Specification 736
Tapered Member
 Specification 349
Tapered Sections 41
Temperature 59
Temperature Load 578
Tension-Only AnalysisSee Anal-
 ysis, Tension Only
Tension-Only
 MembersSee Member, Ten-
 sion
Tension Stress 112
Tension/Compression Spec-
 ification 416
 Spring 416
The Commands
 Elements 275

Timber Code Checking 738
Timber Design Parameter Specification 737
Timber Design Specification 737
Timber Member Selection 738
Time 427
Time History Analysis 84
Time History Load 507
Time History Response 83
Time Varying Load 642
 Response History Analysis 642
Torsion
 FIXED BUT Support 115
Torsion per Publication T114 114
Truss Members 47, 133
Types
 Structures 7

U

UBC 1994 438
UBC 1997 Load Definition 434
UBC Seismic Load Generator 61
Units
 Force 282
 Length 282
 Systems of 9
Unsymmetric Sections 133
User Provided Steel Table 41, 326, 350
User Provided Table 350

User Steel Tabl 326

W

Wdith
 Input 284
Web Tapered Sections 116
 Design 116
Weld Selection Specification 733
Width
 Output 284
Wind Load 503
 Definition 503
Wind Load Generator 62

Index of

Commands

This following is an alphabetic list of all STAAD.Pro commands in the Technical Reference Manual.

A B C D E F G H I J K L M N O
P Q R S T U V W Y X Z

A, B

ALUMINUM MEMBER TAKE OFF
AREA LOAD

C

CALCULATE RAYLEIGH
CHANGE
CHECK CODE
CONCRETE TAKE OFF
CONSTANTS
CUT OFF

D

DEFINE 1893

DEFINE AIJ
DEFINE CFE
DEFINE COLUMBIAN LOAD
DEFINE DAMPING
DEFINE DIRECT
DEFINE ENVELOPE
DEFINE IBC
2000 OR 2003
2006
DEFINE IMPERFECTION
DEFINE MATERIAL
DEFINE MESH
DEFINE MOVING LOAD
DEFINE NRC
1995
2005
DEFINE NTC
DEFINE PMEMBER
DEFINE REFERENCE LOAD
DEFINE RPA

DEFINE SNOW LOAD
DEFINE TIME HISTORY
DEFINE UBC

1997

1994 or 1985

DEFINE WIND

DELETE

DESIGN

E

ELEMENT INCIDENCES

ELEMENT INCIDENCES SOLID

ELEMENT LOAD

JOINTS

PLATE

SOLID

ELEMENT PLANE STRESS

ELEMENT PROPERTY

ELEMENT RELEASES

END CONCRETE DESIGN

END PANEL DEFINITION

END SHEARWALL DESIGN

END STEADY

F

FIXED END LOAD

FLOOR DIAPHRAGM

FLOOR LOAD

FREQUENCY

G

GENERATE ELEMENT

GENERATE FLOOR SPECTRUM

GROUND MOTION

GROUP

H

HARMONIC FORCE

HARMONIC GROUND

I

IGNORE LIST

IGNORE STIFFNESS

INACTIVE

INPUT NODESIGN

INPUT WIDTH

J, K

JOINT COORDINATES

JOINT LOAD

L

LOAD COMBINATION

LOAD GENERATION

LOAD LIST

M

MATERIAL

MEMBER CABLE

MEMBER COMPRESSION

MEMBER CRACKED

MEMBER CURVED

MEMBER FIREPROOFING
 MEMBER INCIDENCES
 MEMBER LOAD
 MEMBER OFFSETS
 MEMBER PRESTRESS LOAD
 MEMBER PROPERTIES
 MEMBER RELEASES
 MEMBER TENSION
 MEMBER TRUSS
 MODAL CALCULATION
 MODE SELECT
 MULTILINEAR SPRINGS

N

NOTIONAL LOAD

O

ONEWAY LOAD
 OUTPUT WIDTH

P, Q

PAGE LENGTH
 PAGE NEW
 PARAMETER
 STEEL & ALUMINUM
 TIMBER
 CONCRETE
 PDELTA ANALYSIS
 PERFORM ANALYSIS
 PERFORM BUCKLING ANALYSIS
 PERFORM CABLE ANALYSIS
 PRINT DIAPHRAGM CR

PERFORM DIRECT ANALYSIS
 PERFORM IMPERFECTION
 ANALYSIS
 PERFORM NONLINEAR ANALYSIS
 PERFORM ROTATION
 PERFORM STEADY STATE
 ANALYSIS
 PRINT
 PRINT FORCE ENVELOPE
 PRINT HARMONIC
 DISPLACEMENTS
 PRINT SECTION DISPLACEMENTS
 PRINT STORY DRIFT
 PRINT STORY STIFFNESS
 PRINT SURFACE FORCE

R

REFERENCE LOAD
 REPEAT LOAD

S

SECTION
 SELECT (MEMBER)
 SELECT OPTIMIZED
 SELECT WELD
 SELFWEIGHT
 SEPARATOR
 SET
 SIZE
 SLAVE
 SNOW LAOD
 SPECTRUM

Index: –

1893 (Indian)

EURO, EURO 2004

IBC 2006

SPRING COMPRESSION

SPRING DAMPING

SPRING TENSION

STAAD

START DECK DEFINITION

START GROUP DEFINITION

START PANEL DEFINITION

START SHEARWALL DESIGN

START USER TABLE

STEADY FORCE FREQUENCY

STEADY GROUND FREQUENCY

STEEL MEMBER TAKE OFF

SUBSTITUTE

SUPPORTS

SURFACE CONSTANTS

SUPPORT DISPLACEMENT

SURFACE INCIDENCES

SURFACE LOAD

SURFACE PROPERTY

T

TEMPERATURE LOAD

TIME LOAD

U, V, W, X, Y, Z

UNIT

Technical Support

These resources are provided to help you answer support questions:

- Service Ticket Manager — <http://www.bentley.com/serviceticketmanager> — Create and track a service ticket using Bentley Systems' online site for reporting problems or suggesting new features. You do not need to be a Bentley SELECT member to use Service Ticket Manager, however you do need to register as a user.
- Knowledge Base — <http://appsnet.bentley.com/kbase/> — Search the Bentley Systems knowledge base for solutions for common problems.
- FAQs and TechNotes — http://communities.bentley.com/Products/Structural/Structural_Analysis___Design/w/Structural_Analysis_and_Design__Wiki/structural-product-tech-notes-and-faqs.aspx — Here you can find detailed resolutions and answers to the most common questions posted to us by you.
- Ask Your Peers — <http://communities.bentley.com/forums/5932/ShowForum.aspx> — Post questions in the Be Community forums to receive help and advice from fellow users.



Bentley Systems, Incorporated
685 Stockton Drive, Exton, PA 19341 USA
+1 (800) 236-8539
www.bentley.com