

# **STAAD.Pro 2006**

## SOFTWARE RELEASE REPORT

(Build 1002)

July, 2006

# Table of Contents

## STAAD.Pro 2006 Build 1002

AD.2006-1002.1 Features Affecting the Pre-Processor (Modelling Mode).....	5
AD.2006-1002.1.1 Automatic calculation of the Response Spectrum graph as per IBC... 6	
Purpose.....	6
Description.....	6
AD.2006-1002.1.2 Enhancement to Physical Member Query Window.....	11
Purpose.....	11
Description.....	11
AD.2006-1002.1.3 AAHSTO (LRFD) Design Code.....	14
Purpose.....	14
Description.....	14
AD.2006-1002.1.4 Node Precision.....	15
Purpose.....	15
Description.....	15
AD.2006-1002.1.5 User Steel Table – Double Angles.....	16
Purpose.....	16
Description.....	16
5.19 User Steel Table Specification:- .....	16
AD.2006-1002.2 Features Affecting the Analysis and Design Engine .....	18
AD.2006-1002.2.1 AAHSTO Design Code .....	19
Purpose.....	19
Description.....	19
2.13 Steel Design per AASHTO Specifications .....	20
2.13.1 AASHTO (ASD).....	21
General Comments.....	21
Allowable Stresses per AASHTO Code .....	21
Axial Stress .....	21
Bending Stress .....	22
Shear Stress.....	22
Bending-Axial Stress Interaction.....	23
AASHTO (ASD) Design Parameters.....	24
AASHTO (ASD) Verification Problem.....	26
VERIFICATION PROBLEM HAND CALCULATION.....	27
2.13.2 AASHTO (LRFD) .....	43
General Comments.....	43
Capacities per AASHTO (LRFD) Code .....	43
Axial Strength.....	43
AASHTO (LRFD) Design Parameters .....	47
AASHTO (LRFD) Verification Problem.....	49
VERIFICATION PROBLEM HAND CALCULATION.....	49
AD.2006-1002.2.2 Updated Parameters on Indian IS 1893 Static Seismic Loading Command.....	59

Purpose.....	59
Description.....	59
5.31.2.5    Definition of Lateral Seismic Load per Indian IS:1893 (Part 1) – 2002	
Code        60	
Purpose.....	60
Description.....	60
Methodology.....	60
Soft Storey Checking.....	62
Identification of floor level.....	63
Example.....	64
AD.2006-1002.2.3 Updated Parameters on Indian IS 1893 Response Spectrum	
Command.....	65
Purpose.....	65
Description.....	65
5.32.10.1.1 Response Spectrum Specification as per the Indian IS:1893 (Part 1)-2002	
Code for Dynamic Analysis.....	66
Methodology.....	66
Torsion.....	70
Soft Storey Checking.....	71
AD.2006-1002.3 Features Affecting the Steel Design Mode.....	72
AD.2006-1002.3.1 Access to Physical Members Created in Pre-Processing Mode. ....	73
Purpose.....	73
Description.....	73
AD.2006-1002.4 Features Affecting the Concrete Design Mode.....	74
AD.2006-1002.4.1 DXF Output of Beam and Column Designs. ....	75
Purpose.....	75
Description.....	75
AD.2006-1002.5 Features Affecting the Advanced Slab Design Mode.....	76
AD.2006-1002.5.1 Advance Slab Design Mode .....	77
Purpose.....	77
Description.....	77
AD.2006-1002.6 Features Affecting the Piping Mode.....	83
AD.2006-1002.6.1 Piping Mode.....	84
Purpose.....	84
Description.....	84
AD.2006-1002.6.2 3D Rendered View of Piping Model .....	90
Purpose.....	90
Description.....	90
AD.2006-1002.7 Additional STAAD.Pro 2006 Build 1001 Features.....	92
AD.2006-1002.7.1 Modifications in REI Security System .....	93
Purpose.....	93
Description.....	93
AD.2006-1002.7.2 Online Help.....	94
Purpose.....	94
Description.....	94

# **STAAD.Pro 2006 build 1002 Release Report**

## **Introduction**

The Software Release Report for STAAD.Pro 2006 build 1002 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2006 build 1001. This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

## **AD.2006-1002.1 Features Affecting the Pre-Processor (Modelling Mode)**

Several new features have been added and existing features have been modified in the pre-processor section of the program, also known as the Modelling Mode. These are explained in the following pages.

## **AD.2006-1002.1.1 Automatic calculation of the Response Spectrum as per IBC**

### **Purpose**

STAAD.Pro previously allowed response spectra to be defined by the user, or calculated as per the Indian IS 1893 or Eurocode EC8. The graphical user interface (GUI) has been enhanced so that it can now automatically generate a response spectrum as per the guidelines of the IBC/ASCE code (\*). This new feature saves users the time and effort involved in establishing the 1 second and short period accelerations indicated in the IBC/ASCE maps and then converting them into spectral data based on the equations provided in the code.

The user enters the latitude and longitude (or select a US zip code), STAAD.Pro then establishes the site coefficients, S1 and SS. The user then selects the appropriate site class (A-F) and with this information, STAAD.Pro calculates the MCE spectral acceleration and design spectral acceleration as per sections 11.4.3 and 11.4.4 of the ASCE7-05 code. Finally the design response spectrum will be generated based on section 11.4.5 of the ASCE7-05 code.

Note that there is no change to the input command structure.

### **Description**

Go to the **General | Load** page and in the **Load** dialog box, open the **Load Case Details** and select the appropriate primary load case with the masses (defined as load items) defined or create a new primary load case and add the masses, if required. Click on the **[Add]** button which will bring up the **Add New: Load Items** dialog box. Select the **Response Spectrum** option from the list on the left.

With the code option set to “**Custom**”, a **[Generate IBC Spectrum]** button will now be available as shown thus:-

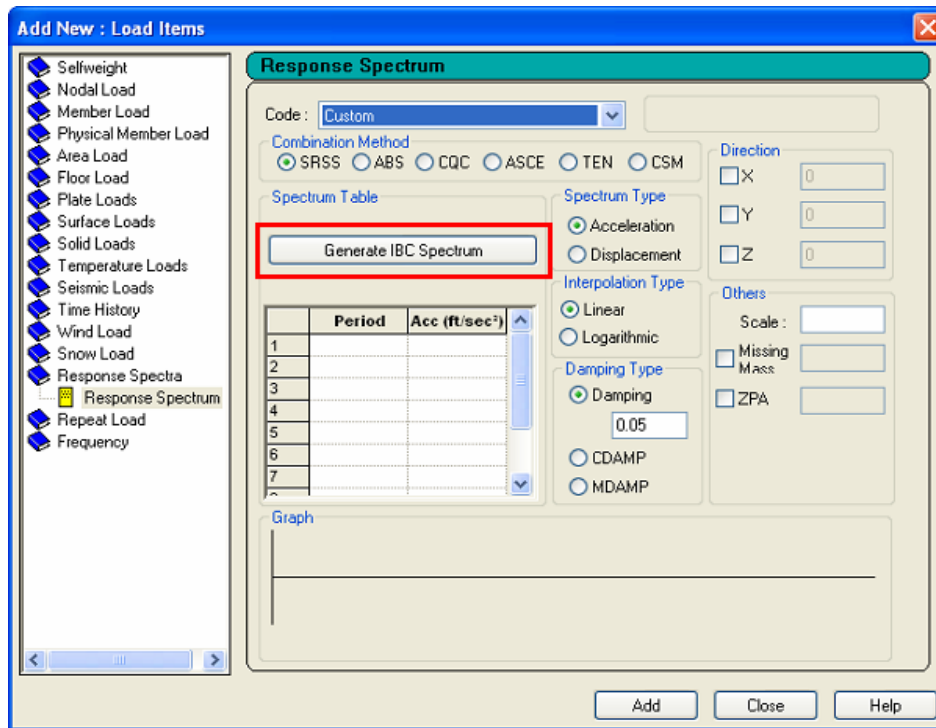


Figure 1

Click on the [**Generate IBC Spectrum**] button to open the **Spectrum Parameters** dialog box thus:-

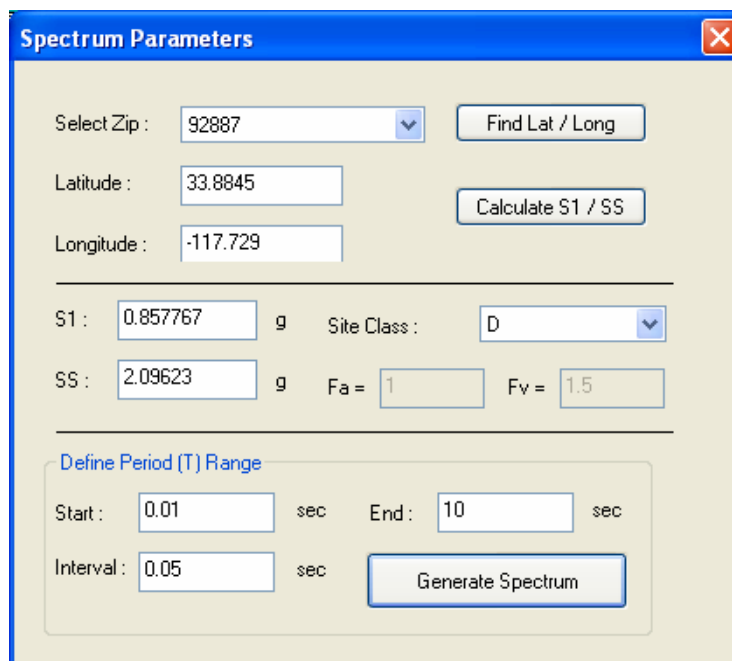


Figure 2

The options available in this dialog box are:-

**Select Zip**

Select the appropriate zip code from the available list of zip codes either by typing or selecting it from the drop-list. This will update the Latitude and Longitude settings.

**Latitude**

This should be entered if the Zip code is not known.

**Longitude**

This should be entered if the Zip code is not known.

**[Find Lat /Long]**

This updates the Latitude and Longitude values with those corresponding to the current Zip setting.

**[Calculate S1/SS]**

If the Latitude and / or the Longitude values are entered by hand rather than set by setting the Zip code, then the site coefficients S1 and SS are set according to this location by clicking on this button.

**S1**

This is the mapped MCE spectral response acceleration at a period of 1 second, determined in accordance with section 11.4.1 of ASCE-7-05. It is calculated by setting the Zip code or by clicking the [Calculate S1/SS] button. Alternatively, this can be set by the user.

**SS**

This is the mapped MCE spectral response acceleration at short period, determined in accordance with section 11.4.1 of ASCE-7-05. It is calculated by setting the Zip code or by clicking the [Calculate S1/SS] button. Alternatively, this can be set by the user.

**Site Class**

This is the classification (A to F) assigned to a site based on the soil type as defined in chapter 20 of ASCE-7-05. This then sets the values of Fa and Fv (see below). If the classification is set to F, then the parameters, Fa and Fv, should be set by the user.

**Fa**

Short period site coefficient determined in accordance with table 11.4 -1 of ASCE-7-05. This is determined by the Site Class, but must be set by the user if the Site Class is set to F.

**Fv**

Long period (1 sec) site coefficient determined in accordance with table 11.4 -2 of ASCE-7-05. This is determined by the Site Class, but must be set by the user if the Site Class is set to F.

**Define Period (T) Range:**

**Start**

This defines the starting value of the time period that is used for generation of the response spectrum.

**End**

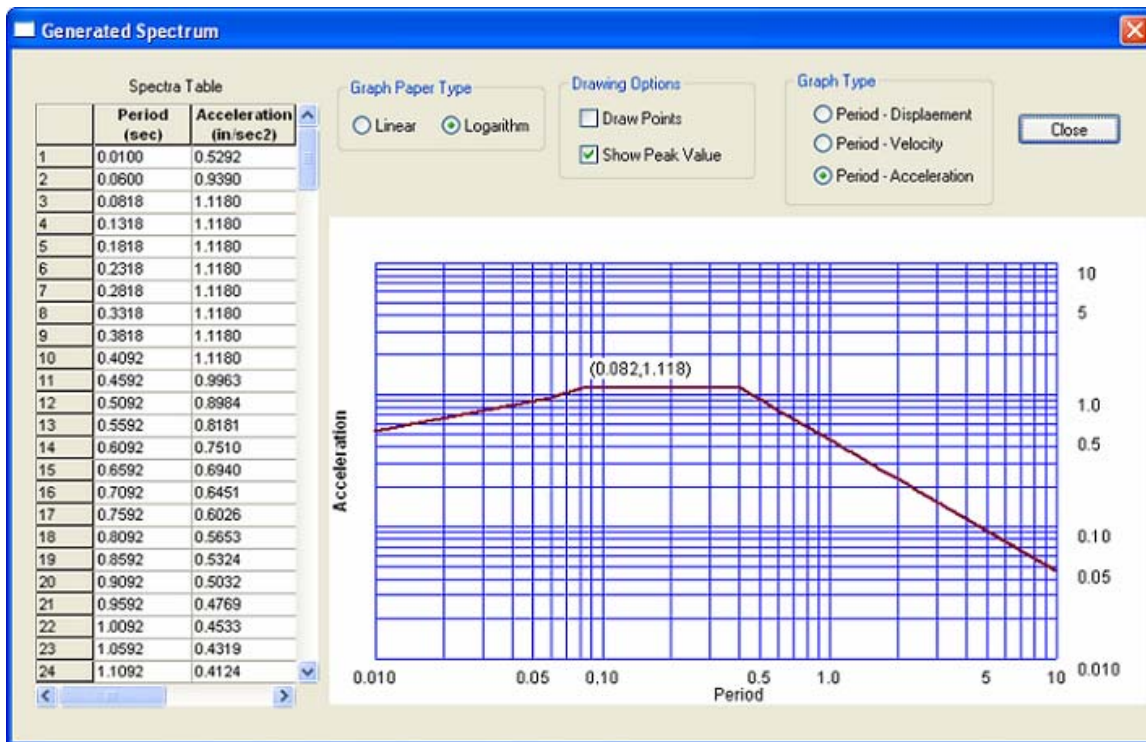
This defines the end value of the time period that is used for generation of the response spectrum.

**Interval**

This defines the time interval at which the spectral data is calculated.

**[Generate Spectrum]**

This generates the spectrum based on the inputted data and as per section 11.4.5 of the ASCE-7-05 code and displays the data in the following window:-



**Figure 3**

To complete the generation of the IBC response spectrum pairs, click on the [Close] button. The completion of the Response Spectrum command continues as for all other Custom defined Response Spectrum commands. Note that the details of the spectrum

Period, Acceleration pairs are displayed in the Response Spectrum window, but cannot be modified at this time.

For more details on the Response Spectrum loading command, refer to section 5.32.10.1 of the Technical Reference Manual.


(\*) IBC document....

## AD.2006-1002.1.2 Enhancement to Physical Member Query Window

### Purpose

With the enhancement of the physical Member in the Pre-Processing Mode, the Query dialog box obtained when a physical member is double clicked.

### Description

Select a Physical Member by using the Physical Member cursor,  and double-click on that to view the **Physical Member Query** dialog box.

### Pre-Processing Information:

Prior to any analysis being performed, the physical member information available is displayed in the following three sheets “Geometry”, “Property” and “Loading”.

All those common physical member attributes, assigned to that particular physical member (i.e. to all the analytical members comprising that physical member) are displayed in these information fields, as shown below:

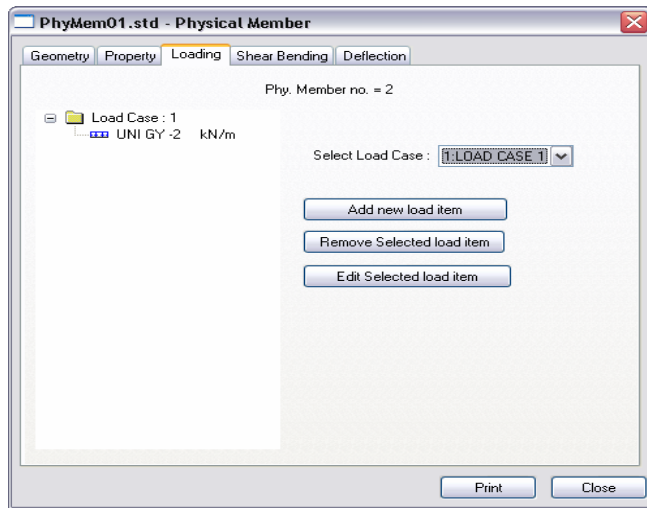
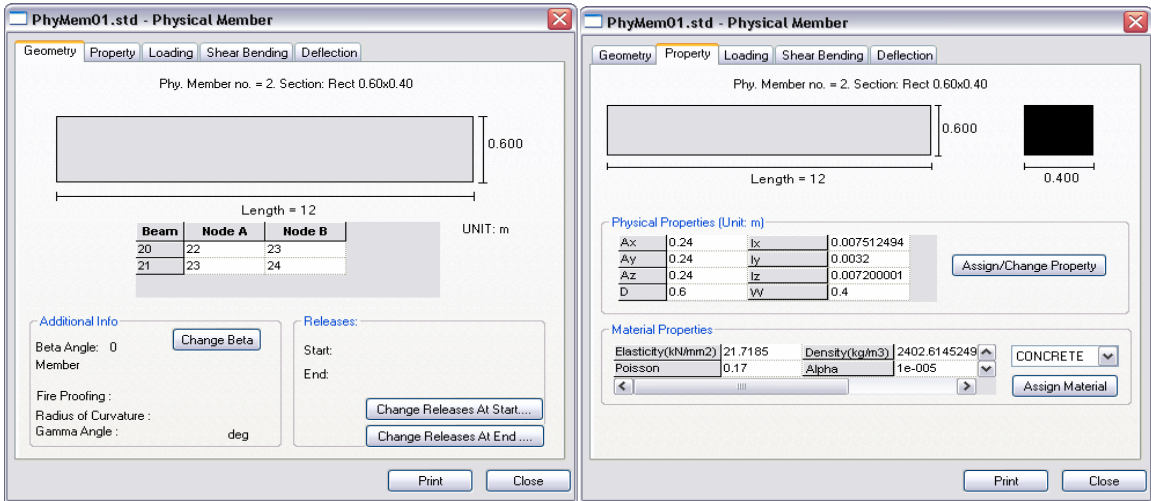


Figure 4

### Post Processing Information:

If an analysis has been performed and there are results available, the following post processing results are made available in two additional sheets “Shear Bending”, to view the Shear Force diagram, Bending Moment Diagram and “Deflection” to display deflected shape information.

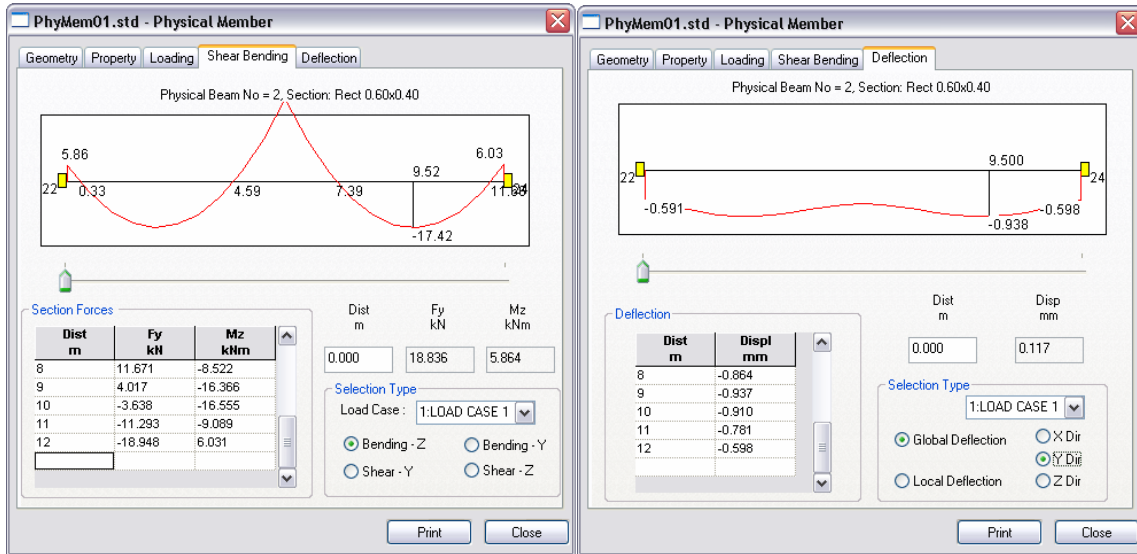


Figure 5

Note: The design of Physical Members is only performed in the Steel Design Mode and note the batch method in the current version of STAAD.Pro.

## AD.2006-1002.1.3 AASHTO (LRFD) Design Code

### Purpose

The STAAD.Pro analysis and design engine has been enhanced with the addition of a new design code, AASHTO (LRFD). The Graphical User Interface (GUI) has been updated to allow the design parameters to be entered and applied as required.

### Description

Go to the **Design | Steel** page and in the **Steel Design** dialog box, select the option **AASHTO (LRFD)** from the Current Code drop-list. Click on the button **[Select Parameters...]** to display the parameters available to be used with this design code. With all parameters displayed in the right Selected Parameters' panel, close the dialog box. Click on the **[Define Parameters...]** to set the values for the parameters which can be added or assigned to selected members. Click on the **[Commands...]** button to add or assign the commands to the file to instruct a check or selection of sections based on the parameter settings.

The AASHTO design parameters listed in previous versions of STAAD.Pro as 'AASHTO' in the Current Code drop-list of the **Steel Design** dialog box, have been re-named as 'AASHTO (ASD)

See also Section AD.2006-1002.2.1 for details

Note that access to design codes is controlled by your licence agreement and security settings. Not all users have access to all design codes. If this code is unavailable, see your STAAD.Pro supplier to obtain the updates needed to activate this or any other design code.

## AD.2006-1002.1.4 Node Precision

### Purpose

A number of models require a higher than usual precision on the node co-ordinates that are used by the analysis engine.

### Description

With STAAD.Pro running, but without having any structural model loaded, click on the menu item:-

File> Configure...

In the resulting dialog, click on the 'Input File Format' sheet, there is a new section Significant Digits which allows the required value to be set:-

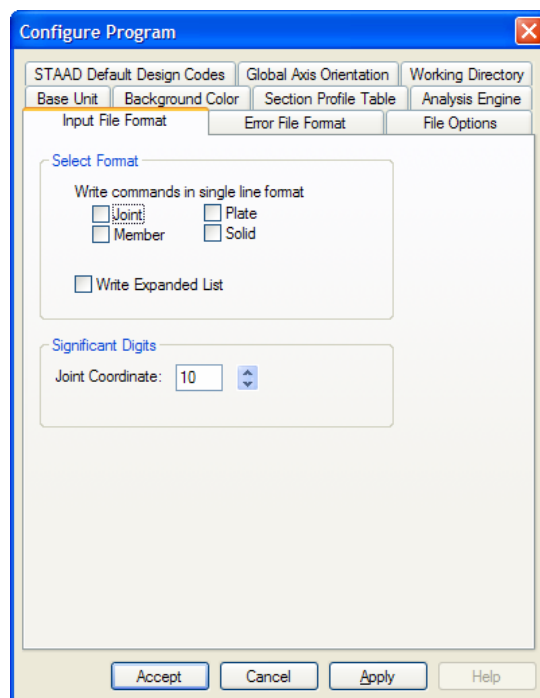


Figure 6

## AD.2006-1002.1.5 User Steel Table – Double Angles

### Purpose

To design a beam that has been defined as a Double Angle from a User table, then an 11th parameter must be provided in the section parameters which defines the RVV value (see International Design Codes Manual section 2B.4). The GUI has been enhanced to support this additional parameter.

### Description

Select the menu option:-

Tools>Create User Table...

Start a new table which is of type 'DOUBLE ANGLE' and click on the [Add New Property] button which will display the following dialog which allows the value of Rvv to be included in the section property definition.

Parameter	Value	Unit
Section Name	Angle	
Inertia about local z (Iz)	360.009	cm <sup>4</sup>
Inertia about local y (Iy)	792	cm <sup>4</sup>
Torsional Constant (Ix)	12.6667	cm <sup>4</sup>
C.G. along local Y (Cy)	2.86842	cm
Shear Area in Y (Ay)	20	cm <sup>2</sup>
Shear Area in Z (Az)	20	cm <sup>2</sup>
Single Angle (rvv)	3.3	cm
D	10	cm
TF	1	cm
WF	10	cm
SP	1	cm

Figure 7

The following is a replacement to the current paragraph for Double Angles in the Technical Reference Manual Section, Section 5.19. The remainder of the section remains unaltered.

### 5.19 User Steel Table Specification:-

## Double Angle

1) D, 2) WF, 3) TF, 4) SP, 5) IZ, 6) IY, 7) IX, 8) CY, 9) AY,  
10) AZ, 11) RVV

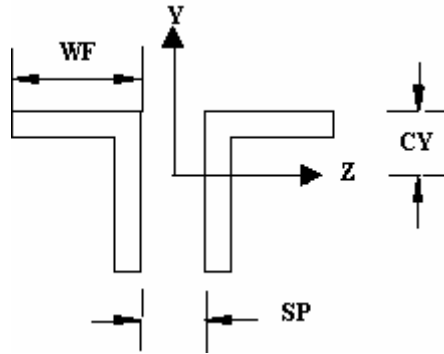


Figure 5.7

Note:-

The parameter RVV is defined as the radius of gyration about the v-v axis (minor principal axis) for a single angle.

## **AD.2006-1002.2 Features Affecting the Analysis and Design Engine**

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

## **AD.2006-1002.2.1 AASHTO Design Code**

### **Purpose**

The STAAD.Pro analysis and design engine has been enhanced with the addition of a new design code, AASHTO (LRFD).

### **Description**

The current AASHTO design code in STAAD.Pro performs design as per the ASD. The following replaces section 2.13 of the Technical Reference Manual.

See also Section AD.2006-1002.2.1 for details of the change to the Graphical User Interface.

Note that access to design codes is controlled by your licence agreement and security settings. Not all users have access to all design codes. If this code is unavailable, see your STAAD.Pro supplier to obtain the updates needed to activate this or any other design code.

## ***2.13 Steel Design per AASHTO Specifications***

STAAD.Pro has implemented design to AASHTO Standard Specifications for Highway Bridges utilizing the ASD and LRFD approaches. These are described in the following two sections.

### **2.13.1 AASHTO (ASD)**

The design of structural steel members in accordance with the AASHTO Standard Specifications for Highway Bridges, 17<sup>th</sup> edition has been implemented.

#### **General Comments**

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 9<sup>th</sup> 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 3<sup>rd</sup> edition AISC LRFD manual) and Composite beams (I shapes with concrete slab on top) are not supported.

#### **Allowable Stresses per AASHTO Code**

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,

$$F_t = 0.55F_y$$

Allowable compressive stress on the gross section of axially loaded compression members is calculated based on the following formula:

$$F_a = \frac{F_y}{F.S.} \frac{(1 - (Kl/r)^2) F_y}{4\pi^2 E} \quad \text{when } (Kl/r) < C_c$$

$$F_a = \frac{\pi^2 E}{F.S. (Kl/r)^2} \quad \text{when } (Kl/r) > C_c$$

$$\text{with } C_c = (2\pi^2 E / F_y)^{1/2} \quad \text{and } F.S. = 2.12$$

It should be noted that AASHTO does not have a provision for increase in allowable stresses for a secondary member and when  $l/r$  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:-

$$F_b = 0.55F_y$$

For similar members with unsupported or partially supported flange lengths, the allowable bending compressive stress is given by

$$F_b = 0.55F_y \left(1 - \frac{(l/r)^2 F_y}{4\pi^2 E}\right) \quad \text{when } (kl/r) < C_c$$

$$\text{with } (r^2) = b^2 / 12$$

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.

### **Shear Stress**

Allowable shear stress on the gross section is given by:-

$$F_v = 0.33F_y$$

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:

$$\frac{f_a}{F_a} + \frac{C_{mx} f_{bx}}{(1 - f_a / F_{ex}) F_{bx}} + \frac{C_{my} f_{by}}{(1 - f_a / F_{ey}) F_{by}} < 1.0$$

**at intermediate points, and**

$$\frac{f_a}{.472 F_y} + \frac{f_{bx}}{F_{bx}} + \frac{f_{by}}{F_{by}} < 1.0$$

**at support points.**

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.

$$\frac{f_a}{F_a} + \frac{f_{bx}}{F_{bx}} + \frac{f_{by}}{F_{by}} < 1.0$$

at intermediate points, and

$$\frac{f_a}{0.472 F_y} + \frac{f_{bx}}{F_{bx}} + \frac{f_{by}}{F_{by}} < 1.0$$

at ends of member.

## 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.

<b>Table 2.3 - AASHTO (ASD) Design Parameters</b>		
<b>Parameter Name</b>	<b>Default Value</b>	<b>Description</b>
<u>FYLD</u>	36 KSI	Yield strength of steel in current units.
<u>FU</u>	Depends on FYLD	Ultimate tensile strength of steel in current units.
<u>RATIO</u>	1.0	Permissible ratio of the actual to allowable stresses.
<u>KY</u>	1.0	K value in local y-axis. Usually, this is minor axis.
<u>KZ</u>	1.0	K value in local z-axis. Usually, this is major axis.
<u>UNF</u>	1.0	Unsupported length factor of the compression flange for calculating the allowable bending compressive strength.
<u>LY</u>	Member Length	Length to calculate slenderness ratio for buckling about local Y axis.
<u>LZ</u>	Member Length	Same as above except in local z-axis.
<u>UNL</u>	Member Length	Unsupported length of compression flange for calculating allowable bending compressive stress.
<u>NSF</u>	1.0	Ratio of 'Net cross section area' / 'Gross section area' for tension member design.
<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>CMY</u> <u>CMZ</u>	0.85 for sidesway and calculated for no sidesway	Cm value in local y & z axes
<u>MAIN</u>	0.0	0.0 =check for slenderness 1.0 =suppress slenderness check
<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 <sup>th</sup> 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>DMAX</u>	1000.0	Maximum allowed section depth (in current length units) for a section to be selected with the SELECT command.
<u>DMIN</u>	0.0	Minimum allowed section depth (in current length units) for a section to be selected with the SELECT command.

<u>PROFILE</u>	None	Used in member section. See section 5.47.1 of Technical Reference Manual for more 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>STIFF</u>	Greater of member length or depth of beam.	Spacing of stiffeners for plate girder design in current length units.
<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>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".
<u>WSTR</u>	0.4 x FYLD	Allowable welding stress
<u>WMAX</u>	See Section 2.12	Maximum welding thickness
<u>WMIN</u>	See Section 2.12	Minimum welding thickness
<u>WELD</u>	1 for open sections 2 for closed sections	Weld type as described in section 2.11. 6.0 = Welding on one side except for wide-flange or tee sections where the web is always assumed welded on both sides. 7.0 Welding on both sides. Closed sections such as pipes and tubes will only be considered as welded on one side.

## AASHTO (ASD) Verification Problem

The following compares the solution of a design performed using STAAD.Pro against a hand calculation.

**TYPE:** AASHTO ASD Steel Design.

**REFERENCE:** Attached 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.

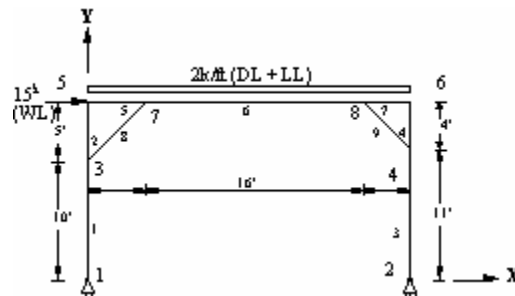


Figure 2.5

Members 1, 2 = W12X26, Members 3, 4 = W14X43

Members 5, 6, 7 = W16X36, Memb8= L40404,

Member 9 = L50506

### SOLUTION COMPARISON:

Governing ratios for the members									
Member No.	1	2	3	4	5	6	7	8	9
Hand Calculation	1.217	1.092	1.203	1.126	.689	0.729	0.808	1.09	.927
STAAD.Pro	1.218	1.903	1.204	1.126	.690	0.732	0.809	1.09	.928

## VERIFICATION PROBLEM HAND CALCULATION

Manual / Code refers to AASHTO Manual.  
Steel Design -

**Member 1**, Size W 12X26, L = 10 ft., a = 7.65 in<sup>2</sup>, Sz = 33.39 in<sup>3</sup>

From observation Load case 1 will govern,

F<sub>x</sub> = 25.0 kip (compression), M<sub>z</sub> = 56.5 k-ft

### Allowable Stress Calculation:

From Table 10.32.1A,

### Bending Minor Axis:

Allowable minor axis bending stress:

F<sub>TY</sub> = F<sub>TZ</sub> = 0.55 x F<sub>Y</sub> = 19.8 ksi

### Bending Major Axis:

$$F_{ax} = \frac{50 \times 10^6 C_b \left( \frac{I_{yc}}{l} \right) \sqrt{0.772 \frac{J}{I_{yc}} + 9.87 \left( \frac{d}{l} \right)^2}}{S_{xc}} \leq 0.55 F_y$$

Where, C<sub>b</sub> = 1.75 + 1.05(M<sub>1</sub>/M<sub>2</sub>) + 0.3x(M<sub>1</sub>/M<sub>2</sub>)<sup>2</sup>

Here M<sub>1</sub> = 0, so C<sub>b</sub> = 1.75

S<sub>zc</sub> = Section modulus w.r.t. compression flange = 204 / (0.5 x 12.22) = 33.38789 in<sup>3</sup>

I<sub>YC</sub> = tb<sup>3</sup> / 12 = 0.38 x 6.49<sup>3</sup> / 12 = 8.6564 in<sup>4</sup>

J = (2 x 6.49 x 0.38<sup>3</sup> + (12.22 - 2 x 0.38) x 0.23<sup>3</sup>) / 3 = 0.28389 in<sup>4</sup>

$$F_{\alpha} = \frac{50 \times 10^6 \times 1.75 \left( \frac{8.6564}{120} \right)}{33.38789} \sqrt{0.772 \frac{0.28389}{8.6564} + 9.87 \left( \frac{11.46}{120} \right)^2} = 64375.03 \text{ psi}$$

From this, calculated FCZ = 64.37. As this is larger than 0.55x $F_Y$

$$FCZ = 0.55 \times F_Y = 19.8 \text{ ksi}$$

### **Axial Compression:**

$$\text{Critical } (kL/r)_y = 1.0 \times 120 / 1.5038 = 79.7978$$

$$C_c = \sqrt{\frac{2\pi^2 E}{F_y}} = \sqrt{\frac{2\pi^2 * 29000}{36}} = 126.099$$

As  $(kL/r)_y < C_c$ , the allowable axial stress in compression

$$F_a = \frac{F_y}{F.S.} \left[ 1 - \frac{(kL/r)^2 F_y}{4\pi^2 E} \right] = \frac{36}{2.12} \left[ 1 - \frac{(79.79)^2 36}{4\pi^2 29000} \right] = 13.58 \text{ ksi}$$


---

### **Shear:**

Allowable shear stress as per gross section,  $F_v = 0.33 \times F_Y = 11.8 \text{ ksi}$

### **Actual Stress Calculation:**

Actual stress ( $f_a$ ) = 25/7.65 = 3.26 ksi.

Actual bending stress = 56.5 x 12/33.4 = 1.691616 x 12 = 20.29932 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.

$$F_{ez} = \frac{\pi^2 E}{F.S. (KL/r)^2} = \frac{\pi^2 * 29000}{2.12 \left(\frac{120}{5.17}\right)^2} = 250.5999$$

From table 10-36A,  $C_{mz} = 0.85$

Equation 10-42

$$\frac{f_a}{F_a} + \frac{C_{mx} f_{bx}}{\left(1 - \frac{f_a}{F'_{ez}}\right) F_{bx}} + \frac{C_{my} f_{by}}{\left(1 - \frac{f_a}{F'_{ey}}\right) F_{by}} = \frac{3.26}{13.58} + \frac{0.85 * 20.299}{\left(1 - \frac{3.26}{250.599}\right) 19.8} + 0 = 1.122$$

For the end section, Equation 10-43

$$\frac{f_a}{0.472 F_y} + \frac{f_{bx}}{F_{bx}} + \frac{f_{by}}{F_{by}} = \frac{3.26}{0.472 * 36} + \frac{20.299}{19.8} + 0 = 1.217$$

The value calculated by STAAD is 1.218

**Member 2**, Size W 12X26, L = 5 ft., a = 7.65 in<sup>2</sup>, Sz = 33.4 in<sup>3</sup>

From observation Load case 1 will govern, Forces at the midspan are

Fx = 8.71 kip (compression), Mz = 56.5 k-ft

### **Allowable Stress Calculation:**

From Table 10.32.1A,

### **Bending Minor Axis:**

Allowable minor axis bending stress:

$$F_{TY} = F_{TZ} = 0.55 \times F_Y = 19.8 \text{ ksi}$$

### **Bending Major Axis:**

$$F_{\alpha} = \frac{50 \times 10^6 C_b}{S_{xc}} \left( \frac{I_{yc}}{l} \right) \sqrt{0.772 \frac{J}{I_{yc}} + 9.87 \left( \frac{d}{l} \right)^2} \leq 0.55 F_y$$

Where,  $C_b = 1.75 + 1.05(M1/M2) + 0.3x(M1/M2)^2$

Here  $M1 = 39.44$ ,  $M2 = 677.96$  so  $C_b = 1.69$

$S_{xc}$  = Section modulus w.r.t. compression flange =  $204 / (0.5 \times 12.22) = 33.38789 \text{ in}^3$

$I_{yc} = tb^3/12 = 0.38 \times 6.49^3/12 = 8.6564 \text{ in}^4$

$J = (2 \times 6.49 \times 0.38^3 + (12.22 - 2 \times 0.38) \times 0.23^3) / 3 = 0.28389 \text{ in}^4$

$$F_{\alpha} = \frac{50 \times 10^6 \times 1.69}{33.38789} \left( \frac{8.6564}{60} \right) \sqrt{0.772 \frac{0.28389}{8.6564} + 9.87 \left( \frac{11.46}{60} \right)^2} = 227337.5 \text{ psi}$$

Since FCZ is larger than  $0.55x F_y$ ,  $FCZ = 0.55x F_y = 19.8 \text{ ksi}$

### **Axial Compression:**

Critical  $(KL/r)_y = 1.0 \times 60 / 1.503 = 39.92$

$$C_c = \sqrt{\frac{2\pi^2 E}{F_y}} = \sqrt{\frac{2\pi^2 * 29000}{36}} = 126.099$$

As  $(KL/r)_y < C_c$ , the allowable axial stress in compression

$$F_a = \frac{F_y}{F.S.} \left[ 1 - \frac{(KL/r)^2 F_y}{4\pi^2 E} \right] = \frac{36}{2.12} \left[ 1 - \frac{(39.92)^2 36}{4\pi^2 29000} \right] = 16.13$$

### **Shear:**

Allowable shear stress as per gross section,  $F_v = 0.33x F_y = 11.8 \text{ ksi}$

### **Actual Stress Calculation:**

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.

Actual bending stress =  $56.5 \times 12 / 33.4 = 1.691 \times 12 = 20.299$  ksi

$(KL/r)_z = 1 \times 60 / 5.16 = 11.618$  11.628

$$F_{\text{cr}} = \frac{\pi^2 E}{F.S. (KL/r)^2} = \frac{\pi^2 * 29000}{2.12 (11.618)^2} = 998.524$$

From table 10-36A,  $C_{mz} = 0.85$

So the design ratio is

Equation 10-42

$$\frac{f_a}{F_a} + \frac{C_{mx} f_{bx}}{\left(1 - \frac{f_a}{F'_{\text{cr}}}\right) F_{bx}} + \frac{C_{my} f_{by}}{\left(1 - \frac{f_a}{F'_{\text{cr}}}\right) F_{by}} = \frac{1.138}{16.13} + \frac{0.85 * 20.299}{\left(1 - \frac{1.138}{1000.22}\right) 19.8} + 0 = 0.942$$

for the end section, Equation 10-43

$$\frac{f_a}{0.472 F_y} + \frac{f_{bx}}{F_{bx}} + \frac{f_{by}}{F_{by}} = \frac{1.138}{0.472 * 36} + \frac{20.299}{19.8} + 0 = 1.092$$

The value calculated by STAAD is 1.093

**Member 3**, Size W 14X43, L = 11 ft., a = 12.6 in<sup>2</sup>, Sz = 62.7 in<sup>3</sup>

From observation Load case 3 will govern, Forces at the end are

Fx = 25.5 kip (compression), Mz = 112.17 k-ft

### Allowable Stress Calculation:

From Table 10.32.1A,

#### Bending:

#### Minor Axis:

Allowable minor axis bending stress:

$$F_{TY} = F_{TZ} = 0.55 \times F_Y = 19.8 \text{ ksi}$$

### Major Axis:

$$F_{\alpha} = \frac{50 \times 10^6 C_b}{S_{xc}} \left( \frac{I_{yc}}{l} \right) \sqrt{0.772 \frac{J}{I_{yc}} + 9.87 \left( \frac{d}{l} \right)^2} \leq 0.55 F_y$$

Where,  $C_b = 1.75 + 1.05(M1/M2) + 0.3x(M1/M2)^2$

Here  $M1 = 0$ , so  $C_b = 1.75$

$S_{zc}$  = Section modulus w.r.t. compression flange =  $428 / (0.5 \times 13.66) = 62.7 \text{ in}^3$

$I_{yc} = tb^3/12 = 0.53 \times 8.0^3/12 = 22.61 \text{ in}^4$

$J = (2 \times 8.0 \times 0.53^3 + (13.66 - 2 \times 0.53) \times 0.305^3) / 3 = 0.913 \text{ in}^4$

$$F_{\alpha} = \frac{50 \times 10^6 \times 1.75}{62.7} \left( \frac{22.61}{132} \right) \sqrt{0.772 \frac{0.917}{22.61} + 9.87 \left( \frac{12.6}{132} \right)^2} = 83187.61 \text{ psi}$$

Since FCZ is larger than  $0.55x F_y$ ,  $FCZ = 0.55x F_y = 19.8 \text{ ksi}$

### Axial Compression:

Critical  $(KL/r)_y = 1.0 \times 11 \times 12 / 1.894 = 69.69$

$$C_c = \sqrt{\frac{2\pi^2 E}{F_y}} = \sqrt{\frac{2\pi^2 * 29000}{36}} = 126.099$$

As  $(KL/r)_y < C_c$ , the allowable axial stress in compression

$$F_a = \frac{F_y}{F.S.} \left[ 1 - \frac{(KL/r)^2 F_y}{4\pi^2 E} \right] = \frac{36}{2.12} \left[ 1 - \frac{(69.69)^2 36}{4\pi^2 29000} \right] = 14.387$$

### Shear:

Allowable shear stress as per gross section,  $F_v = 0.33x F_y = 11.8 \text{ ksi}$

### Actual Stress Calculation:

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.

Actual bending stress =  $112.17 \times 12 / 62.7 = 1.789 \times 12 = 21.467$  ksi

$(KL/r)_z = 1 \times 132 / 5.828 = 22.649$

$$F_{ez} = \frac{\pi^2 E}{F.S. (KL/r)^2} = \frac{\pi^2 * 29000}{2.12 (22.648)^2} = 263.18$$

From table 10-36A,  $C_{mz} = 0.85$

So the design ratio is

Equation 10-42

$$\frac{f_a}{F_a} + \frac{C_{mx} f_{bx}}{\left(1 - \frac{f_a}{F'_{ez}}\right) F_{bx}} + \frac{C_{my} f_{by}}{\left(1 - \frac{f_a}{F'_{ey}}\right) F_{by}} = \frac{2.024}{14.387} + \frac{0.85 * 21.467}{\left(1 - \frac{2.024}{263.209}\right) 19.8} + 0 = 1.069$$

for the end section, Equation 10-43

$$\frac{f_a}{0.472 F_y} + \frac{f_{bx}}{F_{bx}} + \frac{f_{by}}{F_{by}} = \frac{2.024}{0.472 * 36} + \frac{21.467}{19.8} + 0 = 1.203$$

The value calculated by STAAD is 1.204

**Member 4**, Size W 14X43, L = 4 ft., a = 12.6 in<sup>2</sup>, Sz = 62.7 in<sup>3</sup>

From observation Load case 3 will govern, Forces at the end are

Fx = 8.75 kip (tension), Mz = 112.17 k-ft

#### **Allowable Stress Calculation:**

From Table 10.32.1A,

#### **Bending:**

#### **Minor Axis:**

Allowable minor axis bending stress:

$F_{TY} = F_{TZ} = 0.55 \times F_Y = 19.8$  ksi

#### **Major Axis:**

$$F_{cz} = \frac{50 \times 10^6 C_b \left( \frac{I_{yc}}{l} \right)}{S_{zc}} \sqrt{0.772 \frac{J}{I_{yc}} + 9.87 \left( \frac{d}{l} \right)^2} \leq 0.55 F_y$$

Where,  $C_b = 1.75 + 1.05(M1/M2) + 0.3x(M1/M2)^2$

Here  $M1 = -191.36$  Kip-in ,  $M2 = -1346.08$  Kip-in so  $C_b = 1.606$

$S_{zc}$  = Section modulus w.r.t. compression flange =  $428 / (0.5 \times 13.66) = 62.7 \text{ in}^3$

$I_{yc} = tb^3 / 12 = 0.53 \times 8.0^3 / 12 = 22.61 \text{ in}^4$

$J = (2 \times 8.0 \times 0.53^3 + (13.66 - 2 \times 0.530) \times 0.305^3) / 3 = 0.911 \text{ in}^4$

$$F_{cz} = \frac{50 \times 10^6 \times 1.606 \left( \frac{22.61}{48} \right)}{62.7} \sqrt{0.772 \frac{0.911}{22.61} + 9.87 \left( \frac{12.6}{48} \right)^2} = 508752.2 \text{ psi}$$

Since  $FCZ$  is larger than  $0.55 \times F_Y$ ,  $FCZ = 0.55 \times F_Y = 19.8 \text{ ksi}$

### **Axial Compression:**

$F_a = 0.55 \times F_Y = 19.8 \text{ ksi}$

### **Shear:**

Allowable shear stress as per gross section,  $F_v = 0.33 \times F_Y = 11.8 \text{ ksi}$

### **Actual Stress Calculation:**

Axial stress ( $f_a$ ) =  $8.75 / 12.6 = 0.6944 \text{ ksi}$ .

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.

Actual bending stress =  $112.17 \times 12 / 62.7 = 1.789 \times 12 = 21.467 \text{ ksi}$

$$\frac{f_a}{F_a} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{0.6944}{19.8} + \frac{21.467}{19.8} + 0 = 1.119$$

$$\frac{f_a}{0.472 F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{0.6944}{0.472 \times 36} + \frac{21.467}{19.8} + 0 = 1.125$$

The value calculated by STAAD is 1.126

**Member 5**, Size W 16X36, L = 5 ft., a = 10.6 in<sup>2</sup>, Sz = 56.5 in<sup>3</sup>

From observation Load case 3 will govern, Forces at the end are

F<sub>x</sub> = 14.02 kip (compression), M<sub>z</sub> = 57.04 k-ft

**Allowable Stress Calculation:**

From Table 10.32.1A,

**Bending:**

**Minor Axis:**

Allowable minor axis bending stress:

F<sub>TY</sub> = F<sub>TZ</sub> = 0.55 x F<sub>Y</sub> = 19.8 ksi

**Major Axis:**

$$F_{cz} = \frac{50 \times 10^6 C_b}{S_{xc}} \left( \frac{I_{yc}}{l} \right) \sqrt{0.772 \frac{J}{I_{yc}} + 9.87 \left( \frac{d}{l} \right)^2} \leq 0.55 F_y$$

Where, C<sub>b</sub> = 1.75 + 1.05(M<sub>1</sub>/M<sub>2</sub>) + 0.3x(M<sub>1</sub>/M<sub>2</sub>)<sup>2</sup>

Here M<sub>1</sub> = 40.14, M<sub>2</sub> = -684.4 so C<sub>b</sub> = 1.81

S<sub>zc</sub> = Section modulus w.r.t. compression flange = 448 / (0.5 x 15.86) = 56.5 in<sup>3</sup>

I<sub>YC</sub> = tb<sup>3</sup> / 12 = 0.43 x 6.99<sup>3</sup> / 12 = 12.238 in<sup>4</sup>

J = (2 x 6.99 x 0.43<sup>3</sup> + (15.86 - 2 x 0.43) x 0.29<sup>3</sup>) / 3 = 0.5 in<sup>4</sup>

$$F_{\alpha} = \frac{50 \times 10^6 \times 1.81 \left( \frac{12.238}{60} \right)}{56.5} \sqrt{0.772 \frac{0.5}{12.238} + 9.87 \left( \frac{15}{60} \right)^2} = 263079 \text{ psi}$$

Since FCZ is larger than  $0.55 \times F_Y$ ,  $FCZ = 0.55 \times F_Y = 19.8 \text{ ksi}$

### **Axial Compression:**

$$\text{Critical } (KL/r)_y = 1.0 \times 5 \times 12 / 1.52 = 39.474$$

$$C_c = \sqrt{\frac{2\pi^2 E}{F_y}} = \sqrt{\frac{2\pi^2 * 29000}{36}} = 126.099$$

As  $(KL/r)_y < C_c$ , the allowable axial stress in compression

$$F_a = \frac{F_y}{F.S.} \left[ 1 - \frac{(KL/r)^2 F_y}{4\pi^2 E} \right] = \frac{36}{2.12} \left[ 1 - \frac{(39.474)^2 36}{4\pi^2 29000} \right] = 16.149$$

### **Shear:**

Allowable shear stress as per gross section,  $F_v = 0.33 \times F_Y = 11.8 \text{ ksi}$

### **Actual Stress Calculation:**

Axial stress ( $f_a$ ) =  $14.02 / 10.6 = 1.323 \text{ ksi}$ .

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.

Actual bending stress =  $57.04 \times 12 / 56.5 = 1.001 \times 12 = 12.115 \text{ ksi}$

$$(KL/r)_z = 1 \times 60 / 6.5 = 9.231$$

$$F_{\text{cr}} = \frac{\pi^2 E}{F.S. (KL/r)^2} = \frac{\pi^2 * 29000}{2.12(9.231)^2} = 1584.398$$

From table 10-36A,  $C_{mz} = 0.85$

So the design ratio is

Equation 10-42

$$\frac{f_a}{F_a} + \frac{C_{mx} f_{bx}}{\left(1 - \frac{f_a}{F'_{\text{cr}}}\right) F_{bx}} + \frac{C_{my} f_{by}}{\left(1 - \frac{f_a}{F'_{\text{cr}}}\right) F_{by}} = \frac{1.323}{16.149} + \frac{0.85 * 12.115}{\left(1 - \frac{1.323}{1584.398}\right) 19.8} + 0 = 0.602$$

for the end section, Equation 10-43

$$\frac{f_a}{0.472 F_y} + \frac{f_{bx}}{F_{bx}} + \frac{f_{by}}{F_{by}} = \frac{1.323}{0.472 * 36} + \frac{12.115}{19.8} + 0 = 0.689$$

The value calculated by STAAD is 0.69

**Member 6**, Size W 16X36, L = 16 ft., a = 10.6 in<sup>2</sup>, Sz = 56.5 in<sup>3</sup>

From observation Load case 3 will govern, Forces at the end are

Fx = 10.2 kip (compression), Mz = 62.96 k-ft

### Allowable Stress Calculation:

From Table 10.32.1A,

#### Bending:

#### Minor Axis:

Allowable minor axis bending stress:

$$F_{TY} = F_{TZ} = 0.55 \times F_y = 19.8 \text{ ksi}$$

#### Major Axis:

$$F_{\alpha} = \frac{50 \times 10^6 C_b}{S_{xc}} \left( \frac{I_{yc}}{l} \right) \sqrt{0.772 \frac{J}{I_{yc}} + 9.87 \left( \frac{d}{l} \right)^2} \leq 0.55 F_y$$

Where,  $C_b = 1.75 + 1.05(M1/M2) + 0.3x(M1/M2)^2$

Here  $M1 = 8.947$   $M2 = 183.05$ , so  $C_b = 1.69$

$S_{zc}$  = Section modulus w.r.t. compression flange =  $448 / (0.5 \times 15.86) = 56.5 \text{ in}^3$

$I_{yc} = tb^3 / 12 = 0.43 \times 6.99^3 / 12 = 12.238 \text{ in}^4$

$J = (2 \times 6.99 \times 0.43^3 + (15.86 - 2 \times 0.43) \times 0.29^3) / 3 = 0.5 \text{ in}^4$

$$F_{\alpha} = \frac{50 \times 10^6 \times 1.69}{56.5} \left( \frac{12.238}{192} \right) \sqrt{0.772 \frac{0.5}{12.238} + 9.87 \left( \frac{15}{192} \right)^2} = 28789.17 \text{ psi}$$

Since  $FCZ$  is larger than  $0.55 \times F_Y$ ,  $FCZ = 0.55 \times F_Y = 19.8 \text{ ksi}$

### **Axial Compression:**

Critical  $(KL/r)_y = 1.0 \times 192 / 1.52 = 126.29$

$$C_c = \sqrt{\frac{2\pi^2 E}{F_y}} = \sqrt{\frac{2\pi^2 * 29000}{36}} = 126.099$$

As  $(KL/r)_y > C_c$ , the allowable axial stress in compression

$$F_a = \frac{\pi^2 E}{F.S. (KL/r)^2} = \frac{\pi^2 * 29000}{2.12 * (126.29)^2} = 8.46 \text{ ksi}$$

### **Shear:**

Allowable shear stress as per gross section,  $F_v = 0.33 \times F_Y = 11.8 \text{ ksi}$

### **Actual Stress Calculation:**

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.

Actual bending stress =  $62.96 \times 12 / 56.5 = 1.26106 \times 12 = 13.372$  ksi

$$(KL/r)_z = 1 \times 192 / 6.5 = 29.538$$

$$F_{\text{ex}} = \frac{\pi^2 E}{F.S. (KL/r)^2} = \frac{\pi^2 * 29000}{2.12 (29.538)^2} = 154.58$$

From table 10-36A,  $C_{mz} = 0.85$

$$\frac{f_a}{F_a} + \frac{C_{mx} f_{bx}}{\left(1 - \frac{f_a}{F'_{ex}}\right) F_{bx}} + \frac{C_{my} f_{by}}{\left(1 - \frac{f_a}{F'_{ey}}\right) F_{by}} = \frac{0.962}{8.46} + \frac{0.85 * 13.372}{\left(1 - \frac{0.926}{154.58}\right) 19.8} + 0 = 0.691$$

$$\frac{f_a}{0.472 F_y} + \frac{f_{bx}}{F_{bx}} + \frac{f_{by}}{F_{by}} = \frac{0.962}{0.472 * 36} + \frac{13.372}{19.8} + 0 = 0.732$$

The value calculated by STAAD is 0.732

**Member 7**, Size W 16X36, L = 4 ft., a = 10.6 in<sup>2</sup>, Sz = 56.5 in<sup>3</sup>

From observation Load case 3 will govern, Forces at the midspan are

Fx = 24.05 kip (tension), Mz = 62.96 k-ft

### Allowable Stress Calculation:

From Table 10.32.1A,

#### Bending:

#### Minor Axis:

Allowable minor axis bending stress:

$$F_{TY} = F_{TZ} = 0.55 \times F_Y = 19.8 \text{ ksi}$$

#### Major Axis:

$$F_{CZ} = 0.55 \times F_Y = 19.8 \text{ ksi}$$

#### Axial Tension:

$$F_a = 0.55 \times F_Y = 19.8 \text{ ksi}$$

**Shear:**

Allowable shear stress as per gross section,  $F_v = 0.33 \times F_Y = 11.8 \text{ ksi}$

**Actual Stress Calculation:**

Actual stress ( $f_a$ ) =  $24.05 / 10.6 = 2.268 \text{ ksi}$ , hence safe.

From Table 10.32.1A, Allowable stress in bending (compression)

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.

$$\text{Actual bending stress} = 62.96 \times 12 / 56.5 = 1.11433 \times 12 = 13.372 \text{ ksi}$$

So the design ratio is

$$\frac{f_a}{F_a} + \frac{f_{bx}}{F_{bx}} + \frac{f_{by}}{F_{by}} = \frac{2.268}{19.8} + \frac{13.372}{19.8} + 0 = 0.789$$

$$\frac{f_a}{0.472 F_y} + \frac{f_{bx}}{F_{bx}} + \frac{f_{by}}{F_{by}} = \frac{2.268}{0.472 \times 36} + \frac{13.372}{19.8} + 0 = 0.809$$

The value calculated by STAAD is 0.809

**Member 8**, Size L40404, L = 7.07 ft., a = 1.938 in<sup>2</sup>

From observation Load case 1 will govern, Forces

$$F_x = 23.04 \text{ kip (compression)}$$

**Allowable Stress Calculation:**

**Axial Compression:**

$$\text{Critical } (KL/r)_y = 1.0 \times 7.07 \times 12 / 0.795 = 106.73$$

$$C_c = \sqrt{\frac{2\pi^2 E}{F_y}} = \sqrt{\frac{2\pi^2 \times 29000}{36}} = 126.099$$

As  $(KL/r)_y < C_c$ , the allowable axial stress in compression

$$F_a = \frac{F_y}{F.S.} \left[ 1 - \frac{(KL/r)^2 F_y}{4\pi^2 E} \right] = \frac{36}{2.12} \left[ 1 - \frac{(106.73)^2 36}{4\pi^2 29000} \right] = 10.89 \text{ ksi}$$

**Actual Stress Calculation:**

Actual stress ( $f_a$ ) = 23.04 / 1.938 = 11.88 ksi.

$$\text{Ratio } \frac{f_a}{F_a} = \frac{11.88}{10.89} = 1.09$$

The value calculated by STAAD is 1.09

**Member 9**, Size L5x5x3/8, L = 5.657 ft., a = 3.61 in<sup>2</sup>

From observation Load case 1 will govern, Forces

Fx = 48.44 kip (compression)

**Allowable Stress Calculation:**

**Axial Compression:**

Critical  $(KL/r)_y = 1.0 \times 5.657 \times 12 / 0.99 = 68.57$

$$C_c = \sqrt{\frac{2\pi^2 E}{F_y}} = \sqrt{\frac{2\pi^2 * 29000}{36}} = 126.099$$

As  $(KL/r)_y < C_c$ , the allowable axial stress in compression

$$F_a = \frac{F_y}{F.S.} \left[ 1 - \frac{(KL/r)^2 F_y}{4\pi^2 E} \right] = \frac{36}{2.12} \left[ 1 - \frac{(68.57)^2 36}{4\pi^2 29000} \right] = 14.47 \text{ ksi}$$

**Actual Stress Calculation:**

Actual stress ( $f_a$ ) = 48.44 / 3.61 = 13.42 ksi.

$$\text{Ratio } \frac{f_a}{F_a} = \frac{13.42}{14.47} = 0.927$$

The value calculated by STAAD is 0.928

### **2.13.2 AASHTO (LRFD)**

The following outlines the implementation of the AASHTO Standard Specifications for Highway Bridges (LRFD, 1998) which has been implemented in STAAD.Pro.

#### **General Comments**

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.

#### **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 by the user 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

$$P_r = \phi_y P_{ny} = \phi_y F_y A_g$$

$$P_r = \phi_u P_{nu} = \phi_u F_u A_n U$$

$P_{ny}$  = Nominal tensile resistance for yielding in gross section (kip)

$F_y$  = Yield strength (ksi)

$A_g$  = Gross cross-sectional area of the member (in<sup>2</sup>)

$P_{nu}$  = 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.

$\phi_y$  = resistance factor for yielding of tension member

$\phi_u$  = 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:

$$\lambda = \left( \frac{Kl}{r_s \pi} \right)^2 \frac{F_y}{E}$$

if  $\lambda \leq 2.25$

Nominal compressive resistance,  $P_n = 0.66^\lambda F_y A_s$

if  $\lambda > 2.25$

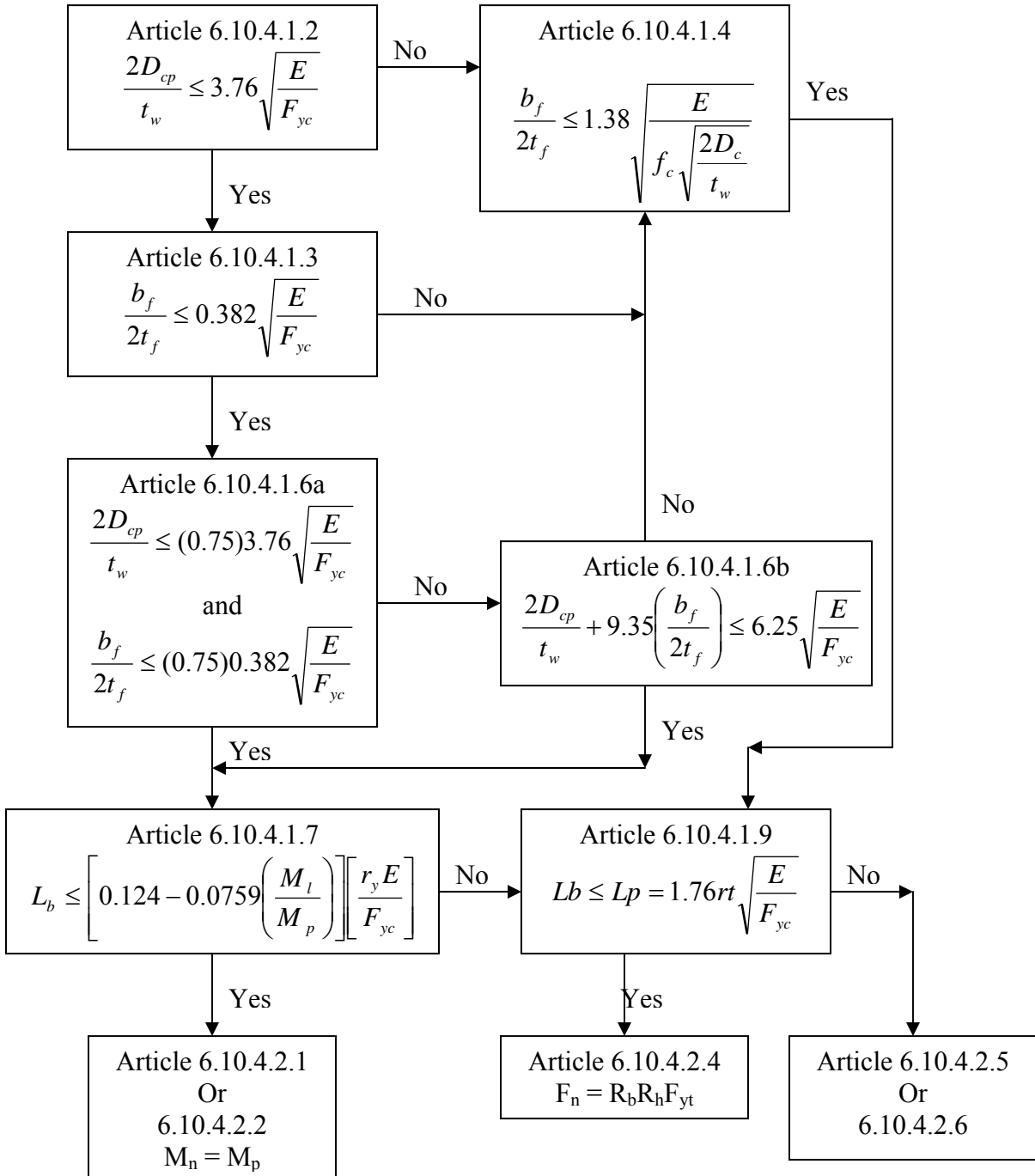
Nominal compressive resistance,  $P_n = \frac{0.88 F_y A_s}{\lambda}$

$A_s$  = 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.



## **Shear Strength**

The nominal shear resistance of un-stiffened webs of homogeneous girders shall be calculated as.

$$\text{If } \frac{D}{t_w} \leq 2.46 \sqrt{\frac{E}{F_{yw}}}, \text{ then}$$

$$V_n = V_p = 0.58 F_{yw} D t_w$$

$$\text{If } 2.46 \sqrt{\frac{E}{F_{yw}}} < \frac{D}{t_w} \leq 3.07 \sqrt{\frac{E}{F_{yw}}}, \text{ then}$$

$$V_n = 1.48 t_w^2 \sqrt{E F_{yw}}$$

$$\text{If } \frac{D}{t_w} > 3.07 \sqrt{\frac{E}{F_{yw}}}, \text{ then}$$

## **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:

$$\text{If } \frac{P_u}{P_r} < 0.2, \text{ then}$$

$$\frac{P_u}{2.0 P_r} + \left( \frac{M_{ux}}{M_{rx}} + \frac{M_{uy}}{M_{ry}} \right) \leq 1.0$$

$$\text{If } \frac{P_u}{P_r} \geq 0.2, \text{ then}$$

$$\frac{P_u}{P_r} + \frac{8.0}{9.0} \left( \frac{M_{ux}}{M_{rx}} + \frac{M_{uy}}{M_{ry}} \right) \leq 1.0$$

## 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 2.4 - AASHTO (LRFD) Design Parameters**

Parameter Name	Default Value	Description
<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>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>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>NSF</u>	1.0	Net Section Factor. Ratio of (Net Area)/(Gross Area)
<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>NSF</u>	1.0	Net section factor for tension members.
<u>UNT</u>	Member Length	Unsupported length of top flange. Used for calculating the moment of resistance when top of beam is in compression.
<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>BEAM</u>	1.0	Identify where beam checks are performed: 0 = 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>MAIN</u>	0.0	Flag for checking slenderness limit: 0.0 =check for slenderness 1.0 =suppress slenderness check
<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>	0	Minimum allowed section depth (in current length units) for a section to be selected with the SELECT command.

<u>DFD</u>	0	"Deflection Length"/Max allowable local deflection If set to 0, (default) then no deflection check is performed.
<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".

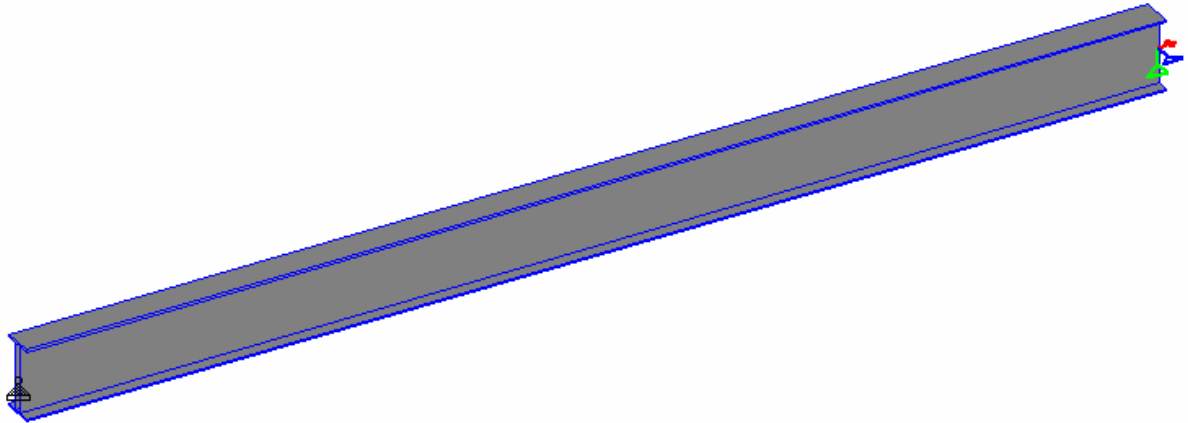
## AASHTO (LRFD) Verification Problem

The following compares the solution of a design performed using STAAD.Pro against a hand calculation.

**TYPE:** AASHTO (LRFD) Steel Design.

**REFERENCE:** Attached 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.



## VERIFICATION PROBLEM HAND CALCULATION

$L = 48 \text{ ft.}$ ,  $A_{xx} = 154.3997 \text{ in}^2$ ,  $S_z = 1199.66 \text{ in}^3$ ,  $DWW = 36 \text{ in.}$ ,  $TWW = 3.6 \text{ in.}$ ,  
 $BFF = 16 \text{ in.}$ ,  $TFF = 1 \text{ in.}$ ,  $IZZ = 21593.85 \text{ in}^4$ ,  $IYY = 814.85 \text{ in}^4$

From observation Load case 1 will govern,

$M_z = 2624 \text{ kip-ft}$

## Allowable Resistance Calculation:

### Axial Compression, Clause 6.9.4:

$$\begin{aligned} r_y &= \sqrt{I_{yy}/A_{xx}} \\ &= \sqrt{814.85/154.3997} \\ &= \mathbf{2.297} \end{aligned}$$

$$\begin{aligned} r_z &= \sqrt{I_{zz}/A_{xx}} \\ &= \sqrt{21593.85/154.3997} \\ &= \mathbf{11.826} \end{aligned}$$

$$(kL/r)_y = 0.333 \times 576 / 2.297 = 83.504$$

$$(kL/r)_z = 1 \times 576 / 11.826 = 48.706$$

$$(kL/r)_{\text{Critical}} = 83.504$$

$$(kL/r)_{\text{limit}} = 120$$

So safe

### Calculation of Width/Thickness ratio for axial compression

Plate buckling coefficient of web,  $k_{\text{web}} = 1.49$

Plate buckling coefficient of flange,  $k_{\text{flange}} = 0.56$

$$\begin{aligned} (\text{Width/Thickness})_{\text{web}} &= (36 - 2 \times 1) / 3.6 \\ &= 9.445 \end{aligned}$$

$$\begin{aligned} (\text{Width/Thickness})_{\text{flange}} &= 16 / (2 \times 1) \\ &= 8.0 \end{aligned}$$

$$\begin{aligned}
(\text{Width/Thickness})_{\text{web\_limit}} &= k_{\text{web}} \times \sqrt{\frac{E}{FYLD}} \\
&= 1.49 \times \sqrt{\frac{29732.7}{36}} \\
&= 42.820
\end{aligned}$$

$$(\text{Width/Thickness})_{\text{web}} < (\text{Width/Thickness})_{\text{web\_limit}}$$

**Clause 6.9.4.2 satisfied**

$$\begin{aligned}
(\text{Width/Thickness})_{\text{flange\_limit}} &= k_{\text{flange}} \times \sqrt{\frac{E}{FYLD}} \\
&= 0.56 \times \sqrt{\frac{29732.7}{36}} \\
&= 16.094
\end{aligned}$$

$$(\text{Width/Thickness})_{\text{flange}} < (\text{Width/Thickness})_{\text{flange\_limit}}$$

**Clause 6.9.4.2 satisfied**

Calculation of Slenderness factor,  $\lambda$

$$\begin{aligned}
\lambda_{zz} &= \left( \frac{KL}{r_{zz}\pi} \right)^2 \frac{F_y}{E} = \left( \frac{1 \times 576}{11.826 \times 3.14159} \right)^2 \frac{36}{29732.7} \\
&= 0.291
\end{aligned}$$

$$\begin{aligned}
\lambda_{yy} &= \left( \frac{KL}{r_{yy}\pi} \right)^2 \frac{F_y}{E} = \left( \frac{0.333 \times 576}{2.297 \times 3.14159} \right)^2 \frac{36}{29732.7} \\
&= 0.855
\end{aligned}$$

$$\lambda = 0.855$$

$$\lambda < 2.25$$

Equation 6.9.4.1-1 is used

$$\begin{aligned}
 \text{The Nominal Compressive Resistance, } P_n &= 0.66^\lambda \times F_y \times A_{xx} \\
 &= (0.66)^{0.855} \times 36 \times 154.3997 \\
 &= 3896.359 \text{ kip}
 \end{aligned}$$

$$\begin{aligned}
 \text{The factored compressive resistance, } P_r &= \phi_c P_n \\
 &= 0.9 \times 3896.359 \\
 &= 3506.723 \text{ kip}
 \end{aligned}$$

### **Bending Major Axis:**

$$\text{Where, } C_b = 1.75 + 1.05(M_1/M_2) + 0.3x(M_1/M_2)^2$$

$$\text{Here } M_1 = 0, M_2 = 0 \text{ so } C_b = 1.75$$

#### Check Clause 6.10.2.1

$$I_{yc} = \text{CompFlangeThick} \times \frac{\text{CompFlangeWidth}^3}{12}$$

$$= 1.0 \times \frac{16^3}{12}$$

$$= 341.334$$

$$\text{ratio} = I_{yc} / I_{yy}$$

$$= 341.334 / 814.85$$

$$= 0.419$$

$$\text{Ratio} > 0.1 \text{ and ratio} < 0.9$$

Check Clause 6.10.2.1, satisfied

#### Check Clause 6.10.2.2

$$\text{Ratio} = \frac{2D_c}{t_w}$$

$$= \frac{2 \times 17.0}{3.6}$$

$$= 9.445$$

$$\text{Ratio} < 6.77 \times \sqrt{\frac{E}{FYLD}}$$

Check Clause 6.10.2.2, satisfied

Check Clause 6.10.4.1.2

Calculation of depth of the web in compression at the plastic moment,  $D_{cp}$  (Clause 6.10.3.3.2)

$$\text{Area of Web} = (36 - 2 \times 1) \times 3.6 = 122.4$$

$$\text{Flange area in Tension} = 16 \times 1 = 16$$

$$\text{Flange area compression} = 16 \times 1 = 16$$

$$D_{cp} = (DWW / (2 \times \text{Area of Web} \times FYLD))$$

$$\times ((FYLD \times \text{Flange area in Tension}) + (FYLD \times \text{Area of Web}) - (FYLD \times \text{Flange area compression}))$$

$$= (34 / (2 \times 122.4 \times 36)) \times ((36 \times 16) + (36 \times 122.4) - (36 \times 16))$$

$$= 17 \text{ in}$$

$$(2 \times D_{cp}) / TWW = (2 \times 17) / 3.6$$

$$= 9.445$$

$$(D/T)_{web\_limit} = 3.76 \times \sqrt{\frac{E}{FYLD}}$$

$$= 3.76 \times \sqrt{\frac{29732.7}{36}}$$

$$= 108.057$$

$$(2 \times D_{cp}) / TWW < (D/T)_{web\_limit}$$

### Check Clause 6.10.4.3

$$\begin{aligned} (B/t)_{\text{flange}} &= 16.0 / 2 \times 1.0 \\ &= 8.0 \end{aligned}$$

$$\begin{aligned} (B/t)_{\text{flange\_limit}} &= 0.382 \times \sqrt{\frac{E}{FYLD}} \\ &= 0.382 \times \sqrt{\frac{29732.7}{36}} \\ &= 10.978 \end{aligned}$$

$$(B/t)_{\text{flange}} < (B/t)_{\text{flange\_limit}}$$

### 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

$$Mp_z = P_z \times F_y$$

$$= 1600.40 \times 36$$

$$= 57614.4$$

$$\begin{aligned} (L_b)_{\text{limit}} &= \left[ 0.124 - 0.0759 \left( \frac{M_1}{Mp_z} \right) \right] \left[ \frac{r_y E}{F_y} \right] \\ &= \left[ 0.124 - 0.0759 \left( \frac{0.0}{57614.4} \right) \right] \left[ \frac{2.297 \times 29732.7}{36} \right] \\ &= 0.124 \end{aligned}$$

Unsupported length,  $L_b = 576$

$$L_b > (L_b)_{\text{limit}}$$

Check clause 6.10.4.1.9

$$I_{rt} = \text{CompFlangeThick} \times \frac{\text{CompFlangeWidth}^3}{12} + \left( \left( \frac{\text{NEUTRAL\_AXIS} - \text{CompFlangeThick}}{3} \right) \times \frac{\text{Tw}^3}{12} \right)$$

$$= 1.0 \times \frac{16^3}{12} + \left( \left( \frac{18-1}{3} \right) \times \frac{3.6^3}{12} \right)$$

$$= 363.365$$

$$A_{rt} = \text{CompFlangeArea} + \left( \frac{\text{NEUTRAL\_AXIS} - \text{CompFlangeThick}}{3} \right) \times \text{Tw}$$

$$= 16 + \left( \frac{18-1.0}{3} \right) \times 3.6$$

$$= 36.4$$

Radius of gyration of a notional section comprised of the compression flange of the steel section plus one-third of the depth of the web in compression taken

about the vertical axis,  $r_t = \sqrt{\frac{I_{rt}}{A_{rt}}}$

$$= \sqrt{\frac{363.365}{36.4}}$$

$$= 3.159$$

$$L_p = 1.76 \times 3.159 \times \sqrt{\frac{29732.7}{36}}$$

$$= 159.782$$

$L_b > L_p$

Clause 6.10.4.2.6

$$I_{yc} = \text{CompFlangeThick} \times \frac{\text{CompFlangeWidth}^3}{12}$$

$$= 1.0 \times \frac{16^3}{12}$$

$$= 341.334$$

$$L_r = 4.44 \sqrt{\frac{I_{yc} d E}{S_{xc} F_y}}$$

$$= 4.44 \sqrt{\frac{341.334 \times 36.00}{1199.66} \times \frac{29732.7}{36}}$$

$$= 408.376$$

Minimum radius of gyration of the compression flange taken about the vertical axis,

$$r_t = \sqrt{\frac{I_{yc}}{A_{rt}}}$$

$$= \sqrt{\frac{341.334}{16 \times 1.0}}$$

$$= 4.619$$

Hybrid factor,  $R_h = 1.0$

For Homogeneous sections, Hybrid factors shall be taken as 1.0 (clause 6.10.4.3.1)

As per clause 6.10.4.3.2

Load-shedding factor,  $R_b$ , clause 6.10.4.3.2

If,  $\text{CompFlangeArea} \geq \text{TensFlangeArea}$

$$\lambda_b = 5.76$$

$$Dc = 17.00$$

$$\begin{aligned}(W/t)_{web} &= 2Dc/Tw \\ &= 2 \times 17.00 / 1.8 \\ &= 18.889\end{aligned}$$

$$\begin{aligned}(W/t)_{web\_limit} &= \lambda_b \times \sqrt{\frac{E}{FYLD}} \\ &= 5.76 \times \sqrt{\frac{29732.7}{36}} \\ &= 165.534\end{aligned}$$

$$As, (W/t)_{web} < (W/t)_{web\_limit}$$

$$Rb\_comp = 1.0$$

$$Rb\_ten = 1.0$$

$$Lr < Lb$$

$$\begin{aligned}Mnz\_compression &= Cbz \times Rb\_comp \times Rh \times \frac{My}{2} \times \left(\frac{Lr}{Lb}\right)^2 \\ &= 1.75 \times 1.0 \times 1.0 \times \frac{36 \times 1199.66}{2} \times \left(\frac{408.376}{576}\right)^2 \\ &= 18995.213\end{aligned}$$

$$\begin{aligned}Mnz\_tension &= Rb\_ten \times Rh \times Fyld \times Z\_ten \\ &= 1.0 \times 1.0 \times 36 \times 1199.66 \\ &= 43187.76\end{aligned}$$

$$Mn = 18995.213 \text{ kip-in}$$

$$\begin{aligned}\text{Resisting Moment, } Mr &= Qf \times Mn \\ &= 1.0 \times 18995.213\end{aligned}$$

$$= 18995.213 \text{ kip-in}$$

$$\text{Actual Moment} = 31488 \text{ kip-in}$$

$$\text{Interaction ratio} = 31488 / 18995.213$$

$$= 1.657$$

Ratio calculated by STAAD is 1.658

## **AD.2006-1002.2.2 Updated Parameters on Indian IS 1893 Static Seismic Loading Command**

### **Purpose**

The static equivalent method for creating loads as per the Indian seismic code, IS 1893 has been enhanced with the option for checking soft storeys.

### **Description**

The command structure has been updated and section 5.31.2.5 of the Technical Reference manual should be replaced with the following:-

### 5.31.2.5 Definition of Lateral Seismic Load per Indian IS:1893 (Part 1) – 2002 Code

#### Purpose

This feature enables one to generate seismic loads per the IS:1893 specifications using a static equivalent approach.

#### Description

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 using the “SET Z UP” command.

#### Methodology

The design base shear is computed by STAAD in accordance with the IS: 1893(Part 1)-2002 equation 7.5.3.

$$V = A_h \cdot W \quad \text{where, } A_h = \frac{Z}{2} \cdot \frac{I}{R} \cdot \frac{S_a}{g}$$

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. User provides seismic zone co-efficient and desired "1893(Part 1)-2002specs" through the DEFINE 1893 LOAD command.
2. Program calculates the structure period (T).
3. Program calculates Sa/g utilizing T.
4. Program calculates V from the above equation. W is obtained from SELFWEIGHT, JOINT WEIGHT(s) and MEMBER WEIGHT(S) provided by the user 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 Section  
5.32.12*

#### General format:

**DEFINE 1893 LOAD**

**ZONE f1      1893-spec**

**SELFWEIGHT  
JOINT WEIGHT**

**joint-list WEIGHT w**  
**MEMBER WEIGHT**

**mem-list**      { **UNI v1 v2 v3** }  
                  { **CON v4 v5**     }

### **CHECK SOFT STORY**

**1893-Spec= {RF f2, I f3, SS f4, (ST f5), DM f6, (PX f7), (PZ f8), (DT f9)}**

where,

**Zone f1** = Seismic zone coefficient . Refer to Table 2 of IS:1893(Part 1)-2002.

**RF f2** = Response reduction factor. Refer Table 7 of IS: 1893 (Part 1) -2002.

**I f3** = 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.

**SS f4** = 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.

**ST f5** = 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.

**DM f6** = 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.

**PX f7** = 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.

**PZ f8** = 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.

**DT f9** = Depth of foundation below ground level. It should be defined in current unit. If the depth of foundation is 30 m or below, 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  $0.5A_h$ .

**w** = joint weight associated with joint list

**UNI** . = specifies 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.

**CON** = specifies a concentrated force with a value of v4 applied at a distance of v5 (from the start of the member). If v5 is omitted, the load is assumed to act at the center of the member.

**CHECK SOFT STORY** indicates that soft story checking will be performed. If omitted from input, there will be no soft story checking. See below for details on soft storey checking.

Note: 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.

### **Soft Storey Checking**

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 storey checking.

#### **Stiffness Irregularities – Soft Storey**

As per this provision of the code, a soft storey is one in which the lateral stiffness is less than 70 percent of that in the storey above or less than 80 percent of the average lateral stiffness of the three storey above.

#### **Stiffness Irregularities – Extreme Soft Storey**

As per this provision of the code, a extreme soft storey is one in which the lateral stiffness is less than 60 percent of that in the storey above or less than 70 percent of the average lateral stiffness of the three storey above.

Thus, if any storey 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 storey. 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 shear walls at a particular floor level constitute the total lateral stiffness of that particular storey or floor level. The program checks soft storey of a building along both global X and Z directions respectively.

### Identification of floor level

Following two ways can identify floor level.

1. Program calculated
2. User defined

#### 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.

#### User defined

Floor heights should be defined before using the DEFINE 1893 or any primary response spectrum load case.

#### General format:

##### FLOOR HEIGHT

$h_1; h_2; h_3; \dots; h_i -$   
 $h_{i+1}; \dots; h_n$

where,

$h_1$  .....  $h_n$  are the different floor heights in current length unit and  $n$  is the number of floor levels.

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 storey checking.

### **Example**

See section 5.32.12 for an example of the correct usage of this command.

## **AD.2006-1002.2.3 Updated Parameters on Indian IS 1893 Response Spectrum Command**

### **Purpose**

The method for creating a response spectrum to the Indian seismic code, IS 1893 has been enhanced with the option for checking soft storeys and documentation on the torsion option added.

### **Description**

The command structure has been updated and section 5.32.10.1.1 of the Technical Reference manual should be replaced with the following:-

### **5.32.10.1.1 Response Spectrum Specification as per the Indian IS:1893 (Part 1)-2002 Code for Dynamic Analysis**

#### **Methodology**

The design lateral shear force at each floor in each mode is computed by STAAD in accordance with the IS: 1893 (Part 1) -2002 equation 7.8.4.5c and 7.8.4.5d.

$$Q_{ik} = A_k * \phi_{ik} * P_k * W_k \text{ and } V_{ik} = \sum_{i=i+1}^n Q_{ik}$$

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. User provides the value for  $\frac{Z}{2} * \frac{I}{R}$  as factors for input spectrum
2. Program calculates time periods for first six modes or as specified by the user.
3. Program calculates Sa/g for each mode utilizing time period and damping for each mode.
4. The program calculates design horizontal acceleration spectrum Ak 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 method (CQC or SRSS or ABS or TEN or CSM) as defined by the user to get the final results.

#### **General format:**

**SPECTRUM {Method} 1893 (TORSION) X f1 Y f2 Z f3 ACC (SCALE f4)  
(DAMP f5 or MDAMP or CDAMP) (MIS f6) (ZPA f7)  
SOIL TYPE f8 or, {P1,V1; P2,V2; P3,V3;.....}  
CHECK SOFT STORY**

The data in the first line above must be on the first line of the command, second line data can be on the first or second line with all but last ending with a hyphen. The third line (SOIL TYPE parameter) must be in a separate line. The fourth line, if present in the input, will perform soft story checking.

where,

**Method = SRSS or CQC or ABS or CSM or TEN**

**SRSS, CQC, ABS, CSM** and **TEN** are methods of combining the responses from each mode into a total response. SRSS stands for square root of summation of squares, CQC for complete quadratic combination and 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 is 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.

Note that if SRSS is given the program will internally check whether or not there are any closely spaced modes. If any such modes are found, it will switch over to the CSM method. In 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 CQC method.

**1893** indicates the analysis as per IS:1893(Part 1)-2002 procedure.

**TOR** indicates to consider horizontal torsional moment arising due to eccentricity between the centre of mass and centre of rigidity. If TOR is entered on any one spectrum case it will be used for all spectrum cases.

**X Y Z** f1, f2, f3 are the factors for the input spectrum to be applied in X, Y, & Z directions. These are the product of  $\frac{Z}{2} * \frac{I}{R}$ . Any one or all directions can be input. Directions not provided will default to zero.

**ACC** indicates Acceleration spectra will be entered.

**SCALE f4** = 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. Default value is 1.0. \*\*

\*\* If site specific spectra curve is used then f4 value is to be multiplied by the scale factor by which spectra data will be multiplied. Usually to factor g's to length/sec<sup>2</sup> units.

**DAMP , MDAMP and CDAMP.** Select source of damping input.

**DAMP f5** = Damping ratio for all modes. Default value is 0.05 (5% damping if 0 or blank entered).

**DAMP** indicates to use the f5 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** is the composite damping of the structure calculated for each mode. One must specify damping for different materials under CONSTANT specification.

**MIS f6** = 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. 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. If the MIS parameter is entered on any spectrum case it will be used for all spectrum cases.

**ZPA f7** = For use with MIS option only. Defaults to 33 Hz if not entered. Value is printed but not used if MIS f6 is entered.

**SOIL TYPE f6** indicates the types of soil. f5 is 1 for rocky or hard soil, 2 for medium soil and 3 for soft soil sites. Depending upon time period, types of soil and damping, average response acceleration coefficient, Sa/g is calculated.

Or,

**{P1,V1; P2,V2; P3,V3;.....Pn,Vn}** : Data is part of input immediately following spectrum command. Period - Value 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<sup>2</sup>). 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).

**CHECK SOFT STOREY** indicates that soft story checking will be performed. If omitted from input, there will be no soft story checking.

Note:

1. The design base shear  $V_B$  calculated from Response Spectrum method is compared with the base shear  $V_b$  calculated empirical formula for fundamental time period. If  $V_B$  is less than  $V_b$ , all 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**  
**ZONE f1 1893-spec**  
**SELFWEIGHT**  
**JOINT WEIGHT**  
**joint-list WEIGHT w**  
**MEMBER WEIGHT**

mem-list      { **UNI v1 v2 v3** }  
                   { **CON v4 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 that this does not cause the program to calculate frequency of the structure. Either the ST parameter or PX and PZ parameters must be entered.)

2. 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:

$$S_a = A e^{-\xi} + \frac{B}{\xi}$$

where,

$S_a$  = Spectrum ordinate

$\xi$  = damping ratio

A, B = Constants

The constants A and B are determined using two known spectrum ordinates  $S_{a1}$  &  $S_{a2}$  corresponding to damping ratios  $\xi_1$  and  $\xi_2$  respectively for a particular time period and are as follows :

$$A = \frac{S_{a1}\xi_1 - S_{a2}\xi_2}{\xi_1 e^{-\xi_1} - \xi_2 e^{-\xi_2}}$$

$$B = \frac{\xi_1 \xi_2 (S_{a2} e^{-\xi_1} - S_{a1} e^{-\xi_2})}{\xi_1 e^{-\xi_1} - \xi_2 e^{-\xi_2}}$$

where,  $\xi_1 < \xi < \xi_2$

3. The storey drift in any storey shall not exceed 0.004 times the storey height as per Clause 7.11.1. To check this following input should be given after analysis command.

### **PRINT STOREY DRIFT**

Warning message will be printed if storey drift exceeds this limitation.

4. If any soft storey (as per definition in Table 5 of IS:1893-2002) is detected a warning message will be printed in the output.

### **Torsion**

As defined in the IS1893-2002 code, provision should 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.

### **Steps**

For a particular mode following steps are performed.

1. Lateral storey shear for each floor is established.
2. Torsional moments M1 and M2 are found out at each floor for two design eccentricities.
3. The nodal forces corresponding to M1 and M2 are found out for each floor.
4. The case, for which sum of nodal forces is maximum, is selected as the critical loading due to torsion for that floor.
5. Static analysis of the structure is performed with these nodal forces due to torsion.
6. The results (i.e. member force, displacement, support reaction, etc) are added to the corresponding modal response quantities.

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.

Note: After the analysis is complete following files are generated.

1. Story shear for each mode for each load case is given in the file **<filename>\_RESP1893.txt**.
2. Rotational stiffness of each floor is given in the file **<filename>\_ROT1893.txt**.
3. Centre of mass, centre 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**.

## **Soft Storey Checking**

For details on the Soft Storey Checking option, see the chapter detailed in Section 5.31.2.5 Definition of Lateral Seismic Load per Indian IS:1893 (Part 1) – 2002 Code.

## **AD.2006-1002.3 Features Affecting the Steel Design Mode**

Several new features have been added and existing features have been modified in the Steel Designer section of the program, also known as the Steel Designer Mode. These are explained in the following pages.

## **AD.2006-1002.3.1 Access to Physical Members Created in Pre-Processing Mode.**

### **Purpose**

Physical members can be created in the Pre-Processor mode and their definition stored in the STAAD.Pro input file (\*.STD). Physical members that have been created in the Pre-Processing Mode are now available in the Steel Designer Post Processing Mode.

Previous versions of STAAD.Pro would require these members being re-defined in the Steel Designer Mode.

### **Description**

Create Physical Members as defined in the manual section AD.2006.1.1. Perform an analysis and enter the Steel Designer Mode. Go to the Member Design | Member Setup Page, and note that the members that were created in the Pre-Processing Mode are ready to be utilised in the Steel Designer Mode.

## **AD.2006-1002.4 Features Affecting the Concrete Design Mode**

Several new features have been added and existing features have been modified in the RC Designer section of the program, also known as the Concrete Design Mode. These are explained in the following pages.

## AD.2006-1002.4.1 DXF Output of Beam and Column Designs.

### Purpose

It is now possible to write the completed design drawing to a raw DXF file, as well as to MultiREBAR format, so that it may be finished off by a detailer. The MultiREBAR format drawing requires a suitable installation of the MultiSuite application to visualize the drawing. The raw DXF format can be viewed by any application that can view DXF files.

### Description

Click on the "Drawing" or "Schedule" page (according to design code), a dialog opens as shown in section 4.1.15.1 of the help, except for the addition of a drop list to choose the output format.

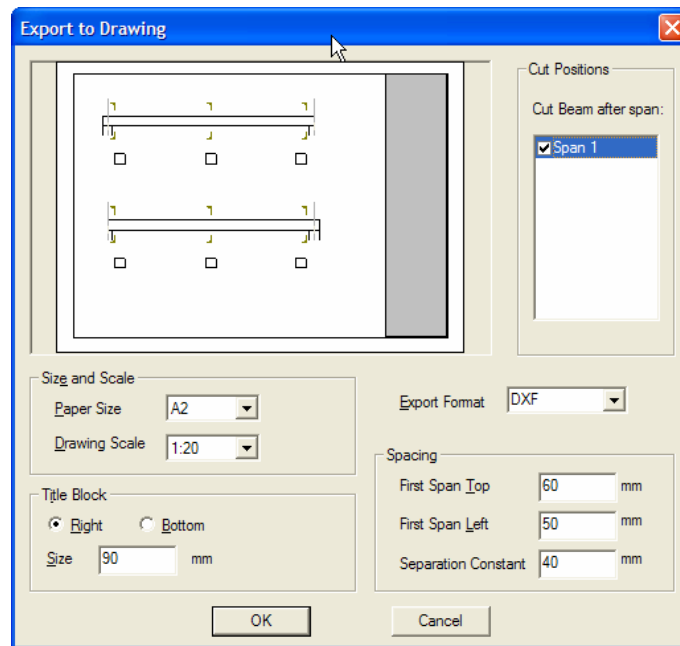


Figure 8

The MultiREBAR export remains as before, the DXF format is new. For DXF output, the concrete outlines, covers and centre lines are all output as simple lines to appropriate layers. The bars are sent as polylines, again to separate layers. The schedule is not exported, but may be printed from the normal report setup. In all other respects the dialog works in the same way as described in the help at 4.1.15.

## **AD.2006-1002.5 Features Affecting the Advanced Slab Design Mode**

The Mode previously known as the ADAPT Slab design has been redefined as the Advanced Slab Design and has been enhanced to support integration with both RAM-Concept and ADAPT-Builder. The following describes the correct use of this mode.

## AD.2006-1002.5.1 Advance Slab Design Mode

### Purpose

Advanced Slab Design Mode enables concrete slabs defined with or without holes to be designed using the RAM-Concept or ADAPT-Builder software. In order to create Slab models compatible with ADAPT-Builder, additional ADAPT Link license is required and ADAPT-Builder software needs to be installed. To generate slab models compatible with RAM-CONCEPT with in STAAD.Pro, apart from RAM-CONCEPT installation on the machine, no additional license is required.

RAM-CONCEPT is a Bentley product and you may contact your account representative at Bentley to procure RAM-CONCEPT.

ADAPT is a third party product, which has its own license and installation. Contact your STAAD.Pro supplier for more details on purchasing this application if you do not currently own a license.

### Description

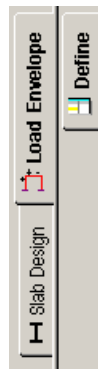
The following steps are to be followed to design a Slab from STAAD.Pro using RAM-CONCEPT or ADAPT:

1. Open the input file (containing the slab to be designed) in STAAD.Pro.
2. Before going into the Advanced Slab Design module, make sure that the input file is analyzed with STAAD.Pro.
3. Now click on the “**Advanced Slab Design**” tab, as indicated in the figure below. This tab will be visible only when both the pre-conditions as stated above are satisfied.



Figure 9

4. The user will be able to view the “**Load Envelope**” and “**Slab Design**” tabs at the top-left side of the model window, as shown in the figure below.



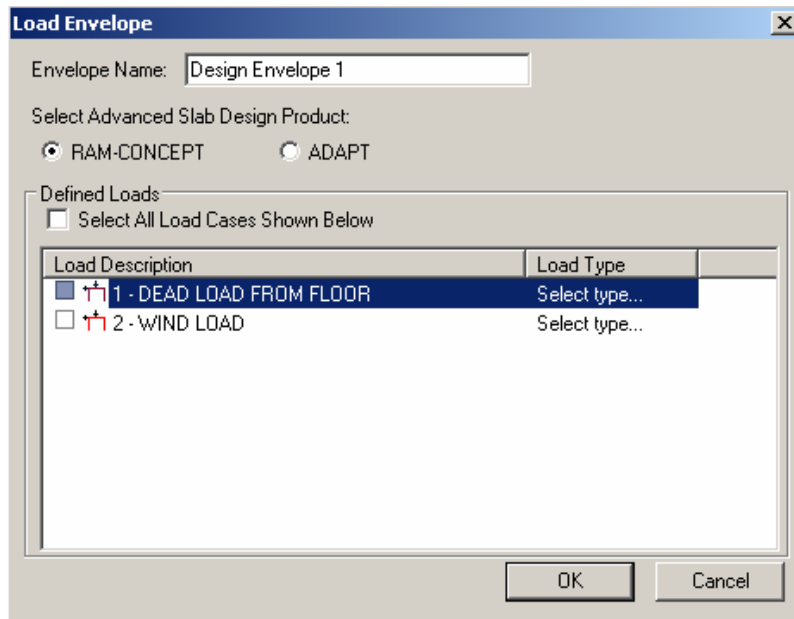
**Figure 10**

5. While in the **Load Envelope | Define** page, click on the [**New Envelope**] button located at the bottom-right corner of the screen.



**Figure 11**

6. The “**Load Envelope**” dialog-box will open, as shown below.



**Figure 12**

If the user has a valid ADAPT link license available on the system, they will be given the option to create load envelopes for RAM-CONCEPT based slab design or ADAPT based design. If the user does not have a valid ADAPT Link license available, the option will not be made available and the created load envelope will be compliant to RAM-CONCEPT based designs only.

A Load Envelope name should be entered, being defined by the chosen load cases by clicking the check boxes corresponding to each load case.

In addition to that the user must specify the “Load Type” for each individual load cases. The “Load Type”-s are classified into Dead / Live / Lateral Load as shown below. Please note that list of “Load Type” will be customized based on user’s choice of “RAM-CONCEPT” based design or “ADAPT” based design.

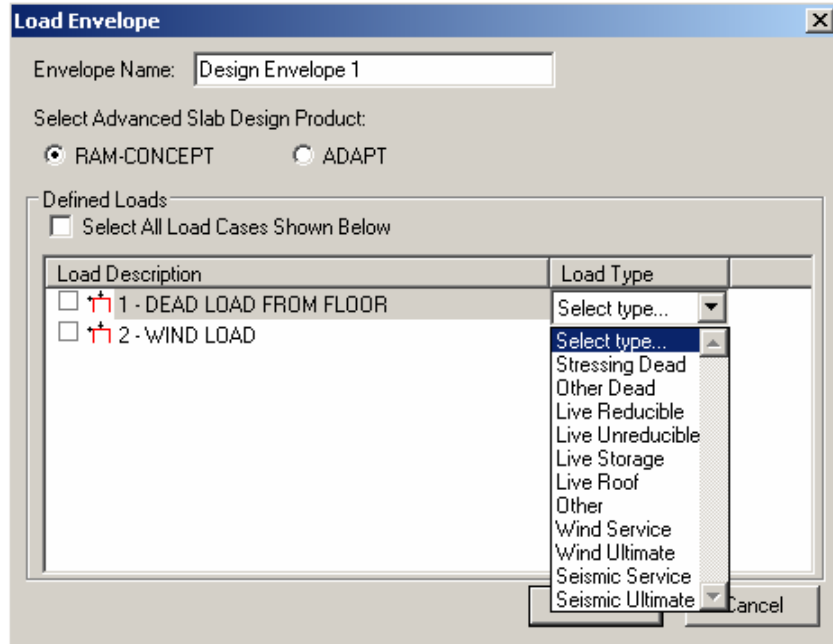


Figure 13

7. Create as many as Load Envelopes as are required for the slab designs. The Load Envelopes are listed in the Envelopes table displayed on the top-right corner of the screen.

Ref:	Name	Load Cases
E1	Design Envelope 1	1 + 2 + 3
E2	Design Envelope 2	2 + 3

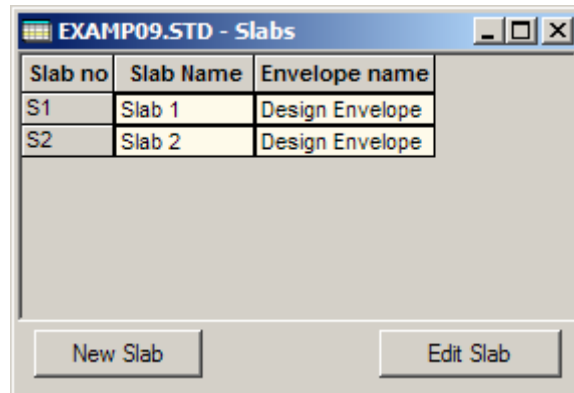
Figure 14

8. Any Load Envelope can be edited by selecting the particular envelope and then by clicking the “Edit Envelope” button.
9. To define a slab model for design, click on the “Slab Design” tab and select the geometry-cursor to select all the geometric entities in a particular floor along with all columns connected to that floor from top and bottom (i.e. beam and plates entities). The selection cursor defaults to “Geometry Cursor” when “Slab Design” tab is selected. The user may also choose the wall plate elements which are connected to the floor from top and bottom.  
**Warning**, please note that wall elements are properly handled by the interface only if RAM-CONCEPT based design is used. While creating an ADAPT compliant slab model, it is suggested that wall elements should NOT be chosen.

After selecting the plate-elements and the associated beams and columns, click the [**New Slab**] button to define the slab. A “**Slab Definition**” dialog-box is displayed from which the required Load Envelope associated with the Slab definition should be selected.

Note that if the selection of plates is such that the plates form two separated entities, say plates that form parallel floors, then STAAD.Pro will create multiple separate slab entities.

10. Multiple slabs can be defined and prepared ready to be designed. The created slab definitions are listed in the Slabs Table displayed in top-right corner of the screen as shown.



**Figure 15**

11. The created slab definitions can be edited, deleted, highlighted by geometry after selecting one Slab definition, and then by selecting the corresponding option from the context (right click) menu. Based on the type of slab model created by the user, either RAM-CONCEPT or ADAPT based design menu items will be available.

12. The slabs are then passed into the appropriate Advanced Slab Design application by selecting the required “Export” or “Run” option from the context (right click) menu. The “Export” options create the required data files, but program control remains with STAAD.Pro. The “Run” options both create the data file and pass control over to the selected Advanced Slab Design application, along with the newly created data.

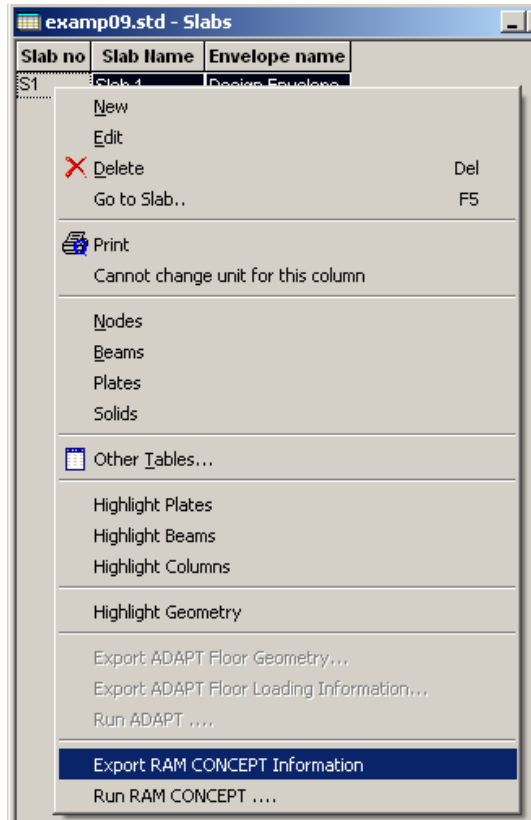


Figure 16

13. Selecting the option “Run RAM CONCEPT...”, the security is checked and if in order, the objects in the selected slab are shown with the option to select only certain objects:-

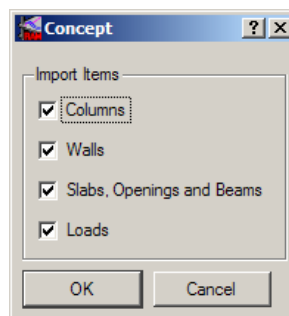


Figure 17

14. Click on the [OK] button and the list of STAAD.Pro objects imported into the RAM CONCEPT application is listed. For further information on RAM CONCEPT, refer to the applications own documentation.

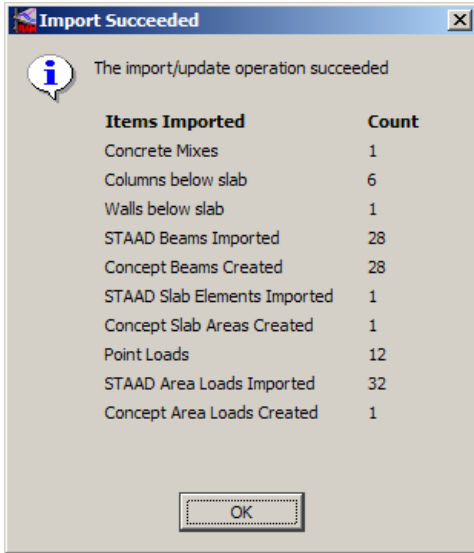


Figure 18

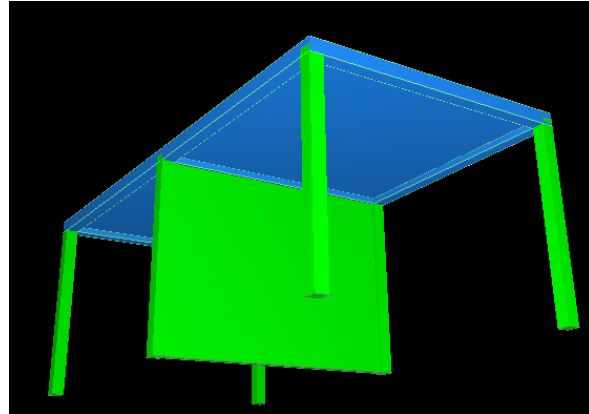


Figure 19

## **AD.2006-1002.6 Features Affecting the Piping Mode**

STAAD.Pro can utilise the pipe layout and reactions created in the applications ADLPipe or AutoPipe. The pipe model can be imported in the Piping Mode. The following describes the method of using the Piping Mode and features recently added to this mode.

## AD.2006-1002.6.1 Piping Mode

### Purpose

The piping system in a plant model is supported by a number of structural components – such as pipe racks, cradles or shoes – constructed exclusively for that purpose or from the existing beams, columns and slabs of the structure. To facilitate the design of structures to sustain the loading imparted by the piping system STAAD.Pro has introduced a seamless transfer of support reactions calculated from an ADLPIPE or AutoPipe analysis to the structure modelled in STAAD.Pro.

We can view the entire piping system along with the structure in STAAD.Pro and the reactions are transferred to the structure once the piping nodes are co-related with the corresponding nodes on the structure.

### Description

To transfer the support reactions from piping analysis to STAAD.Pro requires performing the following steps.

#### 1) Set the mode to 'Piping':-

After opening or creating a suitable support structure in the STAAD.Pro Modelling Mode, enter the **Piping Mode** by either selecting the menu option **Mode | Piping....**

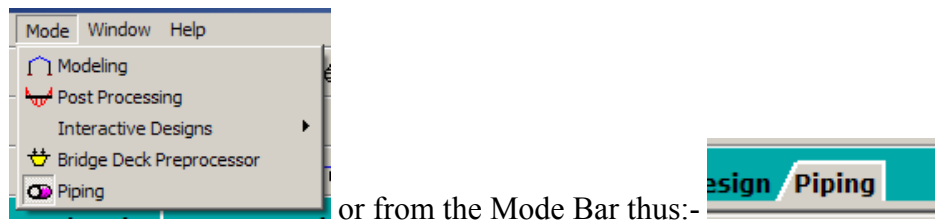


Figure 20

This displays the principal page and two sub pages available in this mode thus:-

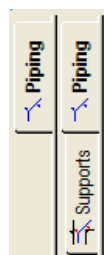
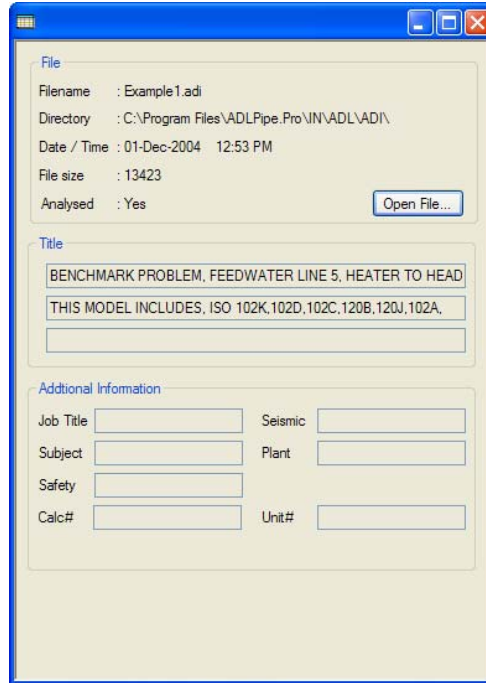


Figure 21

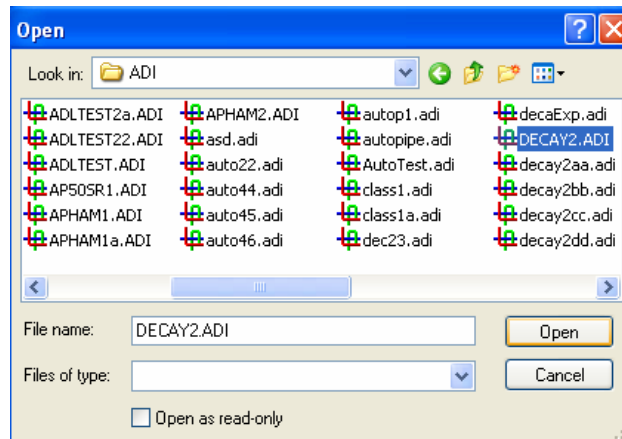
## 2) Open results of valid analysed ADLPIPE / AutoPipe model (\*.ADL) file:-

In the main Piping Sub-Page, the dialog on the right of the screen allows a suitable pipe model to be imported. Click on the **[Open File...]** button to browse for the required data file:-



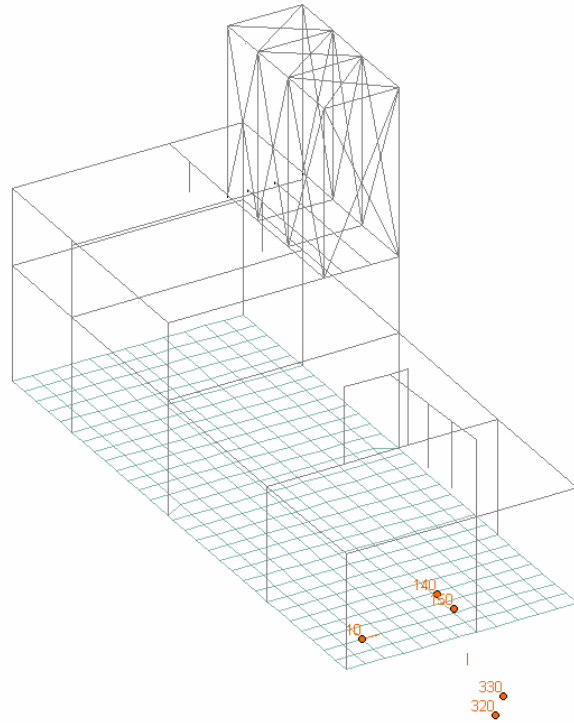
**Figure 22**

Note that typically for ADLPIPE this will reside in the **IN/ADL/ADI** folder of the installation directory of ADLPIPE



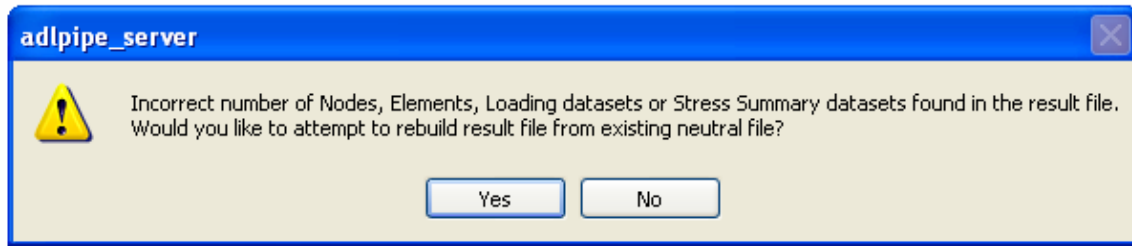
**Figure 23**

If the input file is not complete or contains errors, the complete model will not be displayed, however part of the model may be displayed.



**Figure 24**

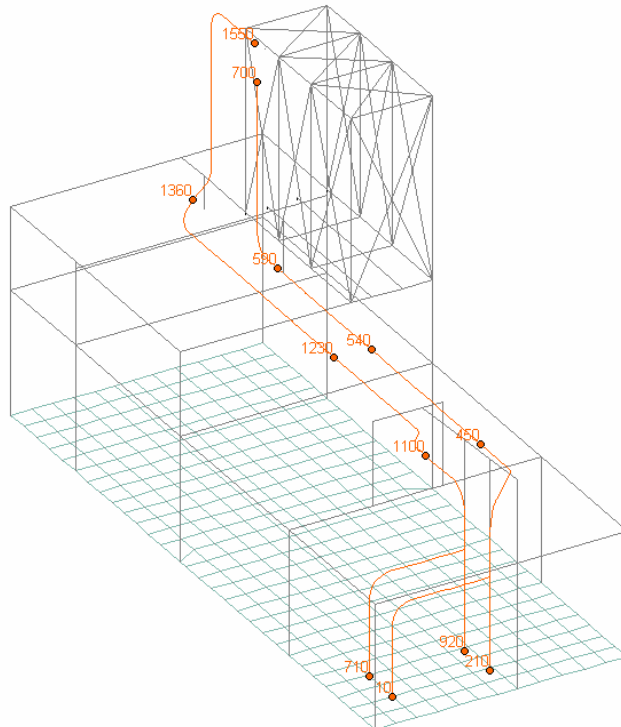
Additionally, if the model does not contain a valid set of analysis results the following error message will appear when trying to display the model:-



**Figure 25**

In such circumstances click [**No**] and return to the piping analysis to create a valid set of piping results.

However, if there is no error the piping model will be shown in STAAD.Pro along with the structure. The nodes on the piping system which have supports attached to them will be shown with their corresponding node numbers.

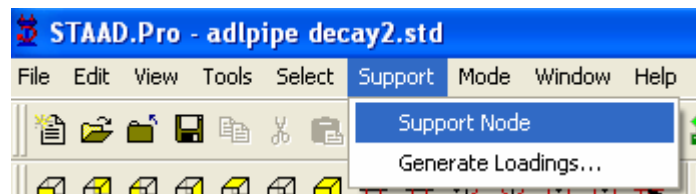


**Figure 26**


### 3) Identify the relationship between support nodes on the pipe model and the nodes on the STAAD.Pro structure:-

Enter the **Piping | Supports** sub-page which displays the Supported Nodes table on the right of the screen. This identifies the node number in the pipe model which can be associated with a node on the STAAD.Pro model by either entering the node number in the table or by using the GUI as described next.

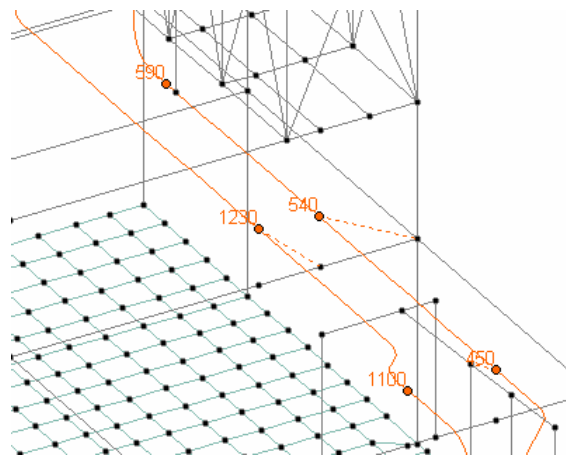
Click on the graphics view window to make it the active window. Then from the select the menu option, Support->Support Nodes:-



**Figure 27**

The cursor changes to , which indicates that it is updating the Supported Nodes table. As described on the status bar, first click on one of the Piping nodes, then on the node to which it is to associate its loading. Repeat the process for all the supported nodes and click on the [esc] button on the keyboard or select an alternative cursor to end the association process. Where there is a separation in 3D space between the location of the pipe node and the STAAD.Pro support node, the association is indicated on the model graphics view by a dotted line. Note how the table is updated with the Supported Type set to Node and the Structure Node set to the STAAD.Pro node number.

To remove an association, select the option 'Ground' from the Supported Type in the Supported Nodes table.



**Figure 28**

#### 4) Generating load on the structure:-

With the association in place, the loading can be transferred onto the STAAD.Pro model. Click on the menu option Support->Generate Loadings

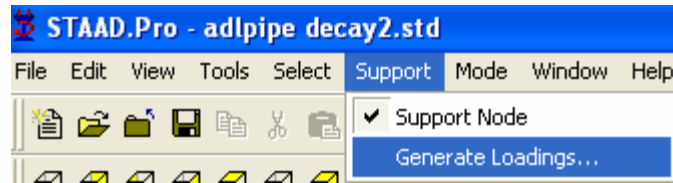


Figure 29

A dialog box is displayed which lists the load cases with reactions defined in the piping model which can be selectively transferred into STAAD.Pro load cases, by moving them into the right hand panel.

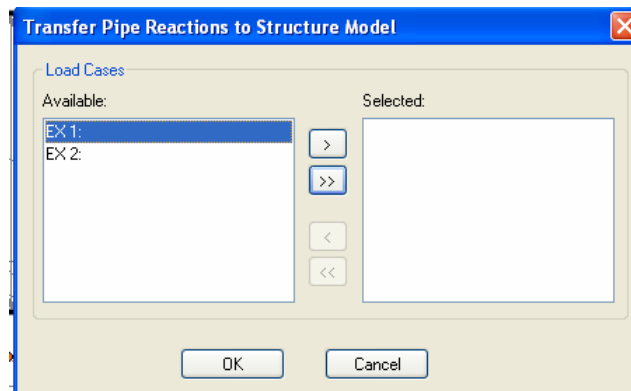


Figure 30

Once the required list of load cases have been defined, click on the [OK] button and that number of new load cases will be created in the STAAD.Pro input file which includes the direct forces and for nodes where the pipe model node and STAAD.Pro model node are separated, an additional moment in directions where there is a resultant lever arm.

The successful completion of the load case generation is displayed with this message box:-



Figure 31

Once complete, return to the Modelling Mode to perform the analysis and design checks required with this loading.

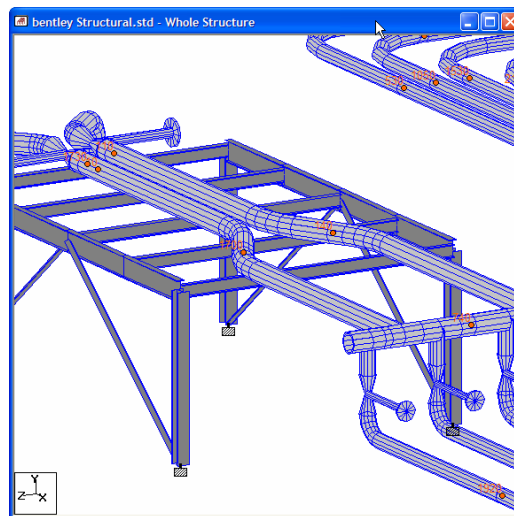
## AD.2006-1002.6.2 3D Rendered View of Piping Model

### Purpose

In order to perform quick, visual clash detections, the 3D properties of the piping model have been added to the 3D section view and Rendered 3D View.

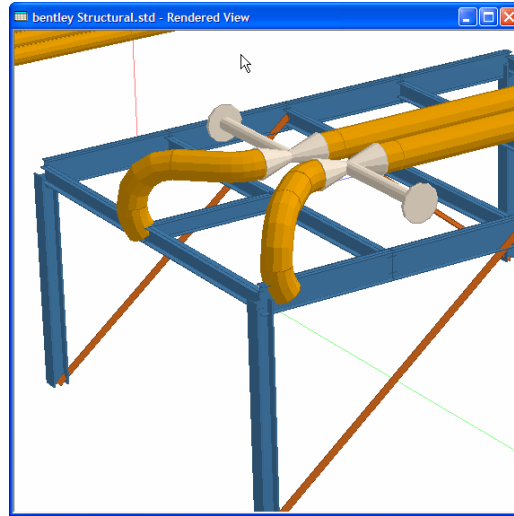
### Description

To view the solid 3D shapes in the graphical window, select the menu option, View > Structure Diagrams... This opens the Diagrams dialog in the Structures sheet. In the 3D Sections group, select the option 'Full Sections' to display a solid 3D shape of the structure model and piping arrangement, thus:-



**Figure 32**

To view a rendered 3D view of the structural model with the piping arrangement, select the menu option, View > 3D Rendering... This creates a new window and displays a 3D perspective view of the model and piping thus:-



**Figure 33**

Note that the piping model is only displayed in the Piping Mode.

## **AD.2006-1002.7 Additional STAAD.Pro 2006 Build 1001 Features**

The following pages outline the additional features that have been added to STAAD.Pro in this version.

## **AD.2006-1002.7.1 Modifications in REI Security System**

### **Purpose**

As part of the integration of the STAAD.Pro product range into the Bentley product range, the STAAD.Pro security system has been enhanced in order that it can be used by clients subscribing to the Bentley SELECT account licensing system.

### **Description**

For full details see the separate document STAAD.Pro Installation Guide or go to the web site <http://www.reiworld.com/Product/Pro/InstallationGuide.asp> .

## AD.2006-1002.7.2 Online Help

### Purpose

In order to assist the user operate STAAD.Pro, a number of context sensitive links have been added to the GUI such that clicking on a Help button or pressing the F1 key will go directly to an appropriate help topic within the online documentation.

### Description

Here are four examples of what the Help file will display when the either the Help button or F1 button on the keyboard is clicked when specific dialogs are open.

#### 1) Color Manager

The dialog is launched from the menu item:-

View -> Set Structure Colors...

The Help link is to the STAAD Graphical Environment Document:-

2.3.3.14 View | Set Structure Colors

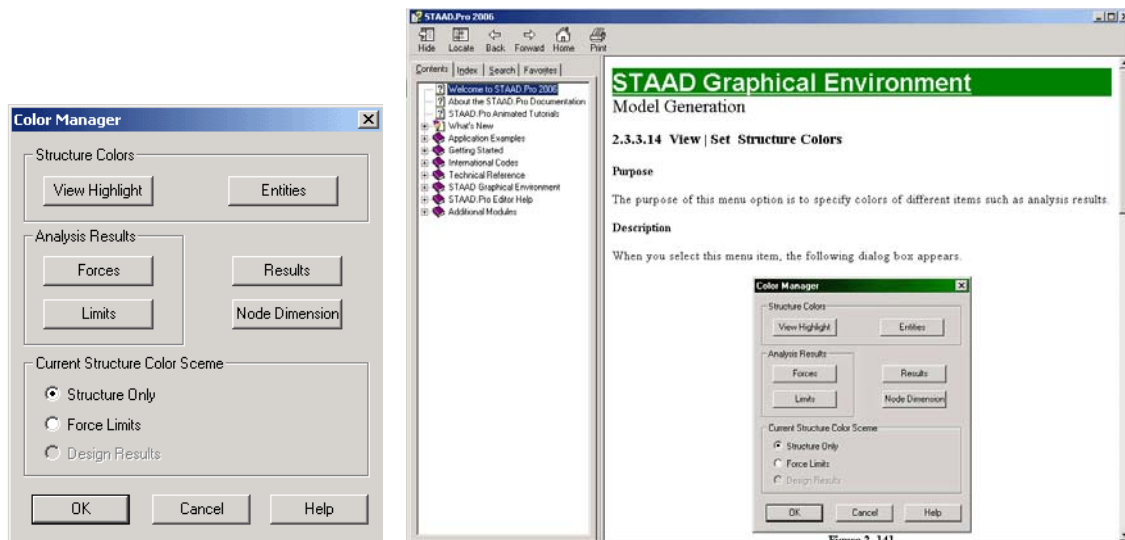


Figure 34

#### 2) Check Duplicate Nodes

The dialog is launched from the menu item:-

Tools -> Check Duplicate -> Nodes

The Help link is to the STAAD Graphical Environment

2.3.4 Tools Menu – Check Duplicate

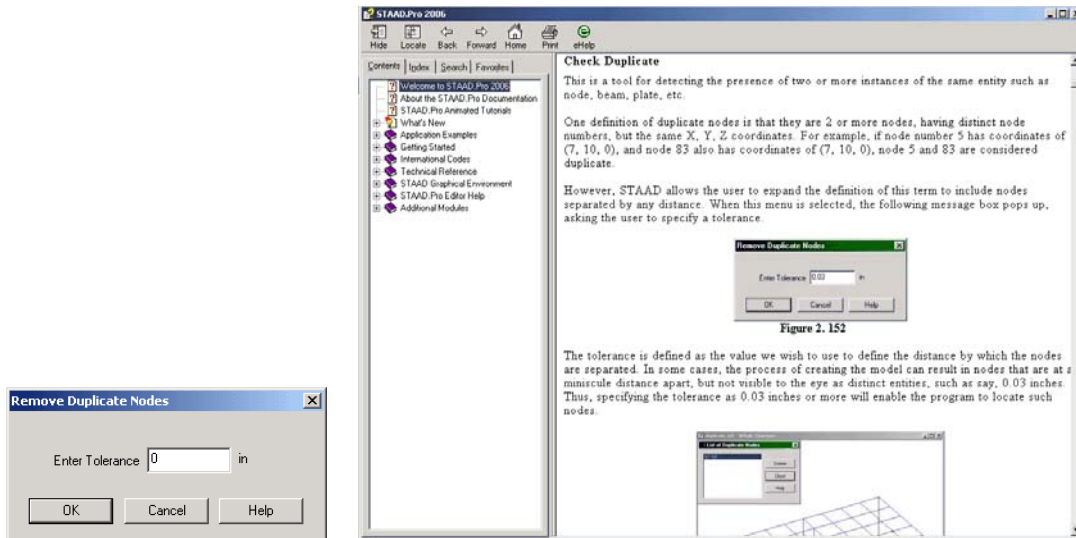


Figure 35

### 3) Circular Repeat

The dialog is launched from the menu item:-

Geometry -> Circular Repeat...

The Help link is to the STAAD Graphical Environment

2.3.6.13 Geometry | Circular Repeat

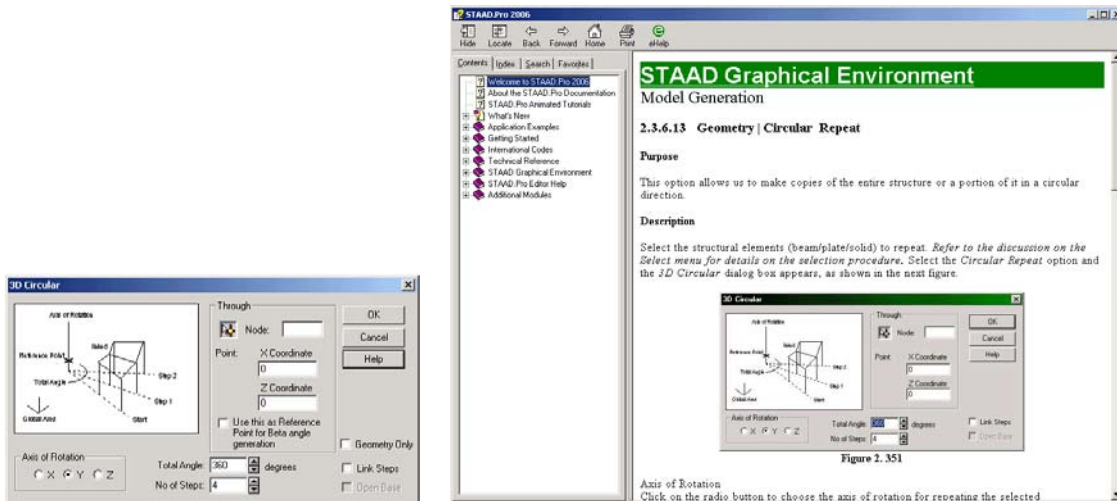


Figure 36

### 4) Beta Angles

The dialog is launched from the menu item:-

Commands -> Geometric Constants -> Beta Angle The Help link is to the STAAD Graphical Environment

2.3.7.5.1 Commands | Geometric Constants | Beta Angle

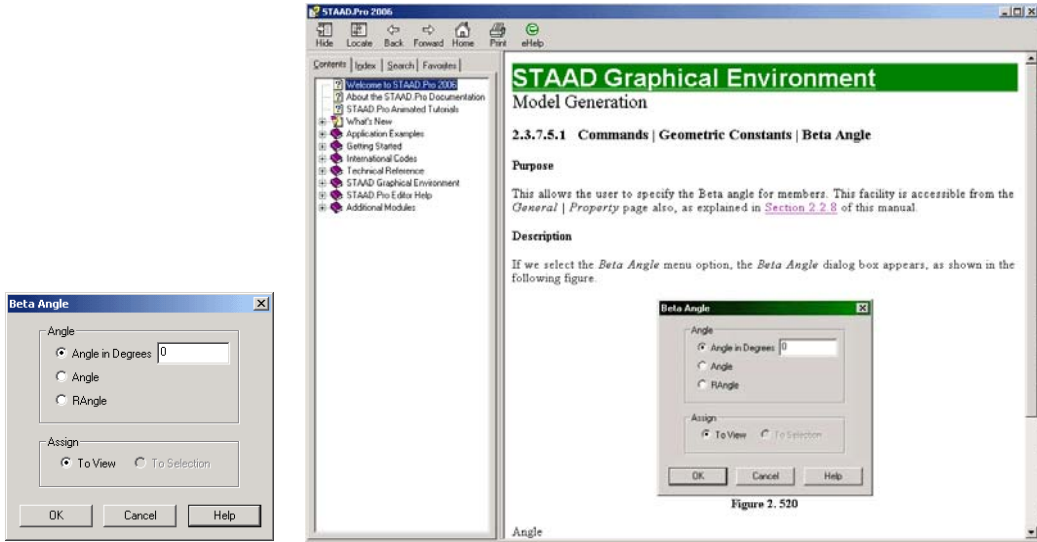


Figure 37