

---

# STAAD.Pro

---

V8i (SELECTseries 3)

***Release Report 20.07.08***

---



**DAA039020-1/0002**  
Last updated: 10 October 2011



# Copyright Information

## TRADEMARK NOTICE

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

## COPYRIGHT NOTICE

© 2011, Bentley Systems, Incorporated. All Rights Reserved.

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

## Acknowledgments

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

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

## RESTRICTED RIGHTS LEGENDS

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

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

Unpublished - rights reserved under the Copyright Laws of the United States and International treaties.

## END USER LICENSE AGREEMENT

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



# Table of Contents

---

<b>Section 1 STAAD.Pro V8i (SELECTseries 3)</b> .....	<b>v</b>
Introduction.....	v
AD.2007-08.1 Features Affecting the General Program.....	v
AD.2007-08.2 Features Affecting the Pre-Processor.....	x
AD.2007-08.3 Features Affecting the Analysis and Design Engine.....	xvi
AD.2007-08.4 Features Affecting Post Processing.....	xxii
<b>Section 2 STAAD.Pro V8i (SELECTseries 2)</b> .....	<b>1</b>
Introduction.....	1
AD.2007-07.1 Features Affecting the General Program.....	1
AD.2007-07.2 Features Affecting the Analysis and Design Engine.....	4
AD.2007-07.3 Features Affecting Post Processing.....	23
AD.2007-07.3.2 RC Designer.....	24
AD.2007-07.4 Additional Features.....	31
<b>Section 3 STAAD.Pro V8i (SELECTseries 1)</b> .....	<b>33</b>
Introduction.....	33
AD.2007-06.1 Features Affecting the General Program.....	33
AD.2007-06.2 Features Affecting the Analysis and Design Engine.....	37
AD.2007-06.3 Features Affecting the RAM Connection Design Mode.....	54
AD.2007-06.4 Features Affecting the Piping Mode.....	55
<b>Section 4 STAAD.Pro V8i</b> .....	<b>61</b>
Introduction.....	61
AD.2007-05.1 Features Affecting the Analysis and Design Engine.....	61
AD.2007-05.1.1 Geometric Nonlinear Analysis.....	62
AD.2007-05.1.2 IS 800:2007.....	65
AD.2007-05.1.3 Eurocode 3 Includes National Annex.....	70
AD.2007-05.1.4 Eurocode 8.....	72
AD.2007-05.1.5 AIJ Concrete Design Update.....	77
AD.2007-05.2 Features Affecting the Concrete Design Mode.....	77
AD.2007-05.2.1 RC Designer Member and Envelope Import.....	78
<b>Section 5 STAAD.Pro V8i(release 20.07.04)</b> .....	<b>79</b>

---

<b>Introduction</b> .....	<b>79</b>
<b>AD.V8i.o New Features Affecting the General Program</b> .....	<b>79</b>
<b>AD.V8i.o New Features Affecting the General Program</b> .....	<b>80</b>
<b>AD.V8i.o New Features Affecting the General Program</b> .....	<b>87</b>
<b>AD.V8i.1 Features Affecting the Pre-Processor (Modeling Mode)</b> .....	<b>91</b>
<b>AD.V8i.1 Features Affecting the Pre-Processor (Modeling Mode)</b> .....	<b>91</b>
<b>AD.V8i.1 Features Affecting the Pre-Processor (Modeling Mode)</b> .....	<b>92</b>
<b>AD.V8i.1 Features Affecting the Pre-Processor (Modeling Mode)</b> .....	<b>95</b>
<b>AD.V8i.1 Features Affecting the Pre-Processor (Modeling Mode)</b> .....	<b>97</b>
<b>AD.V8i.2 Features Affecting the Analysis and Design Engine</b> .....	<b>98</b>
<b>AD.V8i.2 Features Affecting the Analysis and Design Engine</b> .....	<b>98</b>
<b>AD.V8i.2 Features Affecting the Analysis and Design Engine</b> .....	<b>99</b>
<b>AD.V8i.2 Features Affecting the Analysis and Design Engine</b> .....	<b>102</b>
<b>AD.V8i.3 Features Affecting the Post-Processing (Results Mode)</b> .....	<b>103</b>
<b>AD.V8i.3 Features Affecting the Post-Processing (Results Mode)</b> .....	<b>103</b>
<b>AD.V8i.3 Features Affecting the Post-Processing (Results Mode)</b> .....	<b>104</b>
<b>Technical Support</b> .....	<b>109</b>
<b>Index</b> .....	<b>111</b>

## Section 1

# STAAD.Pro V8i (SELECTseries 3)

---

## Introduction

The Software Release Report for STAAD.Pro V8i SELECTseries 3 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i SELECTseries 2 (release 20.07.07). This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

## AD.2007-08.1 Features Affecting the General Program



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

### AD.2007-08.1.1 ISM Integration

STAAD.Pro is now capable of transferring data to and from Bentley's Integrated Structural Modeling technology by means of the StructLink utility.

---

## What is ISM?

Bentley's Integrated Structural Modeling (ISM) is a technology for sharing structural engineering project information among structural modeling, analysis, design, drafting and detailing applications. ISM is similar to Building Information Modeling (BIM), but focuses on the information that is important in the design, construction and modification of the load bearing components of buildings, bridges and other structures.

## Purpose

There are two related purposes for ISM:

1. The transfer of structural information between applications.
2. The coordination of structural information between applications.

To provide for the first purpose (transferring information), ISM provides a means of defining, storing, reading and querying ISM Repositories.

To provide for the second purpose (coordination of information), ISM additionally provides capabilities to detect differences between ISM Repositories and to selectively (based on user selection) update either an ISM Repository or an application's data to provide a user-controlled level of consistency between the two data sets.

## ISM and Application Data

ISM is not intended to store all of the information that all of its client applications contain. Rather, it is intended to store and communicate a *consensus* view of data that is *common* to two or more of its client applications, such as STAAD.Pro.

The client application continues to hold and maintain its own private copy of project data. Some of the application data will duplicate that of the associated ISM Repository. The application data may even conflict with that in the ISM Repository. The client application (or you as its user) may decide that a conflict gives the best data for client application's and ISM's different uses.

The program provided by ISM for accepting or rejecting model data changes is called Structural Synchronizer. Structural Synchronizer provides you with a powerful set of tools for moving data between the applications used in your workflow. Even relatively small structural models have enormous amounts of data and ISM allows you to re-use this data with ease. Care must be taken that only the data you want transferred between applications is sent, though.

When accepting changes made by client applications to an ISM Repository, some attention must be paid to changes are actually being made. A small change in a client application can have unintended repercussions if accepted. A repository is intended to represent the data that is *common* to the client applications. Some client application models will use only a subset of the repository data but changes made to them can affect the entire repository if these changes are accepted when a repository update action is performed.

Let's take a look at an extended example of a multi-story, concrete building with a post-tensioned slabs:

- The architect on the project sends a Revit® model of the building to the structural engineer. From Revit®, an ISM Repository is created. The lateral structure will be analyzed and designed using RAM Elements while RAM Concept will be used to design the slabs.
- The engineer imports the floor and roof slabs into multiple RAM Concept files (each file represents one slab). The 5000 psi (35 MPa) concrete mix used for the slabs at all levels is found to be inadequate for the roof slab. The roof slab is changed to use a 6000 psi (40 MPa) concrete mix.
- An update to the repository from RAM Concept is performed from the roof-level Concept file. ISM detects the addition of the 6000 psi (40 MPa) mix, the removal of the 5000 psi (35 MPa) mix, and the change in the setting of the roof slab material.
- An inspection of the change management Differencing Window will reveal the material properties of all of the slabs (on other levels) that used the 5000 psi (35 MPa) mix have been cleared. RAM Concept's deletion of the 5000 psi (35 MPa) mix is correct in the scope of the roof slab, but this is causing unexpected changes elsewhere.
- These were not the intended changes the engineer wished to make. To cause the correct changes to be made, the engineer must reject the deletion of the 5000 psi (35 MPa) mix and reject the material property changes in all of the slabs except the one at the roof level.

## Overview of Sync ISM tools

A client application can send structural data to and from an ISM Repository through a set of Sync ISM tools. These tools allow you to both create and update client application models as well as ISM repositories. Further, these flexible tools allow you to begin models and move data as your workflow dictates.


These tools are accessed by a sub-menu item found in ISM supported client applications.




When any of these tools are selected, the corresponding Windows Open or Save dialog box opens to select an ISM repository for use. The Change Management environment is used with the update tools to coordinate which changes are to be reflected in the models and repository.

You may review the history of changes made to an ISM Repository by using the View Repository History tool. This tool also allows you to undo any changes made to a repository file.

**Hint:** It is recommend that you save your client application file before and after performing any Sync ISM operation.

Sync ISM Tools in **File > ISM** submenu

If you need to ...	... use this tool	Description
Create a new ISM Repository from an existing client application model	 <b>Create to Repository</b>	Transfers the current model opened in client application and generates a new ISM Repository. This is the most common way in which an ISM Repository is initially

If you need to ...	... use this tool	Description
		created.
Create a new client application model from an existing ISM Repository	 <b>New from Repository</b>	Creates a new client application model from an existing ISM Repository. This is used to transfer model data into other tools used for your workflow.
Update an existing repository to reflect changes made in a client application model	 <b>Update to Repository</b>	Coordinates changes made to the model in the client application and coordinate some or all of those changes with an existing ISM Repository.
Update an existing client application model to reflect changes in an ISM Repository	 <b>Update from Repository</b>	Updates your client application's model with some or all of the changes which have been made to the ISM Repository.

## AD.2007-08.1.2 Export to SACS

A new macro is included with STAAD.Pro which is used to export the current STAAD.Pro model to a SACS model, which can be opened in the SACS system Interactive Modeling program.

(Structural Analysis Computer System) is a finite element structural analysis suite of programs for the offshore and civil engineering industries.

### Exporting a STAAD model to a SACS input file

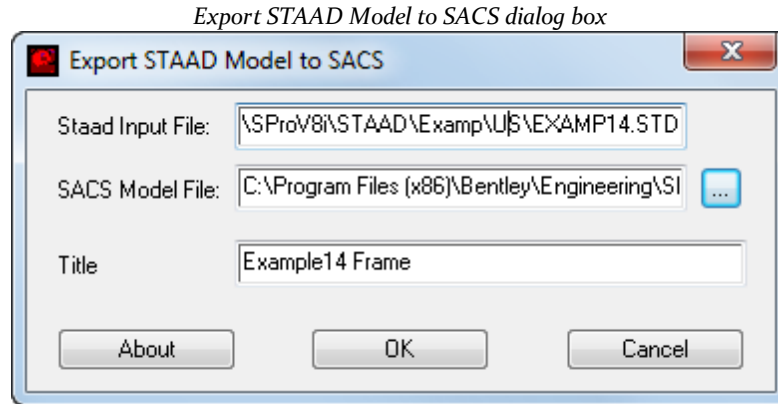
1. Open a input file in STAAD.Pro.
2. Select either

**Tools > User Tools > STAAD to SACS Export**

or

**STAAD to SACS Export** from the **User Tools** tool drop-down menu.

The *Export STAAD Model to SACS* dialog box opens.



3. Type a file path and file name for the **SACS Model file** (file extension .INP) to which you want to export the STAAD.Pro model data.

**Hint:** Click [...] to open a Windows *Save As...* dialog box.

4. (Optional) Type a **Title** to use for the model in SACS.
5. Click **OK**.

A confirmation dialog box opens with the status of the export.

## **AD.2007-08.1.5 European Cold Formed Sections per EN10219-2**

Sections defined in the publication, EN10219-2:1997 *Cold formed welded structural hollow sections of non-alloy and fine grained structural steel. Part 2: Tolerances, dimensions and sectional properties* have been added to library of cross sections available in the program.

The following section profiles are included as separate tables in the program:

- Table 6: Circular Hollow Sections (CHS)
- Table 7: Square Hollow Sections (SHS)
- Table 8: Rectangular Hollow Sections (RHS)

### **Specifying a European cold formed section**

1. Select the **General | Property** page.  
The *Properties - Whole Structure* dialog box opens.
2. In the *Properties - Whole Structure* dialog box, click **Section Database**.  
The *Section Profile Tables* dialog box opens.
3. Select the **Coldformed Steel** tab.
4. Select one of the tables available in the **European Cold Formed** section.
5. Select an entry from the **Select Profile** list.

---

6. Click **Add**.

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

## **AD.2007-08.1.6 Japanese JIS Hollow Sections**

The catalog of available Japanese hot rolled steel sections has been expanded to include the hollow sections from JIS publications.

- JIS G3444:2005 *Design Standard for Steel Structures - Based on Allowable Stress Concept* defines properties for circular hollow sections and are classed as 'General pipe sections'. The existing **PIPE** table has been updated to ensure that it is consistent with this table.
- JIS G3475:2005 *Design Standard for Steel Structures - Based on Allowable Stress Concept* defines properties for circular hollow sections and are classed as 'Architectural pipe sections'. A new table for these Circular Hollow (CHS) will be added.
- JIS G3466:2005 *Design Standard for Steel Structures - Based on Allowable Stress Concept* defines properties for square and rectangular hollow sections. These will be added to the existing Japanese Sections database as two new tables: Rectangular Hollow (RHS) and Square Hollow (SHS)

Refer to Section 12C.4 of the International Design Codes manual for additional details on using the Built-in Japanese Steel Section Library.

### **Specifying a Japanese hollow section**

1. Select the **General | Property** page.

The *Properties - Whole Structure* dialog box opens.

2. In the *Properties - Whole Structure* dialog box, click **Section Database**.

The *Section Profile Tables* dialog box opens.

3. Select the **Steel** tab.

4. Select one of the tables available in the **Japanese** section: Pipe, Rectangular Hollow, Square Hollow, or Circular Hollow.

5. Select an entry from the **Select Profile** list.

6. Click **Add**.

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

## **AD.2007-08.2 Features Affecting the Pre-Processor**

This section describes features that have been added that affect the pre-processor section of the program, also known as the Modeling Mode.

---

## AD.2007-08.2.1 Wind Load Generation per ASCE 7-10

Wind loads intensity values may now be automatically generated per the 2010 edition of SEI/ASCE 7 using the *ASCE-7: Wind Load* dialog in the user interface. This dialog allows you to generate a wind loading pattern based on the parameters used in this specification.

The primary change in wind load evaluation between the 2002 and 2010 editions of the ASCE 7 specification involve the method of calculating the force coefficient,  $C_f$  for solid freestanding walls and solid signs. The formula provided in the commentary related to Figure 6-20 in the specification is used:

$$C_f = \{1.563 + 0.008542 \ln(x) - 0.06148 \cdot y + 0.009011 [\ln(x)]^2 - 0.2603 \cdot y^2 - 0.08393 \cdot y \ln(x)\} / 0.85$$

Where:

$$x = B/s$$

$$y = s/h$$

### Generating a wind load intensity per ASCE 7-10

1. Add a wind load definition to the model.
2. Select this definition in the *Load & Definition* dialog box and click **Add**.  
The *Add New: Wind Definition* dialog box opens.
3. On the Intensity tab, select the as **Custom** in the drop-down list and click **Calculate as per ASCE-7**.  
The *ASCE-7: Wind Load* dialog box opens.
4. On the Common Data tab, select **2010** as the ASCE 7 - edition.
5. Specify or set the parameters needed to define the load.  
Click **Apply** prior to selecting a different dialog tab to update the dialog for the specified parameters.
6. Click **OK**.

## AD.2007-08.2.2 Single Mass Model

A new load type, mass, is available for reference load cases. This is used to create a single mass model for all dynamic loads (i.e., seismic, response spectrum, time history, etc.). This load case can be used for seismic loads in lieu of a weight table, reducing repetitive data entry for analysis methods which would require the same data.

Refer to Section 5.31.6 of the Technical Reference manual for additional details.

### Adding a mass model reference load

A mass model reference load is added in the same way as any other reference load, but with the Loading Type set to Mass.

- 
1. Either

select **Commands > Loads > Definitions > Reference Load**

or

select the **Definitions > Reference Load Definitions** section of the *Load & Definition* dialog box and click **Add**.

The *Add New: Reference Load Definition* dialog box opens.

2. (Optional) Type a reference load identification number in the Number field.

**Hint:** This number is incremented by one from any previously defined reference loads and typically does not need to be changed.

3. Select **Mass** as the **Loading Type**.

4. (Optional) Type a label in the **Title** field.

For example, you may want to label the reference by typing **Mass Model1**.

5. Click **Add**.

The dialog box closes and a new reference load definition is added to the input file.

### **Adding mass loads to the mass model reference load**

1. Select the mass Reference Load Case in the *Load & Definition* dialog box.

2. Click **Add**.

The *Add New: Reference Load Items* dialog box opens.

3. Select the type of load you want to add from the tree.
4. Specify load parameters (e.g., magnitude, direction, etc.).
5. Click **Add**.
6. Repeat steps 3 through 5 to add additional loads.
7. Click **Close**.

### **Using the mass model reference load in a seismic load**

1. Select a previously defined Seismic Definition in the *Load & Definition* dialog box.

2. Click **Add**.

The *Add New: Seismic Definitions* dialog box opens.

3. Select the Reference Load leaf on the left panel.
4. Select the Reference Load case in the **Available Load Cases** list and click **>**.
5. Click **Add**.
6. Click **Close**.

---

## AD.2007-08.2.1 Eurocode Load Combination Generator

A new macro has been included with the program to generate load combinations for the Strength limit state per *Eurocode – Basis of structural design, BS EN 1990:2002+A1:2005*.

The load combination generator is capable of creating load combinations per equations 6.10, 6.10a, or 6.10b found in Cl. 6.4.3.2.

These equations specify the following combinations of loads:

$$\sum_{j=1}^n \gamma_{G,j} G_{k,j} + \gamma_P P + \gamma_{Q,1} Q_{k,1} + \sum_{i>1}^n \gamma_{Q,i} \gamma_{o,i} Q_{k,i} \quad (6.10)$$

Alternatively, for the strength limit state, the less favorable of equations 6.10a and 6.10b may be used:

$$\sum_{j=1}^n \gamma_{G,j} G_{k,j} + \gamma_P P + \gamma_{Q,1} \gamma_{o,1} Q_{k,1} + \sum_{i>1}^n \gamma_{Q,i} \gamma_{o,i} Q_{k,i} \quad (6.10a)$$

$$\sum_{j=1}^n \gamma_{G,j} G_{k,j} + \gamma_P P + \gamma_{Q,1} Q_{k,1} + \sum_{i>1}^n \gamma_{Q,i} \gamma_{o,i} Q_{k,i} \quad (6.10b)$$

Where:

G<sub>k</sub> = Permanent actions

P = Prestress actions

Q<sub>k</sub> = Variable actions

**Note:** The effects in each of the above equations are always additive. If any effect is negative (that is, would reduce the final sum), its effect is taken as zero.

### Generating load combinations per Eurocode 0

1. Open a input file in STAAD.Pro.
2. Select either

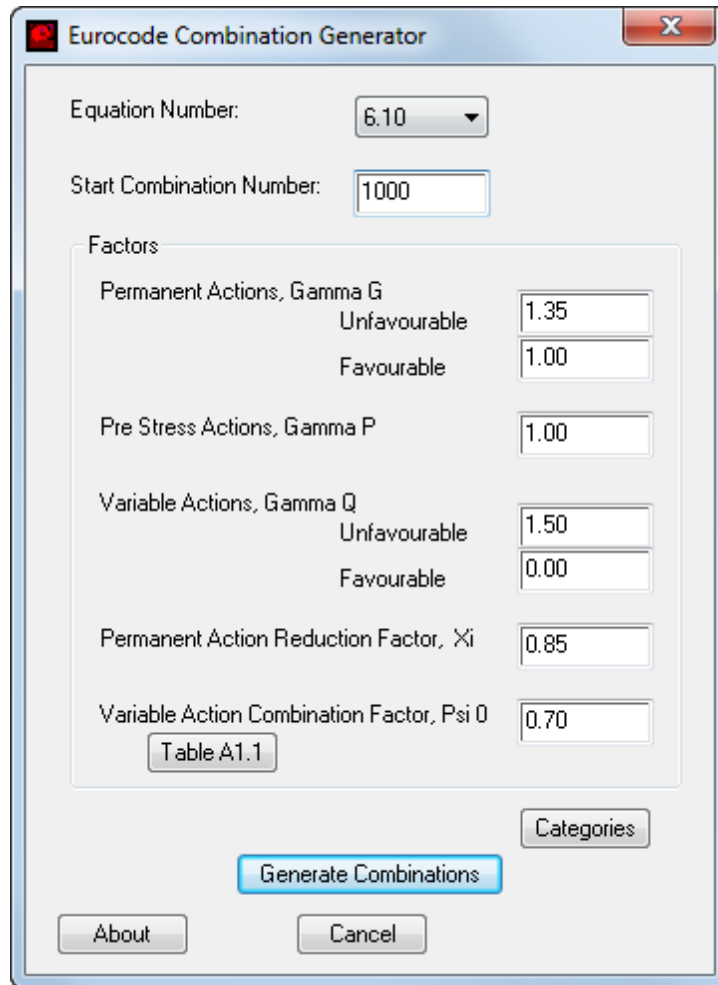
**Tools > User Tools > Euro Code Load Combination Generator**

or

**Euro Code Load Combination Generator** from the **User Tools** tool drop-down menu.

The *Eurocode Combination Generator* dialog box opens.

*Eurocode Combination Generator dialog box*



3. Select the **Equation Number** to use for the generation of load combinations.
4. (Optional) Specify a **Start Combination Number**.
5. (Optional) Specify load Factors for use in the combination equations.

Factor title	Equation notation
Permanent Actions, Gamma G	$\gamma_G$
Pre Stress Actions, Gamma P	$\gamma_P$
Variable Actions, Gamma Q	$\gamma_Q$
Permanent Action Reduction Factor, Xi	$\xi$
Variable Action Combination Factor, Psi o	$\psi_o$

**Note:** The default values are taken from those provided in ECo.

6. (Optional) Click **Table A1.1** to select one of the recommended values for the Permanent Action Reduction Factor, Psi ( $\psi_0$ ).
7. Click **Categories** to specify in which load action classification each STAAD.Pro load category is to be assigned.

Action	Included Loads
(None)	Loads of this type will <i>not</i> be included in generated load combinations.
Permanent	Gk (permanent) By default, Dead loads are included.  <b>Hint:</b> You may also want to include Mass or Gravity load.
Variable	Q (variable) By default, Live, Roof Live, Wind, and Snow are included.  <b>Hint:</b> You may also want to include loads such as Seismic, Temperature, etc.
Pre Stress	P (prestress) No loads are included by default in this category.  <b>Hint:</b> As STAAD.Pro does not use a load category for prestressing forces, a less-common load category such as <b>Imperfection</b> can be used for this action.

8. Click **Generate Combinations**.

A confirmation dialog box opens with the status of the export. Load cases are generated with the selection equation and load case number in the title.

## AD.2007-08.2.4 Rigid Floor Diaphragms

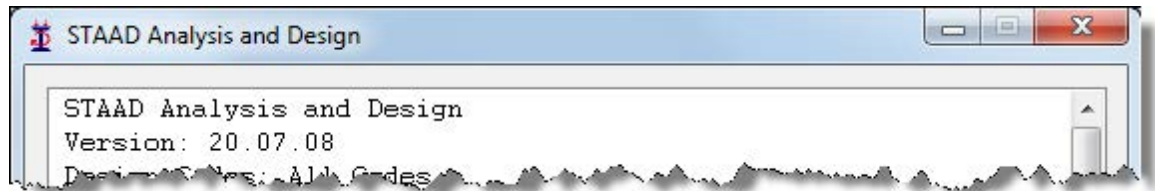
A new feature has been added to easily model a rigid floor diaphragm without the need to specify a master joint at each. When specified, this command directs the engine to perform the following:

- a. calculate the center of mass for each rigid diaphragm (where master joint is to be located) considering the mass model of the structure. The mass must be modeled using **the new mass reference load feature**.
- b. create, internally, an analytical node at the center of mass location to be included during analysis (unless a master node is specified) if an existing analytical node exists at this point, then the existing joint is used in lieu of creating a new joint.
- c. search all nodes available within a diaphragm and add them as slave nodes; with the master node located at the center of mass for the diaphragm (or at the specified master node)

Refer to Section 5.28.2 of the Technical Reference manual for additional details.

---

## AD.2007-08.3 Features Affecting the Analysis and Design Engine



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

### AD.2007-08.3.1 API 2A WSD 21st Ed. Update

Joint checking for tubular members per the American Petroleum Institute 2A-WSD code has been updated to the 21st Edition (December 2000) of that code, including errata and supplements 1 to 3 (latest: Supplement 3 - March 2008). Additionally, the process of joint design has been simplified.

**Note:** Only simple joints and overlapping joints will be considered by the program. Other type such as grouted joints, joints with ring stiffeners etc are not be considered.

The clauses/sections in the API code that have been dealt with are:

- 4.2.1 Material strength
- 4.2.3 Minimum Capacity
- 4.2.4 Joint Classification
- 4.3 Simple joints
- 4.4 Overlapping joints

Refer to Section 22 of the International Design Codes manual for additional information on the methodology used.

### External Joint Data File

In previous versions, the **LEG** design parameter was used to direct the program to generate a separate joint data file or to check a user-specified file. This file is now automatically generated or checked as needed.

As the API code allows for mixed joint types, the **PUNCH** column in the input file has been replaced with **K**, **X**, and **Y** columns which are used to designate fractional contributions of each joint class. Similarly, overlapping joints are indicated by using a negative value for the **GAP** between braces and the member number of the overlapping member in the **OB** column (replaces **THETAT** used in previous versions).

If the **FILENAME.PUN** file is not detected in the same folder as the current STAAD input file (where "FILENAME" is the same as the **.STD** file), then the program assumes this is the initial joint design and this file is created. If, however, this file is detected, the program assumes that the that the joint design has been performed at least once and will use this file to perform the joint checks.

---

## Checking tubular member joints per API

The checking of joints is an iterative process done by means of an automatically generated text file.

1. Model a structure as you normally would.

**Note:** Only circular pipe members are considered.

**Hint:** Using TRUSS specifications helps to reduce analysis time.

2. In the **Steel Design - Whole Structure** dialog box, select **API** as the design code.
3. (Optional) Specify the factor of safety used for joint checks using the new **FSJ** design parameter.
4. (Optional) Specify all other necessary design parameters.

The **LEG** parameter is no longer used to generate or check joint data files.

5. Specify **CODE CHECK** or **SELECT MEMBER** commands as needed and perform the analysis.  
If this the **FILENAME.PUN** file is not detected in the same folder as the input file, the program assumes this is the first time the structure is being analyzed and generates this input file with default data for the detected joints. Each joint is assumed to be a Y joint by default.
6. Modify the default joint data in the **FILENAME.PUN** as needed to describe the actual joint conditions.

A text editor can be used to make changes to this file. Be sure to save changes once complete.

7. Re-analyze the STAAD input file.  
Joint check results follow the steel design output.
8. Repeat steps 3 through 7 as needed to make changes in the structure.

## AD.2007-08.3.2 Shear Buckling per EC3

The design code checks performed per Eurocode 3 have been updated to include checks for shear buckling in I Sections and PFC Sections. Eurocode 3 – Part 1 (EN 1993-1-1:2005) states in C1.6.2.6 that the checks for Shear buckling are to be based on the procedure in EN 1993-1-5. STAAD.Pro performs checks based on the methods Section 5 of EN 1993-1-5:2006.

In the case of an unstiffened web, the program will check the unstiffened web capacity. If the demand due to applied loads is greater than this capacity, the program will calculate a suitable spacing for transverse stiffener plates in order to meet the demand.

In the case of a web with transverse stiffeners, the program will check the capacity of the web considering the provided stiffener spacing. The program will consider both the buckling capacity of the web as well as the flange. If the demand due to applied loads is greater than this capacity, the program will calculate a reduced stiffener spacing for transverse stiffener plates in order to meet the demand.

**Note:** Only transverse stiffeners are taken into account. The effect of any longitudinal stiffeners is ignored.

---

The distance between transverse stiffeners is specified by the **STIFF** parameter, in the current units of length. If no value is specified, the program assumes a spacing equal to either the member length or depth of beam, whichever is greater.

The output file provides recommendations on the evaluation of stiffeners (e.g., adding stiffeners at a spacing or increasing the web thickness).

Refer to Section 7C.5.3 of the International Design Codes manual for additional information.

### **AD.2007-08.3.3 IS800:2007 Working Stress Method**

The IS:800-2007 Steel code was deviated in concept from its -1984 version (based on Working Stress Method) and introduced the Limit State Method of Design. The entire 2007 version of the code is devoted to the Limit State Method of Design, except Chapter 11. This Chapter comprises of a couple of pages and has the guideline for the Design of Steel sections as per working stress method (WSM). The approach of this new working stress method is different from its earlier version and utilizes the concept of Section Slenderness and Section Classification.

At the time of first implementation of this code in STAAD.Pro, the working stress method was kept out-of-scope as the primary need of the industry was the Limit State Method. However, with the time, the demand of the working stress method gradually increases. Some of the design sectors, more specifically the PEB Industry, use WSM widely. To cater the needs of the industry, it was later decided by the Product Management Team to include WSM of IS:800-2007 in STAAD.Pro.

The program now includes design per the Working Stress Method (WSM) methodology in addition to Limit State method for design of steel structures per IS800:2007.

Some minor corrections to the Limit State Design option have also been made.

#### **Specifying a design using IS 800:2007 Working Stress Method**

1. In the Modeling mode, select the **Design | Steel** tab.  
The *Steel Design - Whole Structure* dialog box opens.
2. In the **Current Code** drop-down menu, select **IS800 2007 WSD**.
3. Click **Define Parameters...**  
The *Design Parameters* dialog box opens.
4. Click **Add**.

This will insert the following commands into the STAAD input file:

```
CODE IS800 WSD
```

### **AD.2007-08.3.4 Surface Element Selfweight**

A new surface load has been added to include the self weight of surface elements. This command can be used to calculate and include the weight of surface elements in the analysis of a structure. The

---

## Adding a surface selfweight load via the Graphical Interface

1. Select the **General | Load & Definition** page.

The *Load & Definition* dialog box opens.

2. Select the load case to which you wish to add a surface selfweight load by expanding the Load Case details entry.
3. Click **Add**.

The *Add New: Load Items* dialog box opens.

4. Select the **Surface Loads > Selfweight Load** tab.
5. (Optional) Select a **Direction**.

The global Y direction is the default.

6. (Optional) Type a load **Factor**.
7. Click **Add**.

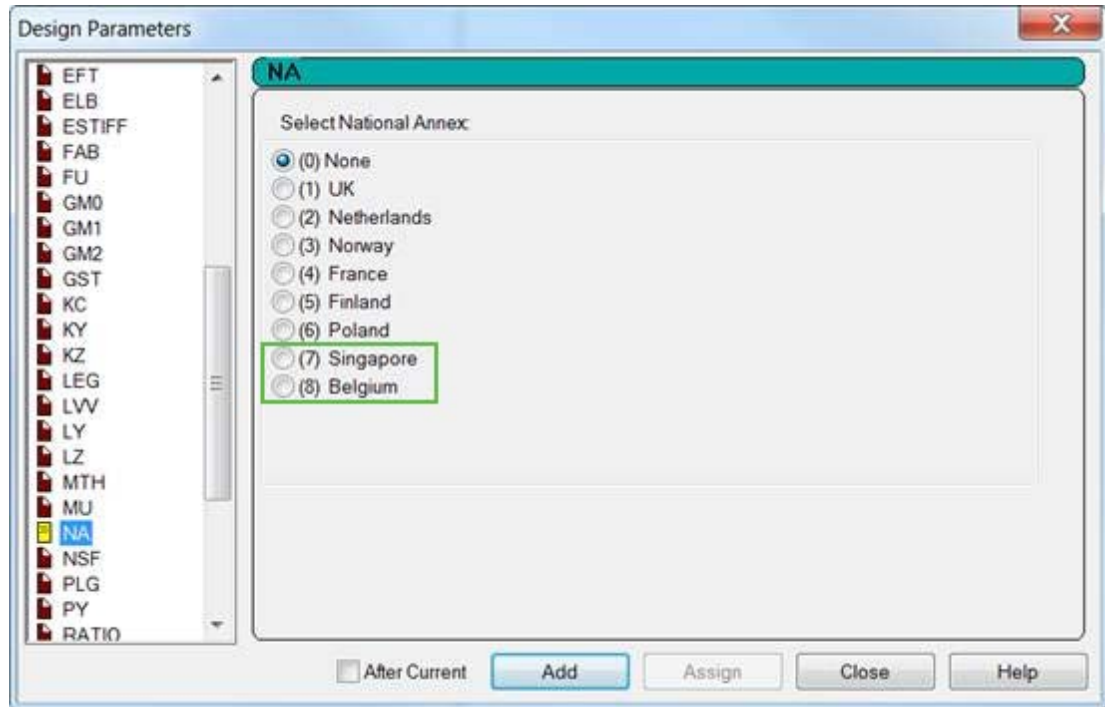
The **SSELFWT** command, along with direction and factor, are added to the load case.

8. Assign the load as you would any other load item.

Refer to Section 5.32.9.2 of the Technical Reference Manual for additional details on the **SSELFWEIGHT** command.

### AD.2008-06.2.3 Eurocode 3 National Annex

Two additional country's National Annex to Eurocode 3 have been incorporated into the Steel design module in STAAD.Pro: Singapore and Belgium. As with the other National Annexes to EC-3, this implementation will make use of the **NA** parameter.



#### AD.2007-08.3.5.1 Belgian National Annex to Eurocode 3 (EN 1993-1-1:2005)

The Belgian National Annex document referred to is “NBN EN 1993-1-1:2005”.

When the Belgian National Annex to EC<sub>3</sub> is used for design, the output section title is revised to include the Belgian National Annex (National Annex to NBN-EN 1993-1-1). Additionally, the partial safety factors used are included in the output and are as specified in the Belgian NA. The value for C<sub>1</sub> and k factors used in the calculation of the elastic critical moment are also included in the report.

**Note:** For additional information, please refer to Section 7D.10 and Section 7C. of the International Design Codes manual.

### Specifying a design using the Belgian NA to EC3

1. In the Modeling mode, select the **Design | Steel** tab.  
The *Steel Design - Whole Structure* dialog box opens.
2. In the **Current Code** drop-down menu, select **EN 1993-1-1:2005**.
3. Click **Define Parameters....**  
The *Design Parameters* dialog box opens.
4. Select the NA parameter in the list box.
5. Select the option for (8) Belgium.
6. Click **Add**.

This will insert the following commands into the STAAD input file:

---

**CODE EN 1993-1-1:2005**

**NA 8**

### **AD.2007-08.3.5.2 Singaporean National Annex to Eurocode 3 (EN 1993-1-1:2005)**

The Singaporean National Annex document referred to is “SS EN 1993-1-1:2005”.

When the Singaporean National Annex to EC3 is used for design, the output section title is revised to include the Singaporean National Annex (National Annex to SS-EN 1993-1-1). Additionally, the partial safety factors used are included in the output and are as specified in the Singaporean NA. The value for  $C_1$  and  $k$  factors used in the calculation of the elastic critical moment are also included in the report.

**Note:** For additional information, please refer to Section 7D.9 and Section 7C. of the International Design Codes manual.

## **Specifying a design using the Belgium NA to EC3**

1. In the Modeling mode, select the **Design | Steel** tab.  
The *Steel Design - Whole Structure* dialog box opens.
2. In the **Current Code** drop-down menu, select **EN 1993-1-1:2005**.
3. Click **Define Parameters...**  
The *Design Parameters* dialog box opens.
4. Select the NA parameter in the list box.
5. Select the option for (7) Singapore.
6. Click **Add**.

This will insert the following commands into the STAAD input file:

**CODE EN 1993-1-1:2005**

**NA 7**

## **AD.2007-08.3.6 EC3 Slender Circular Hollow Sections**

Slender circular hollow (pipe) sections may now be designed per Eurocode 3 (EN 1993-1-6:2007).

Eurocode 3 – Part 1 (EN 1993-1-1:2005)—hereafter EC3-6—states in Cl. 6.2.2.5 (5) that the design of slender circular hollow sections is to be based on the procedure in EN 1993-1-6. EC3-6 deals with the design of shell structures. EC3-6 does not, however, specify additional or modified safety factors. Therefore, the default safety factors from EN 1993-1-1 are used.

**Note:** You can change these values through the **GM0**, **GM1**, and **GM2** design parameters.

The program checks the plastic and buckling limit states for primary stresses based on the stress design method described in EC3-6.

---

Refer to Section 7C.5.6 in the International Design Codes manual for details on the methodology and calculations used in this design.

### AD.2007-08.3.7 User Defined Section for EC3

The feature to design user-provided table (UPT) general sections has now been introduced steel members designed per Eurocode 3 (EN 1993-1-1:2005). However, rather than assuming that the section will behave like an I section, you are given the option of choosing the 'section-type' he would like to design the member for.

This is achieved through the introduction of a new design parameter, **GST**, that has the following values:

- o. I-Section (Default)
  1. Single Channel
  2. Rectangular Hollow Section
  3. Circular Hollow Section
  4. Angle Section
  5. Tee Section

Unless specified using the **GST** parameter, a general section will be assumed to be an I- Section.

**Note:** This parameter will be ignored if assigned to any section other than a General Section.

The design procedure will then account for the section type and proceed with the design as necessary. The output report will also indicate the section type considered for the design of the UPT section. The design output will indicate the section as follows:

```
*      1 ST   IPE100      (UPT: DESIGNED AS I-SECTION)
                                FAIL      EC-6.3.2 LTB      8.591      1
                                0.00      0.00      -31.25      2.50
|-----|
| CALCULATED CAPACITIES FOR MEMB      1 UNIT - kN,m SECTION CLASS 1 |
| MCZ=  9.2 MCY=  2.1 PC=  12.3 PT=  242.1 MB=  3.6 PV=  68.7 |
| BUCKLING CO-EFFICIENTS C1 AND K : C1 = 1.132 K = 1.000 |
| PZ=  242.05 FX/PZ = 0.00 MRZ=  9.2 MRV=  2.1 |
|-----|
```

Refer to Section 7.C.6 of the International Design Codes manual for additional information on all EC3 design parameters.

## AD.2007-08.4 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the Post Processing modes. These are explained in the following pages.

### AD.2007-08.4.1 Eurocode 2:2004 Slab Design

The RC Designer module (Concrete Design mode) can now be used to design slabs per Eurocode 2 (EC2 EN 1992-1-1:2004).

Refer to the What's New section for the Concrete Mode for additional information.

## Section 2

# STAAD.Pro V8i (SELECTseries 2)

---

## Introduction

The Software Release Report for STAAD.Pro V8i (SELECTseries 2) contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i (SELECTseries 1) (release 20.07.06) This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

## AD.2007-07.1 Features Affecting the General Program



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

---

## AD.2007-07.1.1 Academic Licensing

In order to ensure that the next generation of engineer that emerges from the higher education system is up to speed using our applications, Bentley has a policy of providing software to Universities and Colleges at a favorable rate.

Students can now use STAAD.Pro under an Academic License, which is obtained through a SELECT account. Contact your regional engineer or visit [Bentley.com](http://Bentley.com) to obtain a license.

**Note:** When using an Academic License, the program window title bar and About window indicate this. Similarly, all output (Analysis files and Reports generated from STAAD.Pro) are marked as "Academic License User."

**Warning:** The Advanced Analysis Engine is not available when using the program under an Academic Licence.

## AD.2007-07.1.2 StructLink and PipeLink Plug-ins

Two plug-ins are available to you when installing STAAD.Pro V8i (SELECTseries 2): StructLink and PipeLink.

- StructLink is a utility used for the bi-direction exchange of data between STAAD.Pro and ProSteel V8i.
- PipeLink is an all new utility used for the exchange of pipe stress model data between STAAD.Pro and AutoPIPE V8i. Refer to [section AD.2007-07.3.4](#) for additional the bi-directional data exchange capabilities made available through this utility.

**Note:** During the installation of STAAD.Pro V8i, select the option to install additional programs and utilities in order to have these two utilities installed.

Refer to the documentation included with these plug-ins for additional information on their use.

## AD.2007-07.1.3 Structural Dashboard Integration

Bentley's Structural Dashboard is now integrated into STAAD.Pro V8i.

### Description

Bentley's Structural Dashboard V8i is a free utility application which allows you to manage workflows and project files as well as keep up to date with latest products, news, and Be Communities happenings. This program can now be accessed from within STAAD.Pro and will launch whenever STAAD.Pro is started to assist you in managing your entire project workflow.

When STAAD.Pro is first launched after installing Structural Dashboard, a welcome dialog opens to allow you to set the automatic launch option. To launch the program and continue allowing it to launch

---

whenever a Bentley Structural program starts, leave the option selected and click the **OK** button. Otherwise, you can de-select this option before proceeding.



### **Launch Structural Dashboard on program startup**

Set this option to open the Structural Dashboard application whenever STAAD.Pro is opened.

**Hint:** This setting can be changed at any time from within the Structural Dashboard program.

### **Show me this dialog on startup**

Set this option to display this dialog whenever STAAD.Pro is opened.

**Hint:** This setting can be changed at any time from the Configure Program dialog File Options tab.

### **> To launch Structural Dashboard from STAAD.Pro**

1. Select **File > Structural Dashboard...**

The Bentley Structural Dashboard V8i program opens.



**Note:** If Structural Dashboard has not been installed, this menu item is inactive. You can download the program from <http://www.bentley.com/en-US/Promo/ISM/downloads/>.

**Note:** Refer to Section 2.3.1 of the User Interface Manual for additional help in using the Structural Dashboard with STAAD.Pro.

## AD.2007-07.2 Features Affecting the Analysis and Design Engine



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

**Note:** Items labelled with an asterisk (\*) were added in the QA&R release of V8i (SELECTseries 2) (Build 20.07.07.31).

---

## AD.2007-07.2.1 Time History Spectrum Enhancements

### Purpose

New options have been added to the spectrum input for a Time History definition which allow you to output time history input data, a Response Spectrum for a Time History load, or to use frequency-spectrum pairs. These options can be added by modifying the input command file.

### Description

Two new output options are available for reporting time history input and synthetic time history ground acceleration data used by the program for a time history load with the spectrum generation option. You can control the amount of output generated (as this can be quite large) as well.

A new option has also been added to allow you to instruct the program to use frequency-spectra pairs in lieu of period-spectra pairs for the time history spectrum input.

#### > To output time history data

1. Create a structure with a time history definition using the Spectrum function option, either through the STAAD.Pro Editor or the Graphical Interface.
2. Select the **STAAD Editor** tool from the File toolbar.

or

Select **Edit > Edit Input Command File...**

The STAAD Editor window opens.

3. After the **DEFINE TIME HISTORY** command, in the **SPECTRUM** options, add the command **THPRINT f18**.

Where **f18** directs the program to output either the beginning and last 54 data points (**f18 = 1**) or the entire curve (**f18 = 2**) or a select number of beginning and last data points (**f18 ≥ 10**).

4. Save the command input file and exit the STAAD Editor window to return to the STAAD.Pro graphical interface.
5. Run the analysis as you normally would.

The Spectrum Input Parameters are included in the STAAD Output (*.ANL*) file along with the Time History Output (limited to the **f18** value specified).

#### > To generate spectrum output for a time history

1. Create a structure with a time history definition using the Spectrum function option, either through the STAAD.Pro Editor or the Graphical Interface.
2. Select the **STAAD Editor** tool from the File toolbar.

or

Select **Edit > Edit Input Command File...**

---

The STAAD Editor window opens.

3. After the **DEFINE TIME HISTORY** command, in the **SPECTRUM** options, add the command **SPRINT f19**.

Where *f19* represents an integer value after the **SPRINT** command to instruct the program to only output the beginning and last number of values equal to this integer.

4. Save the command input file and exit the STAAD Editor window to return to the STAAD.Pro graphical interface.
5. Run the analysis as you normally would.

A summary of the Spectrum input and the curve points are included in the STAAD Output (*.ANL*) file.

#### > To use the Frequency-Spectra pairs in a Time History load

1. Create a structure with a time history definition using the Spectrum function option, either through the STAAD.Pro Editor or the Graphical Interface.
2. Select the **STAAD Editor** tool from the File toolbar.

or

Select **Edit > Edit Input Command File...**

The STAAD Editor window opens.

3. After the **DEFINE TIME HISTORY** command, in the **SPECTRUM** options, add the command **FREQ**.
4. Save the command input file and exit the STAAD Editor window to return to the STAAD.Pro graphical interface.
5. Run the analysis as you normally would.

**Note:** Refer to Section 5.31.4 of the Technical Reference Manual for additional information on using the updated *options-spec* for the Spectrum input option.

## AD.2007-07.2.2 Response Spectrum Signed Results and IMR Load Cases

### Purpose

Two methods to produce signed response spectrum results have been added to the STAAD.Pro analysis engine. The Dominant and Sign commands may be used in the input file to produce signed output. Additionally, STAAD.Pro now includes an option to automatically generate new load cases based on a specified number of modes from the response spectrum.

### Signed Results

STAAD.Pro can now assign a mathematical sign (positive or negative) to the modal results by one of two means. The first method allows you to select a DOMINANT mode, the sign of which will then be applied to all other modes. The second method will produce signed values for all results. The sum of squares of

---

positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign.

## Individual Modal Response Case Generation

The Individual Modal Response (IMR) load cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere. You can use subsequent load cases with Repeat Load combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). You can also use the Repeat Load capability to combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

The modal accelerations are multiplied by the nodal masses to produce equivalent static lateral forces for each modal load case.

**Note:** When the IMR option is entered for a Spectrum case, then a Perform Analysis & Change must be entered after each such Spectrum case.

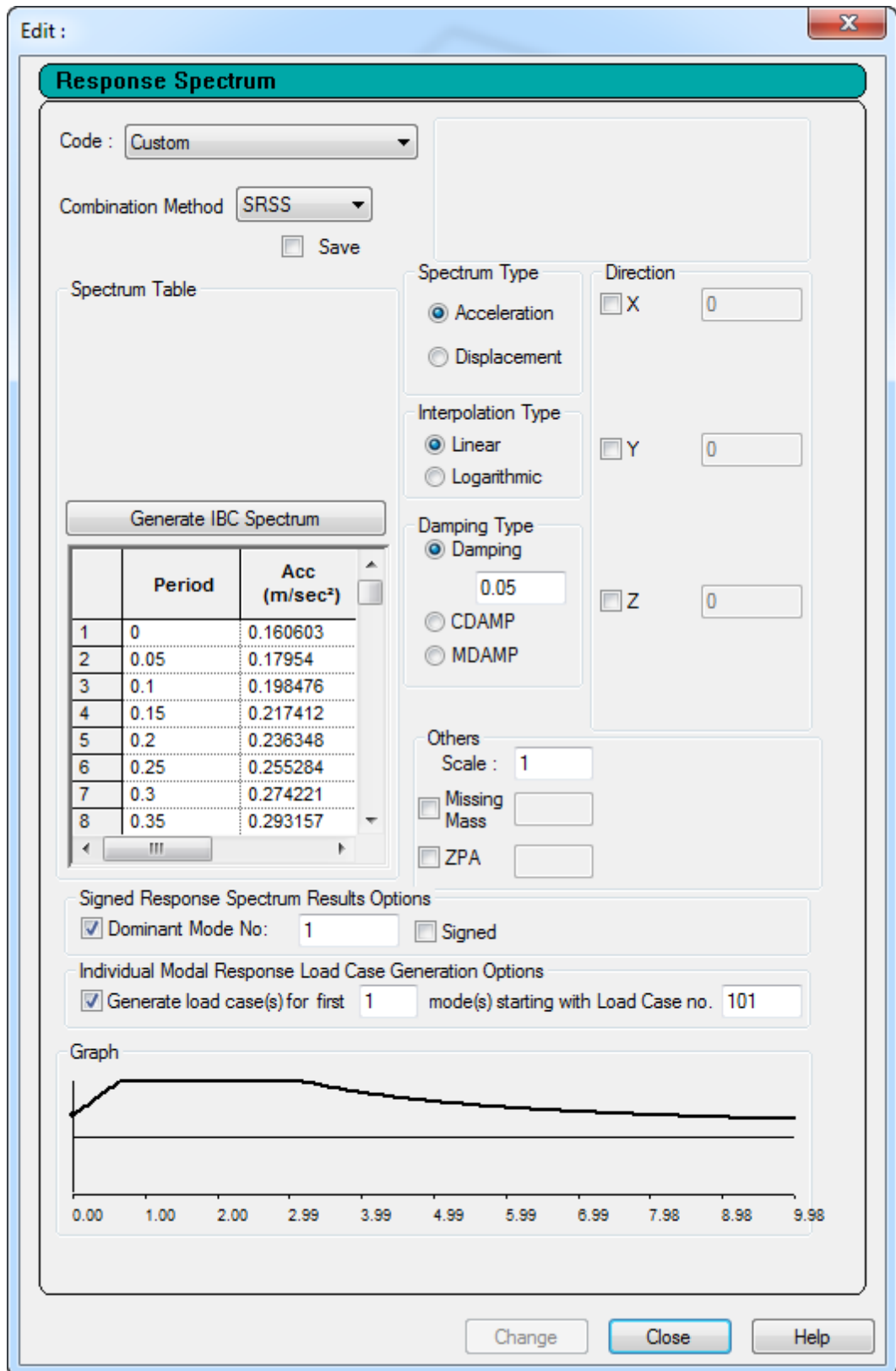


Figure - Updated Response Spectra dialog containing controls for generating signed results and IMR load cases

---

### > To add Signed results to a Response Spectrum

1. Select **Commands > Loading > Load Commands**.

or

Select the General | Load & Definition page and then click the **New...** button.

The Create New Load Items dialog opens.

2. Select the Response Spectra tab.
3. Select the Code you wish to use.

**Note:** See below for using IMR generation options. All other parameters are same as previous versions.

4. Select the option to use the Dominant Mode No. to assign the same sign as the selected mode to all modes.
5. (Optional) Select to provide Signed results to

**Note:** Selecting this option will not use the Dominant Mode No., but rather will create signed values for all results by comparing the sum of the squares values for positive and negative values to determine the governing sign.

6. Click the **Add** button to add this response spectrum load.

### > To add Individual Modal Response results to a Response Spectrum

1. Select **Commands > Loading > Load Commands**.

or

Select the General | Load & Definition page and then click the **New...** button.

The Create New Load Items dialog opens.

2. Select the Response Spectra tab.
3. Select the Code you wish to use.

**Note:** See above for options to add signed results. All other parameters are same as previous versions.

**Note:** The Individual Modal Response case generation is not available for SNIp II code response spectra.

4. Select the option to Generate load cases for ... to individual modal response load cases.
5. (Optional) Specify the number of modes for which load cases will be generated.

---

**Note:** Selecting this option will not use the Dominant Mode No., but rather will create signed values for all results by comparing the sum of the squares values for positive and negative values to determine the governing sign.

6. (Optional) Specify a beginning load case number for the first primary load case generated from the IMR.
7. Click the **Add** button to add this response spectrum load.

Refer to section 5.32.10.1 of the Technical Reference Manual for additional information on the new parameters available to the SPECTRUM commands.

### **AD.2007-07.2.3 Design of Class 4 Steel Sections per S16-01**

#### **Purpose**

An update to the Canadian Steel Design code has been added for the design of Class 4 (slender) steel sections per CAN/CSA-S16-01. Previous versions of STAAD.Pro were capable of designing Section Classes 1, 2, or 3.

#### **Description**

The design of slender Class 4 steel sections does not require any different actions or input. The analysis engine will determine if a section meets the criteria for a Class 4 section and then perform the necessary checks, if design checks have been requested for that member.

#### **Methodology**

Refer to Section 3B.6 "Member Resistances" of the International Codes Manual for a detailed description of the methodology used in STAAD.Pro for performing the design of Class 4 sections per S16-01. A verification problem using this feature has also been added to Section 3B.10.

### **AD.2007-07.2.4 Von Mises Stresses per AIJ 2002 and 2005**

#### **Purpose**

Design per AIJ (Japanese) steel design codes has been updated to include checking members in accordance with Von Mises stress criteria in AIJ 2005. This check is a requirement for the design of steel structures in nuclear power plants in Japan.

#### **Description**

The von Mises stress equation is calculated when the new **MISES** parameter has been set to a value of 1 (the default value of 0 does *not* check this condition). The calculated forces and moments are combined per the von Mises stress criteria.

---

> **To specify a von Mises stress check in an AIJ 2002 or AIJ 2005 design**

1. Create a model with steel members.
2. Select either **AIJ 2002** or **AIJ 2005** for the Current Code on the **Design | Steel** page.
3. Click the **Define Parameters...** button.  
The Design Parameters dialog opens.
4. Select the **MISES** tab in the parameters list.
5. Select option 1 to instruct STAAD.Pro to perform the von Mises stress check as part of the steel design.
6. Click the **Add** button to add this parameter.
7. Close the Design Parameters dialog.
8. Assign the **MISES** parameter to members as needed, just as you would any other design parameter.

How the von Mises check results are included in the output depends on the level of detail (**TRACK** parameter) selected:

Track 0 or 1	The von Mises stress is reported if this ratio is the critical condition.
Track 2	The value for $f_m = \sqrt{(\sigma_x^2 + 3 \tau_{xy}^2)}$ (numerator in the von Mises stress ratio equation) is displayed in the Stresses output category. When the von Mises check ratio is the critical condition, the value of the ratio is reported.
Track 4	Used for deflection checks only. Von Mises checks are not reported.

## Methodology

Refer to section 10B.10(A) or 10B.4(B) "Von Mises Stresses Check" of the International Codes Manual for a detailed description of the methodology used in STAAD.Pro for performing von Mises stress checks per AIJ 2002 or 2005.

## AD.2007-07.2.5 NORSOK N-004 Tubular Steel Design

### Purpose

The design of tubular steel (European round pipe sections) members per NORSOK N-004 Rev 2, October 2004 has been included in STAAD.Pro. The program will perform the member design for ultimate limit states (and optional deflection checks for serviceability). The tubular joints can also be automatically generated and checked per the code.

---

## Description

The NORSOK code has been added to the steel design code list available in STAAD.Pro. Selecting this code allows you to assign parameters, including defining the water level above the origin (for calculating hydrostatic pressure) or the

**Note:** N-004 refers to the superseded version of Eurocode 3 (DD ENV 1993-1-1) in several places. In such cases, the corresponding clause from the latest version of EC-3 (EN 1993-1-1:2005) has been used in the STAAD.Pro implementation.

### > To perform a member design per the NORSOK N-004 code

1. Create a model with steel tubular members.

**Warning:** The Norsok code only supports pipe sections. Errors will be presented in sections other than pipe members are used.

2. Select **NORSOK** for the Current Code on the **Design | Steel** page.
3. Click the **Define Parameters...** button.  
The Design Parameters dialog opens.
4. Specify parameters as required.

**Note:** The height of water level above the origin is specified using the **HYD** parameter. Alternatively, the **PSD** parameter may be used to define the water pressure.

5. Close the Design Parameters dialog.
6. Assign the torsion-related parameters to members as needed, just as you would any other design parameter.

### > To perform a joint check per the NORSOK N-004 code

1. Add the **CHECK JOINT** command to a new **PARAMETER** manually in the STAAD.Pro input file using the Editor.
2. Perform an preliminary design by selecting **Analyze > Run Analysis...**  
The program creates an external text file titled *FILENAME\_JOINTS.NGO* which contains the automatically generated chord and brace definitions associated with the nodes included in the **CHECK JOINT** command. All joints are classified as Y by default.
3. Open the text file using a text editor program (i.e., Notepad or STAAD Editor).
4. Manually edit the joint classifications as needed.
5. (Optional) Edit the Brace and Chord definitions as needed.

---

**Note:** The Brace and Chord members at each joint are assumed based on the relative cross section dimensions. Lengths of Chord and Brace members are taken as the analytical beam member length.

6. Save the text file and the re-analyze the structure

## Methodology

Refer to section 19B of the International Codes Manual for a detailed description of the methodology used in STAAD.Pro for performing steel tube member design per NORSOK N-004.

## AD.2007-07.2.6 EC3 Torsion Design

### Purpose

Design per EC3 [EN 1993-1-1:2005] has been enhanced to include the design of members subject to torsion. You may select to have the program execute basic or detailed torsion stress checks. Torsion design checks can be performed on I-sections, H-Sections, Channel sections, and structural hollow sections (RHS, SHS, CHS).

**Note:** The default behavior is to neglect torsion. The new TORSION parameter must be set to either 1 (basic) or 2 (detailed) to perform torsion design.

### > To include torsion design for EC3 steel design members

1. Create a model with steel members.
2. Select EN 1993-1-1:2005 for the Current Code on the **Design | Steel** page.
3. Click the **Define Parameters...** button.  
The Design Parameters dialog opens.
4. Select the TOR(sion) tab in the parameters list.
5. Select either option 1 (von Mises check excluding warping effects) or option 2 (detailed checks including warping effects) to include design for torsion and click the **Add** button to add this parameter.
6. Specify the loading and support conditions of members subject to torsion using the CMT tab in the parameters list and click the **Add** button to add this parameter.
7. (Optional) For the cases of a concentrated torque (CMT = 2,3, or 6) somewhere along the member length (other the default of mid-span), specify the location of the torque using the ALH tab in the parameters list and click the **Add** button to add this parameter.
8. (Optional) Specify the effective length of members for torsion using the EFT tab in the parameters list and click the **Add** button to add this parameter.
9. Close the Design Parameters dialog.

- 
10. Assign the torsion-related parameters to members as needed, just as you would any other design parameter.

## Description

Torsion design in EC3 is given in Cl. 6.2.7 of EN 1993-1-1:2005. Therefore, this clause is used primarily for this implementation.

EN 1993-1-1:2005 does not deal with members subject to the combined effects of torsion and lateral torsional buckling. However, EN 1993-1-6 considers such a condition in Appendix A. Therefore, STAAD.pro uses Appendix A of EN 1993-1-6 to check for members subject to combined torsion and LTB.

The following clauses from EC3 are then considered:

- Cl. 6.2.7(1)
- Cl. 6.2.7(9)
- Cl. 6.2.7(5)
- EC-3 -6 App A

When torsion design is included (TOR = 1 or 2), then the EC3 design output includes the following sections:

- Basic (TORSION = 1) - The ratio calculated for stress interaction per EC-6.2.7(5) is displayed for each load case, along with the calculated values of axial force, shear in Y and Z, Bending about Y and Z, and torsion.
- Detailed (TORSION = 2) - The additional clauses viz. 6.2.7(1), 6.2.7(9) and EC3-6 A-1 will be included in the output. The stress interaction ratio per each is displayed for each load case, along with the calculated force and moment values used. Additional torsion calculation details are provided as well.

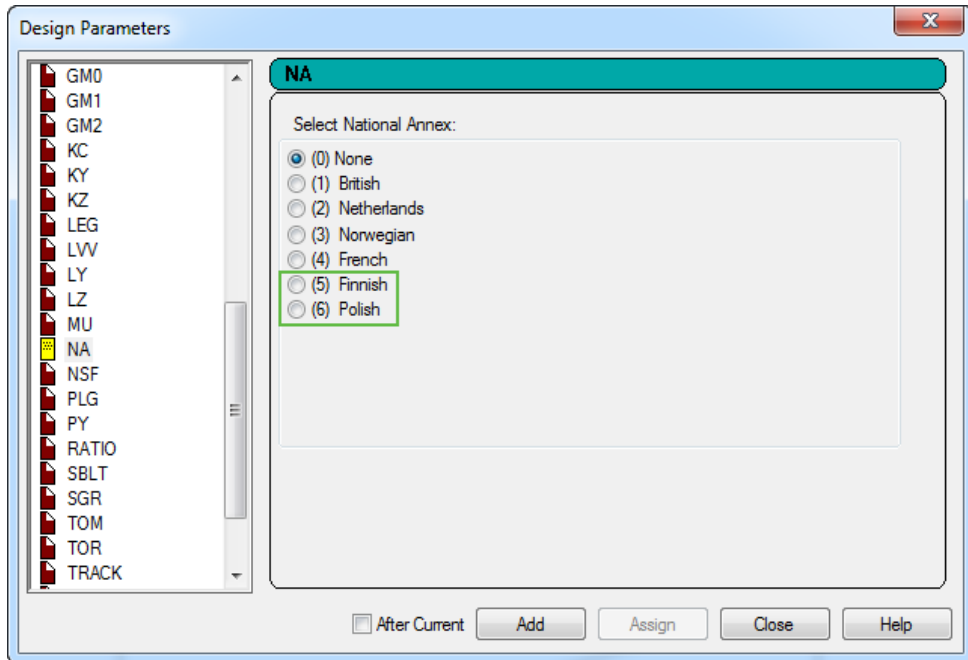
**Note:** If Torsion design is selected for a member which does not have any torsional moment, a warning is displayed in the output.

## Methodology

Refer to section 5B.5(B).4 "Design of Torsionally Loaded Members" of the International Codes Manual for a detailed description of the methodology used in STAAD.Pro for performing torsion stress checks per EC3.

### AD.2007-06.2.3 Eurocode 3 National Annex

Two additional country's National Annex to Eurocode 3 have been incorporated into the Steel design module in STAAD.Pro: Finland and Poland. As with the other National Annexes to EC-3, this implementation will make use of the **NA** parameter.



### AD.2007-06.2.3.2 Finnish National Annex to Eurocode 3 (EN 1993-1-1:2005)

The Finnish National Annex document referred to is "National Annex to Standard SFS-EN 1993-1-1".

#### > To Initiate a EC3-Finnish NA Steel Design

1. In the Modeling mode, click the **Design > Steel** tab.
2. In the Current Code drop-down menu, select **EN 1993-1-1:2005**.
3. Click the **Define Parameters...** button.

The Design Parameters dialog opens.

4. Select the NA parameter in the list box.
5. Select the option for (5) Finland.
6. Click the **Add** button.

This will insert the following commands into the STAAD input file:

```
CODE EN 1993-1-1:2005
NA 5
```

**Note:** For additional information, please refer to the International Design Codes manual, sections 5B.(B) "Steel Design to Eurocode 3" and 5B.(C) "EC3 National Annexes."

When the Finnish National Annex to EC3 is used for design, the output section title is revised to include the Finnish National Annex (National Annex to SFS-EN 1993-1-1). Additionally, the partial safety factors used are included in the output and are as specified in the Finnish NA. The value for C1 and k factors used in the calculation of the elastic critical moment are also included in the report.

---

## AD.2007-06.2.3.2 Polish National Annex to Eurocode 3 (EN 1993-1-1:2005)

The Polish National Annex document referred to is "National Annex to Standard PN-EN 1993-1-1".

### > To Initiate a EC3-Polish NA Steel Design

1. In the Modeling mode, click the **Design > Steel tab**.
2. In the Current Code drop-down menu, select **EN 1993-1-1:2005**.
3. Click the **Define Parameters...** button.  
The Design Parameters dialog opens.
4. Select the NA parameter in the list box.
5. Select the option for (6) Poland.
6. Select the new PLG parameter in the list box.

**Note:** This parameter is used to select if additional checks per clause 6.3.3 will be performed for designs using the Polish National Annex.

7. Click the **Add** button.

This will insert the following commands into the STAAD input file:

```
CODE EN 1993-1-1:2005
NA 6
```

**Note:** For additional information, please refer to the International Design Codes manual, sections 5B.(B) "Steel Design to Eurocode 3" and 5B.(C) "EC3 National Annexes."

When the Polish National Annex to EC3 is used for design, the output section title is revised to include the Polish National Annex (National Annex to PN-EN 1993-1-1). Additionally, the partial safety factors used are included in the output and are as specified in the Polish NA. The value for  $C_1$  and  $k$  factors used in the calculation of the elastic critical moment are also included in the report.

## AD.2007-07.2.8 AS4100 Physical Member Design

### Purpose

The workflow for design steel members per AS 4100:1998 has been updated to incorporate the use of physical members. Physical members are groups composed of a series of analytical beam elements of the same section and which are colinear (analytical beams are the beams used in modeling in STAAD.Pro). Using physical beams allows you to design for the actual conditions of the structure and assign specifications based on the true conditions of a steel member.

Some of the physical member design updates apply to design codes other than AS 4100, such as checks in the STAAD.Pro analysis engine for physical member overlapping and colinearity. These checks were

---

previously made in the graphical interface but now they will be checked again in the engine in the event you have manually generated the STAAD.Pro input file.

## Description

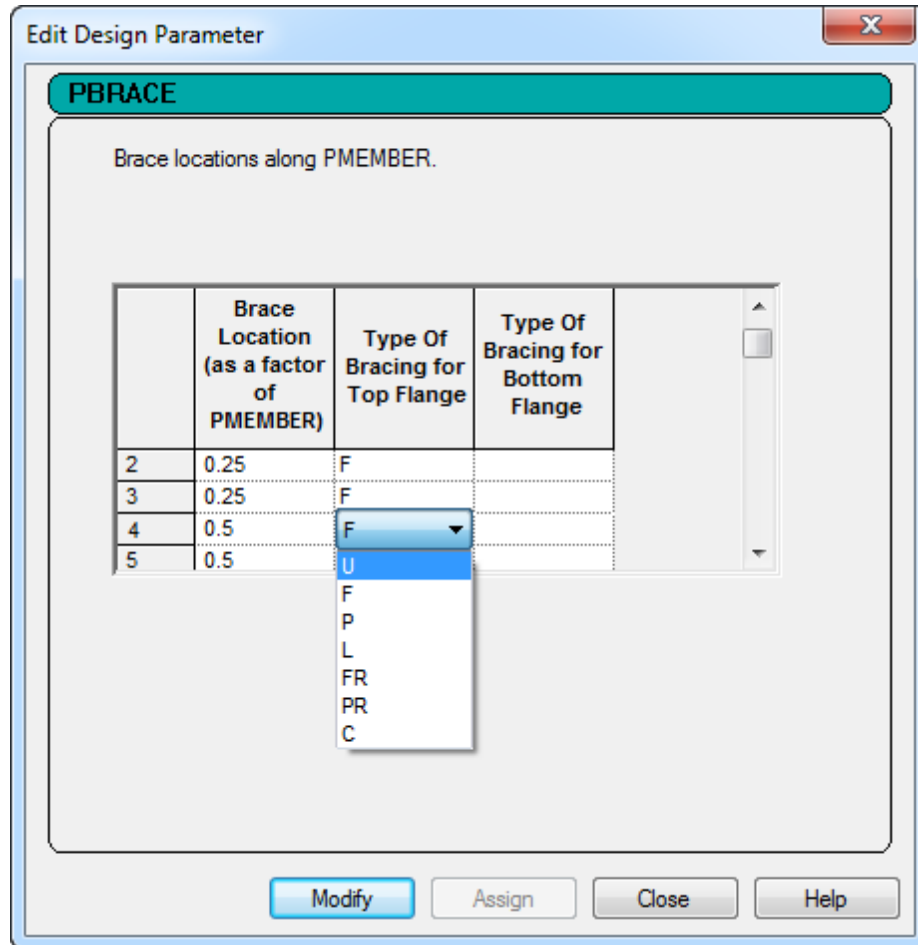
The physical member mode is initiated through the Toggle Physical Member mode tool found in the Steel Design toolbar (which is docked on the left hand side of the screen by default). This "mode" is used when modeling the structure and any parameter or specification added while this tool is toggled on will then only be available for physical member groups.

**Hint:** Command entries in the tree, material properties, and specifications will be designated with "(Physical)" when added in this mode.

Physical members are then formed or selected using the tools in the Steel Design toolbar. Refer to "Using Physical Members for Design" in Section 1.4 of the Graphical User Interface manual for additional information.

## Physical Member Restraints

A new parameter has been added for AS 4100 physical members to describe the bracing conditions/locations on a physical member. This parameter describes where the restraint is located along the length of the physical member and the type of restraint on the top or bottom flange.



> **To manually add physical member restraints**

**Hint:** The restraint details can be automatically generated using a new **Place Restraints on Physical Members** item found in the Tools menu. Refer to section 2.3.4 of the STAAD.Pro Graphical User Interface manual for additional information on using this feature.

1. Create a STAAD.Pro model with steel members.
2. Select the **Design | Steel** page.
3. Form one or more physical members in the model.
4. Select **AS4100** as the Current Code.
5. Click the **Define Parameters...** button.  
The Design Parameters dialog opens.
6. Select the PBRACE item in the parameters list.
7. For each brace point, specify a fraction of the total physical member length in the Brace Location cell.

8. Select the type of restraint present for the top or bottom flange.

**Note:** Only top or bottom flange restraints can be described using a single PBRACE command. If both top and bottom bracing is added in the Design Parameters dialog, this will generate two new command lines in the input file.

9. Click the **Add** button.

The new command appears in the tree as a child element of the current design parameter set.

10. Assign the torsion-related parameters to members as needed, just as you would any other design parameter.

The PBRACE specification is tagged as (Physical), and therefore can only be assigned to physical members (PMEMBER groups).

## Engine Physical Member Validation

When creating physical members, the STAAD.Pro graphical interface will check to ensure that the analytical members included in a physical member definition are both interconnected and colinear. However, it is not uncommon for input files to be generated outside of the STAAD.Pro graphical environment. Thus, these checks are now performed by the STAAD.Pro Analysis & Design engine again when an analysis is performed. You will be alerted if either condition is not met.

**Hint:** The analytical members contained in a physical member definition must be colinear (or, all lying in a straight line). Each adjacent analytical member must be within 5° of one another to meet this condition.

Refer to section 1B.8 "Design Parameters" of the International Codes Manual for additional information using the new **SGR** (steel grade) and **LHT** (load height position) design parameters for steel design per AS4100. Refer to section 1B.12 "Physical Member Design" for additional information on using the PBRACE parameter and performing the design of physical members per AS4100.

## AD.2007-07.2.9 SNIIP Steel Design Update

### Purpose

Several minor enhancements have been made to STAAD.Pro regarding steel design per the SNIIP 2.23-81 code.

### Description

The following corrections and enhancements were made to the SNIIP 2.23-81 steel design code implementation in STAAD.Pro:

- PHI and NIU factors messages extensions explaining different design results cases
- Additional bending check for non axial compression/ tension with small eccentricity

- 
- Compressed steel member design by weakened section
  - Full Check by combinations envelope for different sections of each steel member. Print of Analysis results for each member section
  - Additional parameters for Steel grade by EC3 in EN 10025-2 steel tables
  - Correction of other minor bugs and errors

## **AD.2007-07.2.10 Geometric Nonlinear Analysis Cycle Control**

### **Purpose**

You are now able to limit the analysis cycle using a displacement limit control. These controls can be found on the Analysis/Print Commands dialog Nonlinear Analysis tab or input manually in the input command file.

### **Description**

The displacement limit control allows you to select a nodal displacement degree of freedom to be monitored during a geometric nonlinear analysis. A target displacement is set and, if the number of load steps set is two or greater, the analysis will proceed step-by-step until the target displacement is met or exceeded. This provides you with an additional, practical means of limiting the number of steps used in a GNL analysis.

#### **> To specify a GNL analysis with a target displacement**

1. Select the **Analysis/Print** page.

or

Select **Commands > Analysis > Non-linear Analysis...**

The Perform Nonlinear Analysis dialog opens (or opens in the Analysis/Print Commands dialog Nonlinear Analysis tab).

2. Specify at least two Load Steps.
3. Specify a Node number for the program to monitor during each load step.  
If node was selected prior to opening the dialog, then click the [...] button to use that node.
4. Select a DOF (degree of freedom) which will be monitored for the specified Node.
5. Specify a Target Value for the selected DOF.
6. Click the **Add** button to add this analysis command to the input command file.

**Note:** Refer to Section 5.37.8 of the Technical Reference Manual for additional details on using the GNL Analysis Cycle Control.

---

## AD.2007-07.2.11 Jindal Steel Section Database

A number of Jindal Power & Steel Limited (JPSL) catalog sections have been added to the section database.

### > To add a section from the available JPSL catalog

1. Select the **General | Property** page.
2. In the *Properties - Whole Structure* dialog, click **Section Database**.  
The *Section Profile Tables* dialog opens.
3. On the Steel tab, select the **Jindal** entry in the table families list.
4. Select the table, section, and type specification.
5. Click **Add**.

## AD.2007-07.2.12\* Design per ASME NF 3000 2004 Code

The design of steel sections according to the requirements in the American Society of Mechanical Engineers (ASME) specifications, Rules for the Construction of Nuclear Power Plant Components, Section III – Subsection NF has been implemented per the 2004 edition of this code and the **Design | Steel** page has been updated to allow the design parameters to be defined and assigned.

### To perform a steel design per ASME NF 3000 2004

Use the following procedure to specify post analysis steel design code checking requirements for the ASME NF code.

1. Create a model with steel members.
2. Select **ASME NF3000 2004** for the Current Code on the **Design | Steel** page.
3. Click **Define Parameters...**  
The *Design Parameters* dialog opens.
4. Specify parameters as required.
5. Close the *Design Parameters* dialog.
6. Assign parameters to members as needed.
7. Select **Analysis > Run Analyze** (or press CTRL+F5).

For more information on the technical requirements of this design code, including the full set of parameters and default values, refer to Section 18D. of the International Design Codes Manual.

**Note:** The STAAD Nuclear Code pack is required to perform designs per an ASME NF 3000 code.

---

## AD.2007-07.2.13\* Update to ANSI AISC N690 1984 & 1994 Codes

Four new Stress Limit Coefficients (SLC) parameters have been added for designs per ANSI/AISC N690 1984/1994 codes.

These parameters, **SFC**, **SFT**, **SMZ**, and **SMY**, all default to 1.0 and are used to control the interaction equations in Section Q1.6 of the ANSI/AISC N690 1984/1994 codes.

Equations Q1.6-1a, Q1.6-1b, Q1.6-2 and Q1.6-3 of code ANSI/AISC N690 1994 will be rewritten as follows:

- Members subjected to both axial compression and bending stresses are proportioned to satisfy equation Q1.6-1a:

$$\text{SFC} \cdot f_a / F_a + \text{SMY} \cdot C_{my} \cdot f_{by} / [(1 - f_a / F'_a) F_{by}] + \text{SMZ} \cdot C_{mz} \cdot f_{bz} / [(1 - f_a / F'_a) F_{bz}] \leq 1.0$$

and Q1.6-1b

$$\text{SFC} \cdot f_a / (0.6 \cdot F_y) + \text{SMY} \cdot f_{by} / F_{by} + \text{SMZ} \cdot f_{bz} / F_{bz} \leq 1.0$$

when,  $f_a / F_a > 0.15$ , as per section Q1.6.1 of the code.

- Otherwise, equation Q1.6-2 must be satisfied:

$$\text{SFC} \cdot f_a / F_a + \text{SMY} \cdot f_{by} / F_{by} + \text{SMZ} \cdot f_{bz} / F_{bz} \leq 1.0$$

- Members subjected to both axial tension and bending stress are proportioned to satisfy equation Q1.6-1b:

$$\text{SFT} \cdot f_a / (0.6 \cdot F_y) + \text{SMY} \cdot f_{by} / F_{by} + \text{SMZ} \cdot f_{bz} / F_{bz} \leq 1.0$$

Refer to Section 17B.2.6 of the International Design Codes Manual for additional information on using the ANSI N690 1984 and 1994 codes.

## AD.2007-07.2.14\* Load Combination Enhancements

It is now possible to refer to a previously defined load combination within a new load combination. For example, a SRSS combination of individual response spectrum cases can now be referenced in a load combination along with dead load, live load, etc.

There are no changes to the input file syntax. Load combination definitions may now refer to

There is no limit to the amount of load combination "nesting" which can be done in STAAD, other than the total limit of load cases and load combinations allowed by the program.

Refer to Section 5.35 of the Technical Reference Manual for additional information on using Load Combinations.

## AD.2007-07.2.15\* Enhancement to Maximum Number of Response Spectrum Load Cases

STAAD.Pro now supports up to 50 response spectrum load cases, instead of the previous limit of four.

Refer to Section 5.32.10.1 of the Technical Reference Manual for additional information on using Response Spectra.

---

## AD.2007-07.3 Features Affecting Post Processing

Several new features have been added and existing features have been modified in the Post Processing modes. These are explained in the following pages.

### AD.2007-07.3.1 RAM Connection V8i (SELECTseries 1) Support

#### Purpose

The enhancements included in Bentley's RAM Connection V8i (releases 6 and 7) are now available in STAAD.Pro. This includes new connection types, new codes, and design for seismic loads.

#### Description

Some of the new features and enhancements include:

- Now compatible with versions of RAM Connection V8i up through release 7.0 (SELECTseries 3).
- British Design Code - Connection design per BS5950-1:2000 (British standard) has been added. This code can now be selected along side AISC codes.
- AISC Seismic Provisions - The seismic provisions of AISC 341-05 have now been added for connections per AISC codes.
- Seismic Frame Management - A new Seismic Frames page has been added to assign the lateral seismic resisting system classification to frames for connection design. This page is also used to add plastic hinge locations to beam members.
- Base Plate Design is now available for AISC (ASD & LRFD) connections. Column base plates are available in the Smart Connections dialog and Column-Brace gusset base plates are available in the Gusset Connections dialog.
- The RAM Material dialog has been expanded to accommodate for various materials from different countries. UK steel, bolt, and weld types have been added, as well as concrete and anchor bolt materials for US base plate design.

**Note:** A notification message may be displayed when selecting the RAM Connection mode that you need to provide some additional material properties.

- Several new selection methods have been added to the **Select > By Joints >** sub-menu.
- Reports have been enhanced with code references and display of formulas used.

**Note:** Refer to the What's New section in the RAM Connection mode Help for additional information on using these features.

---

## AD.2007-07.3.2 RC Designer

Several new features have been added and existing features have been modified in the RC Designer mode. These are explained in the following pages.

### AD.2007-07.3.2.1 ACI 318 Metric

A new design code has been added for beams, columns, and slab element designs in the RC Design module for the metric version of ACI 318-05.

ACI 318M-05 is the standard published by the American Concrete Institute which describes the equations to be used for metric design. It differs from the normal ACI 318-05 standard in that it has been converted to a "soft metric" form, where conversions have been applied to the numbers used in the formulae and to the bar sizes, and then some rounding has been done. This means that the bars to be used are the same as with US customary units, but they are quoted to the nearest millimeter.

**Note:** Refer to Section 3.4.1(B) of the RC Designer manual for additional information.

**Note:** A valid license for the U.S. Design Codes (Standard) package is required to use this feature.

#### > To create a ACI 318M-05 Design Brief for columns or beams

1. Open a model in RC Designer.
2. Select the Groups/Briefs Page in the page control.
3. Click the **New Brief** button at the bottom of the Design Briefs table.  
The New Design Brief dialog opens.
4. Enter a title in the B# field.
5. Select **ACI 318M-05** in the Design Code list.
6. Select the appropriate Design Type (either Beam or Column).
7. Click the **OK** button.

The corresponding ACI 318M-05 Beam Brief or ACI 318M-05 Column Brief dialog opens, depending on the choice made above.

8. Specify parameters as needed.

**Note:** The design briefs are identical to those for ACI 318-05 checks with all entries being input in "soft" metric units as indicated.

9. Click the **OK** button.

---

## AD.2007-07.3.2.2 GB50010

### Purpose

Concrete beam and column design per the GB 50010-2002 code is now available.

### Description

The GB50010 checks are initiated by selecting this code for the design brief and selecting the appropriate parameters. Refer to Sections 3.4.14 and 4.5.3.14 of the Concrete Design Mode help for additional information on using this new feature.

**Note:** A valid license for the Asian Design Codes package is required to use this feature.

### > To create a GB50010 Design Brief for columns or beams

1. Open a model in RC Designer.
2. Select the Groups/Briefs Page in the page control.
3. Click the **New Brief** button at the bottom of the Design Briefs table.

The New Design Brief dialog opens.

4. Enter a title in the B# field.
5. Select **GB50010** in the Design Code list.
6. Select the appropriate Design Type (either Beam or Column).
7. Click the **OK** button.

The corresponding GB50010 Beam Brief or GB50010 Column Brief dialog opens, depending on the choice made above.

8. Specify parameters as needed.
9. Click the **OK** button.

## AD.2007-07.3.2.3 IS456 with Seismic Design per IS13920

### Purpose

Design of columns and beams per IS 456 is now performed per the 2000 edition of that code. The option to perform additional reinforcement detailing checks for seismic conditions per IS 13920:1993 is also now available. This code is used for the ductile detailing of reinforced concrete structures subjected to seismic forces

### Description

The IS456 checks are initiated by selecting this code for the design brief and selecting the appropriate parameters. Additional seismic checks per IS 13920 are specified by selecting this option on the design brief

---

General tab. Refer to Sections 3.4.10 and 4.5.3.10 of the Concrete Design Mode help for additional information on using this new feature.

**Note:** A valid license for the Indian Design Codes package is required to use this feature.

### > To create an IS456 Design Brief for columns or beams

1. Opens a model in RC Designer.
2. Select the Groups/Briefs Page in the page control.
3. Click the **New Brief** button at the bottom of the Design Briefs table.

The New Design Brief dialog opens.

4. Enter a title in the B# field.
5. Select **IS456** in the Design Code list.
6. Select the appropriate Design Type (either Beam or Column).
7. Click the **OK** button.

The corresponding IS456 Beam Brief or IS456 Column Brief dialog opens, depending on the choice made above.

8. Specify parameters as needed.
9. Select the option to Check for IS 13920 on the General tab to perform additional seismic design.

**Note:** See the following procedure for designing for IS 13920.

10. Click the **OK** button.

### > Design longitudinal and shear reinforcement in beams per IS13920

1. In the IS456 Beam Design Brief dialog General tab, select the Check for IS13920 option.
2. Perform a beam design.
3. Select the Earthquake page.
4. Select the appropriate load case is selected to perform the seismic checks to allow for a strong column-weak beam mechanism.

The program internally calculates the moment capacities of the beams and columns framing into a joint and assign a 'pass/fail' status depending on the corresponding beam and column capacities at that joint for the direction under consideration.

5. Select the Earthquake | Reinforcement page to review the status of the IS 13920 design.

## AD.2007-07.3.3 Enhanced Geometric Nonlinear Post Processing

### Purpose

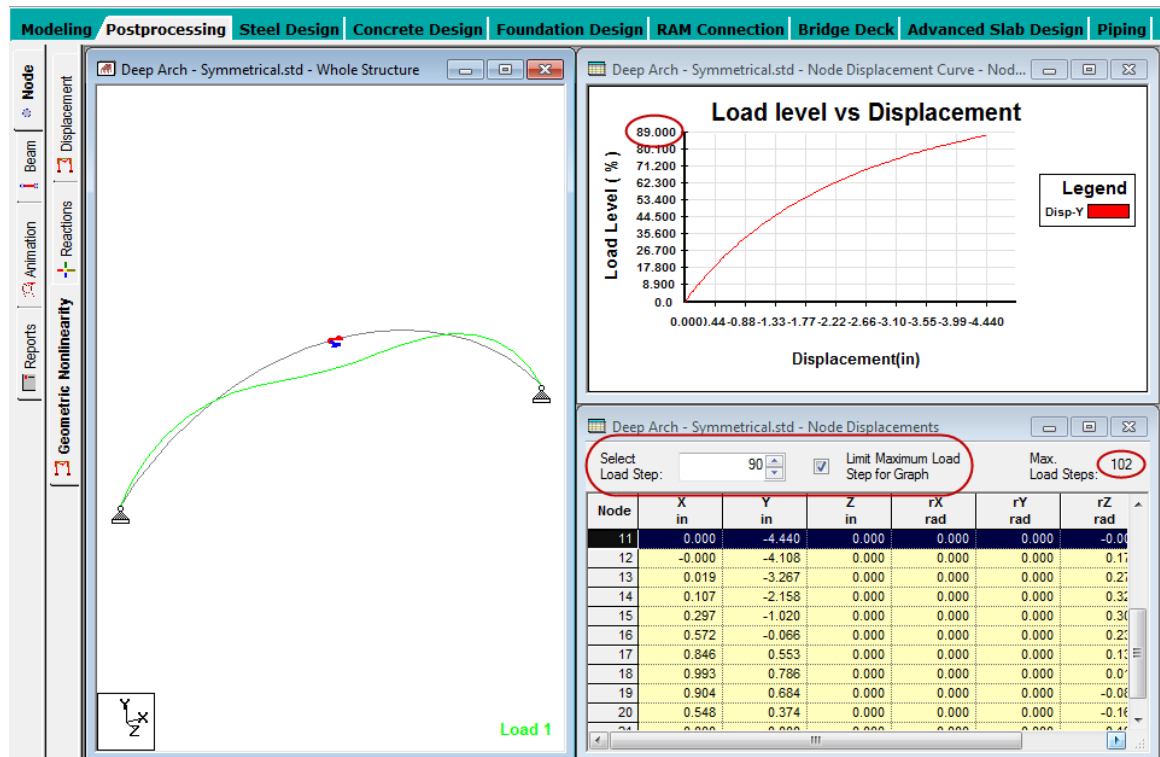
The graphical display of the results of a geometric nonlinear analysis have been improved. The Graphs are now easier to read and a new control has been added to limit the maximum Load Step displayed.

### Description

When a structure is analyzed with a geometric nonlinear analysis, the nodal displacements can be viewed by selecting the Node | Nonlinearity page in the Post-Processing mode. The Node Displacements Curve has been enhanced and a new control has been added to the Node Displacements table to limit the maximum Load Step plotted.

In the event that a load level is specified which exceeds the non-linear buckling capacity of a structure, the analysis performed by STAAD.Pro will produce exceedingly large post-buckling displacements. This signifies that the load steps in this post-buckling level are beyond the scope of designed performance of the STAAD engine (and likely beyond the load level and effects intended by the engineer). In these cases, the scale of the non-linear load displacement curve was did not adequately display the pre-buckling characteristics due to the very large scale required to display the post-buckling displacements.

Now, in the event of a large post-buckling load, the maximum load level scale of the Load Level vs. Displacement graph can be limited to the currently selected load step by selecting this option on the Node Displacements table, thus allowing you to see the pre-buckling behavior.



---

## AD.2007-07.3.4 AutoPIPE V8i (SELECTseries 2) and PipeLink support

### Overview

STAAD.Pro is now capable of two-way data exchange with AutoPIPE via a new PipeLink utility. Additionally, the tools for transferring loads between the pipe model and the structural model have been enhanced such that you can now select, update, and remove loads to be applied to the structural model.

**Note:** For details on the updates to Bentley AutoPIPE V8i, refer to the documentation included with that product.

### Description

Some of the new features and enhancements include:

- The integration between STAAD.Pro and AutoPIPE has been enhanced to allow bi-directional transfer of data between STAAD.Pro and AutoPIPE. This is accomplished through a new plug-in program called PipeLink, which is used to generate common database files for use in both AutoPIPE and STAAD.Pro.
- The system now allows pipe models to be imported from a number of databases and these models may all be stored locally. Work is then performed on the active model.
- The loading transfer process has been significantly revised. The mappings between the pipe loadings and the generated structure loadings are recorded so that these structure loadings may be revised or removed rather than the previous once-only method. Further, loads can now be modified individually.
- The program now connects pipe supports – rather than pipe nodes – to the structure. This allows both node and support labels to be used. This allows additional filters in the connection wizard and also allows STAAD.Pro to import multiple supports at a single support node.
- Pipe supports are now graphically displayed with icons which reflect the nature of the pipe support. The display of these icons, along with other pipe model elements, can be controlled through a new tab in the Diagrams dialog.
- A new section of the online documentation has been added for the Piping mode. This can be found Section 7 of the STAAD.Pro User Interface help.

Refer to the What's New in the Piping Mode section for additional information on using these new and updated features.

## AD.2007-07.3.5 Transverse IRC Loading in STAAD.Beava

### Overview

For the specific bridge width, IRC (Indian Road Congress) chapter 6-2000, table 2, clause 207.4 defines the rules to combine the live loads. This new feature allows you to either use these IRC live load rules or use an iterative, custom method. If the IRC rule option is selected, this function uses the appropriate live

loads and number of design lanes and the generates all the possible load combinations as stated for this particular bridge width. Otherwise, you can select a specific live load and design lane to generate combinations.

## Description

Loading rules per IRC Chapter 6 are applied in much the same way as previous codes. The defined roadway for the selected deck(s) is divided into design lanes and the selected load class is applied to the structure achieve the specified actions.

### > To specify loading per IRC Chapter 6

1. Open an analyzed bridge model in the Bridge Deck mode.
2. Create Deck and Roadway definitions.
3. Generate influence surfaces for the structure.
4. Select **Loading > Run Load Generator...**

The Load Generation Parameters dialog General tab opens.

5. Select IRC Chapter 3 for the Design Code and select the appropriate Limit State.  
The <code> tab updates to display IRC Loading.
6. Select the IRC Loading tab.

	Multiple Presence Factor
1	1.000
2	1.000
3	0.900
4	0.800
> 4	

7. Select the appropriate Loading Class.

**Note:** Combinations of the AA, B, and 70R vehicles have been added to the included vehicle definitions. These may be reviewed in the Vehicle Database dialog.

8. (Optional) Specify an impact factor or modify the Multiple Presence Factors as needed.

- 
9. Specify the decks for consideration on the Decks tab and the load effects to dictate load placement on the Node Displacements, Support Reactions, Plate Center Stresses, and Beam End Forces tabs.
  10. Click the OK button.

The program places the selected loads in design lanes to produce the maximum or minimum effects requested. A text file containing a summary of the generated loads and corresponding effects is opened in a text editor for review.

## **AD.2007-07.3.6 STAAD.foundation V8i Integration**

### **Overview**

The export of support geometry and reactions to STAAD.foundation V8i can now be initiated from within STAAD.Pro using the Foundation Design mode. This feature is similar to the **Import STAAD.Pro File** capability included in STAAD.foundation.

### **Description**

When selected, the Foundation Mode opens the Foundation page which contains a view of the whole structure and the Foundation Design Options dialog.

From here, you can select to include all supports, you can graphically select supports, or you can specify a list of support numbers for exporting to a STAAD.foundation project. Similarly, the load cases from the analysis are listed for inclusion in the STAAD.foundation project.

**Hint:** Models containing a large number of supported nodes or load cases may result in slow performance on older computer hardware. Exporting a limited set of data can be used to improve performance in STAAD.foundation in these cases.

Planned future enhancements also include the export of mat foundations modeled in STAAD.Pro for design in STAAD.foundation.

Refer to Section 6 of the Graphical User Interface manual for more information on using the Foundation Mode.

## **AD.2007-07.3.7 Additional Section Databases in RAM Connection mode**

Steel section databases for the following countries are now available for use when design connection in RAM Connection mode:

- Indian
- European
- Japanese
- Australian

---

**Note:** Connection design is only performed per the US and British codes available in RAM Connection.

## **AD.2007-07.4 Additional Features**

The following features have yet to undergo testing and are presented "as is."

**Note:** Items labeled with an asterisk (\*) were added in the QA&R release of V8i (SELECTseries 2) (Build 20.07.07.32).

### **Beta Features**

The following features have yet to undergo testing and are presented "as is."

- (None)

### **AD.2007-07.4.1\* Design of Class 4 "Slender" Sections in IS800:2007**

The design of slender classified sections (only rolled or welded I sections) per IS:800-2007 has been added to STAAD.Pro.

The IS:800-2007 code does not provide any clear guidelines about what method should be adopted for the design of slender section. The "Flange Only" concept has been adopted where it is assumed that flexure is taken by the flanges alone and the web will resist shear with adequate shear buckling resistance. This means that the flange elements must be non-slender with slender web element to qualify for slender section that can be designed. If any of the flanges become slender, the design will not be performed for Bending and a warning message is displayed.

Refer to Section 9E of the International Design Codes manual for additional information on the design procedures used for slender sections for IS800:2007 as well as a verification example problem.



## Section 3

# STAAD.Pro V8i (SELECTseries 1)

---

## Introduction

The Software Release Report for STAAD.Pro V8i (SELECTseries 3) contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i (release 20.07.05). This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

## AD.2007-06.1 Features Affecting the General Program



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

## AD.2007-06.1.1 CIS/2 Translator Update

### Purpose

The STAAD.Pro tool to import and export models with the CIS/2 translator has been enhanced for the transfer of models into 3D modeling, such as Intergraph SmartPlant® 3D (SP3D).

### Description

The CIS/2 (CimSteel Integration Standard, Version 2) allows for the transfer of steel models using a prescribed data standard in the STEP (Part 21) format. These files can contain different models including analysis models. In previous versions of STAAD, CIS/2 files could not be imported into an existing STAAD file. The import process has now been updated so that new STAAD input files can be created or existing input files updated from a CIS/2 file.

STAAD will retain all relevant information generated by SP3D - including the object IDs (GUIDs) - when importing CIS/2 files. Further modeling operation will be done in STAAD which includes special purpose load generations, analysis, design and member selections and modifications. You may then export out to a CIS/2 STEP file and retain all information inherited from SP3D STEP file and addition/deletion/modification information performed in STAAD.

Using import and export of STEP files in SP3D, further modification can be made in SP3D and the STAAD model can be updated its model. Only geometry, member properties, boundary condition information are within the update scope of STAAD. While updating the STAAD model no other information will be considered. This round-trip process can be repeated an unlimited number of times

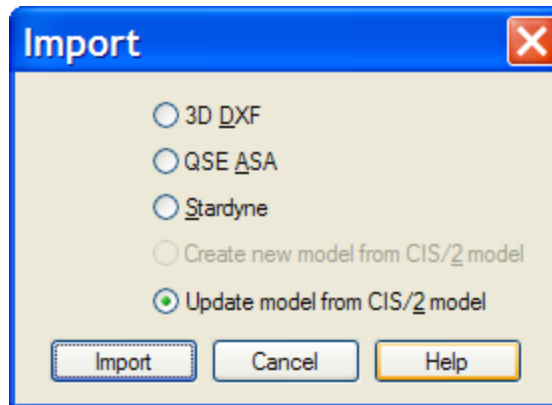
Additionally, two new VBS Macros have been included in the User Tools to aide in the verification of model integrity before and after update operations using the Import dialog. These may be found in the User Tools drop-down menu in the File toolbar or under Tools > User Tools.

#### > Importing or Updating with CIS/2 Files

The initial structural model will be created in your external 3D modeling software and exported as a CIS/2 STEP file, consisting of both analytical and physical model definitions. Limited load modeling can be done in some modeling software - such as SmartPlant® 3D - before the initial export.

1. In STAAD.Pro, select File > Import...

The Import dialog opens.



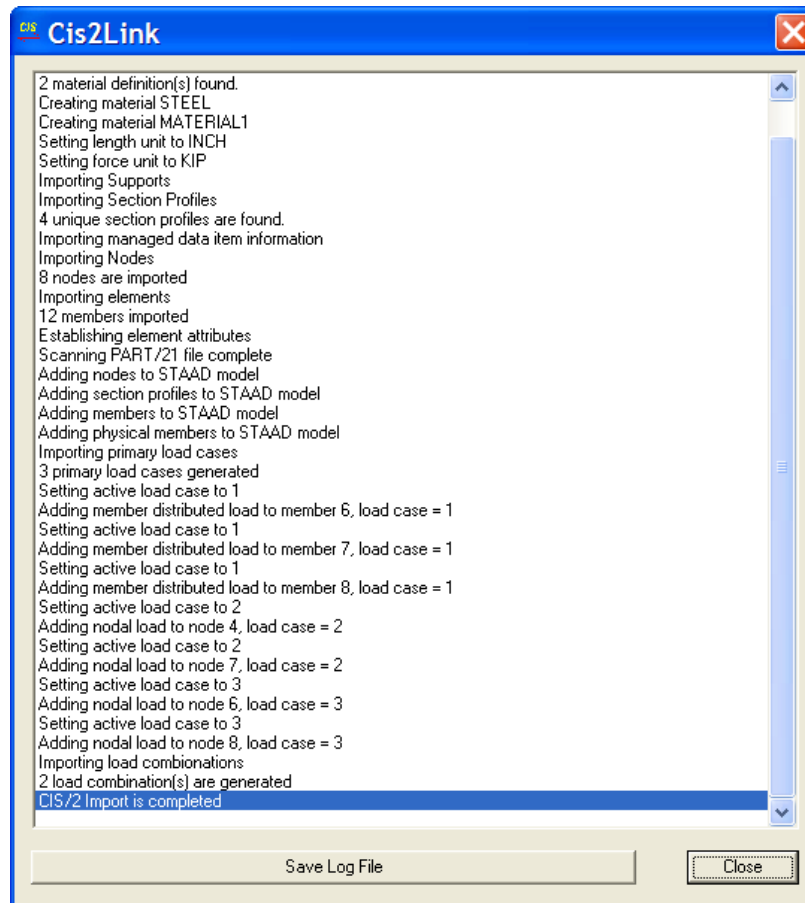
2. Select either to Create new model from CIS/2 model or Update mode from CIS/2 model.

**Note:** The option to Create new model from CIS/2 model will only be available in an empty STAAD.Pro project. For STAAD.Pro projects with model content, the option to update will be available.

3. Click the Import button.  
The Cis2Link dialog opens.
4. Click the Import Model button.

**Note:** If you are updating an existing STAAD model, the button changes to display Update STAAD Model.

The data is read and the import progress is reported in the log window. Click the Save Log File button to save this information to a text file.



5. Click the Close button.

The model data has been imported into your STAAD project.

#### > Export to CIS/2 Files

1. Select File > Export...

The Export dialog opens

2. Select the CIS/2 format.
3. Click the Export button.

The Cis2Link dialog opens.

4. Click the Export Model button.

The data export progress is displayed in the log window. Click the Save Log File button to save this information to a text file.

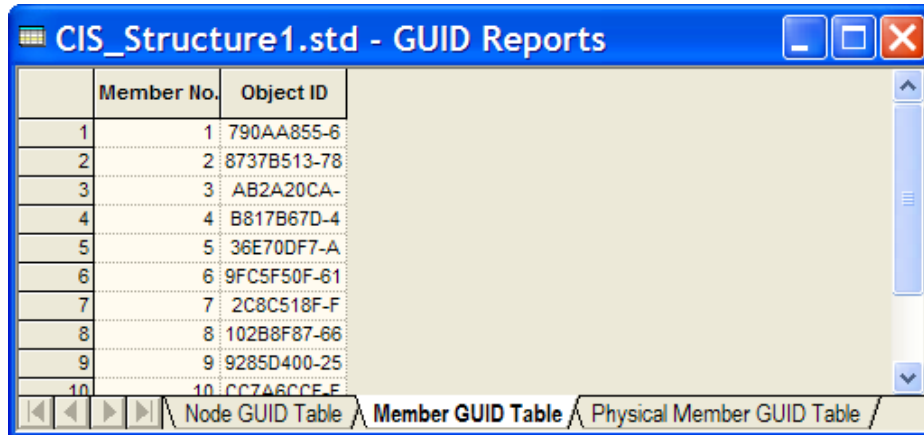
5. Click the Close button.

## New User Tools (VBS Macros)

Two VBS macro are developed to assist users to confirm the model integrity before and after the update operations.

### List Object GUIDs

Opens the GUID Report tables. This is useful to verify that the GUIDs are same as those in SP3D.

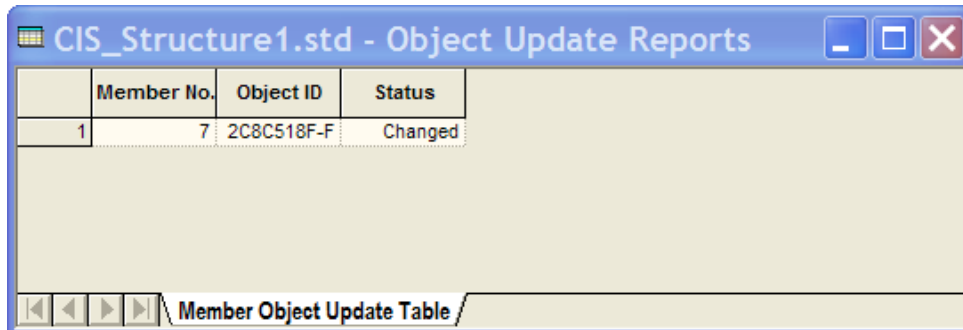


	Member No.	Object ID
1	1	790AA855-6
2	2	8737B513-78
3	3	AB2A20CA-
4	4	B817B67D-4
5	5	36E70DF7-A
6	6	9FC5F50F-61
7	7	2C8C518F-F
8	8	102B8F87-66
9	9	9285D400-25
10	10	CC7A6CCE-E

Node GUID Table / **Member GUID Table** / Physical Member GUID Table

### CIS/2 Object Update Report Tool

Used to capture model state before an update operation (Create Pre-Update Report) and then same tool can be invoked again after the update process to generate a report showing what exactly has been updated (Create Post-Update Report).



	Member No.	Object ID	Status
1	7	2C8C518F-F	Changed

Member Object Update Table

## AD.2007-06.2 Features Affecting the Analysis and Design Engine



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

### **AD.2007-06.2.1 ANSI/AISC N690-1994 Design Code**

#### **Purpose**

For steel design, STAAD compares the actual stresses with the allowable stresses as defined by the "ANSI/AISC N690-1984: Nuclear Facilities - Steel Safety-Related Structures for Design, Fabrication, and Erection."

#### **Description**

The parameter **CODE AISC N690 1984** is used to initiate code checking per ANSI/AISC N690-1984.

The full details for this code - including parameters, commands, and technical background - are in the International Design Code manual section 17B "ANSI/AISC N690-1984 Code".

#### **> Use the ANSI/AISC N690 1984 code**

1. In the modeling mode, select the Design page and Steel material sub-page.
2. Select AISC N690 1984 in the Current Code drop-down list.

### **AD.2007-06.2.2 Update to Russian Concrete Design**

The Russian SNIIP concrete design routines have been updated to accommodate new reinforcement and concrete class definitions. In order that these new classes can be assigned to members that are to be designed, the following changes have taken place in the RCL, BCL and RHS parameters.

Refer to the International Codes Manual, section 12A "Russian Codes – Concrete Design per Russian Code (SNIIP 2.03.01-84\*)" for additional information.

Additionally a few other minor updates have been incorporated to ensure axial tension is ignored in column design and that the provided area of steel in both directions is not less than the minimum.

**Note:** STAAD.Pro supports design of concrete beams and columns to Russian SP52 code is supported in the interactive RC Designer module (Concrete Design Mode in the GUI). For more details of the RC Designer see the RC Designer manual found in the Additional Modules section of the online help. For details on the SP52 details see "AD.2007-1001.4.1 Beam and Column Designs to the Russian Concrete Code SP52."

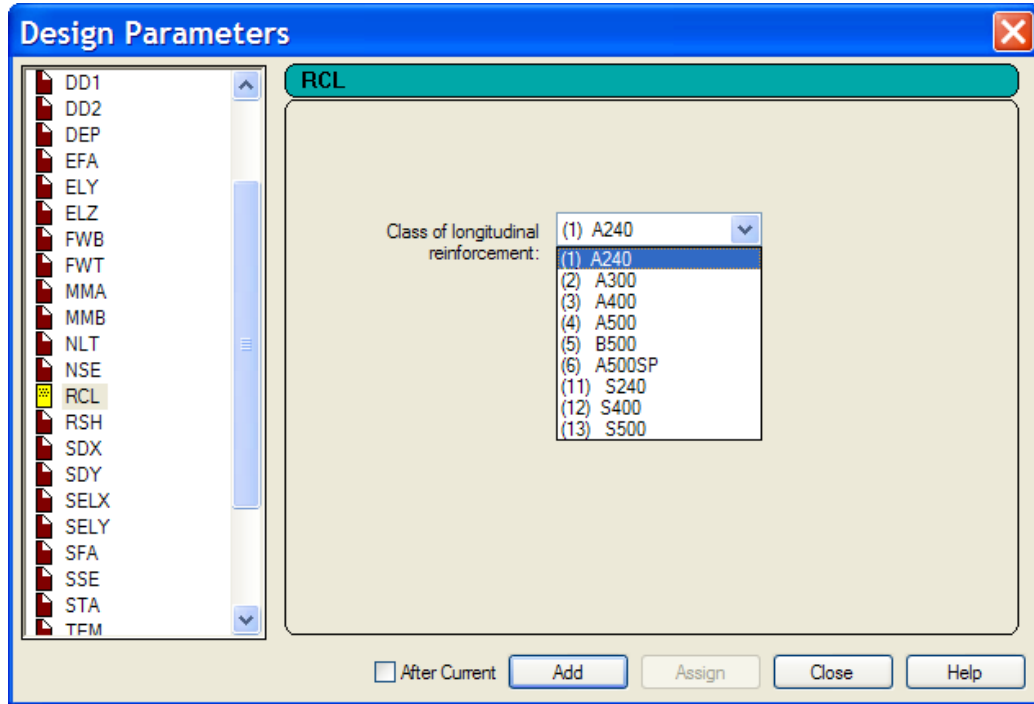
**> Select reinforcement or concrete class definitions**

1. In the modeling mode, select the Design page and Concrete material sub-page.

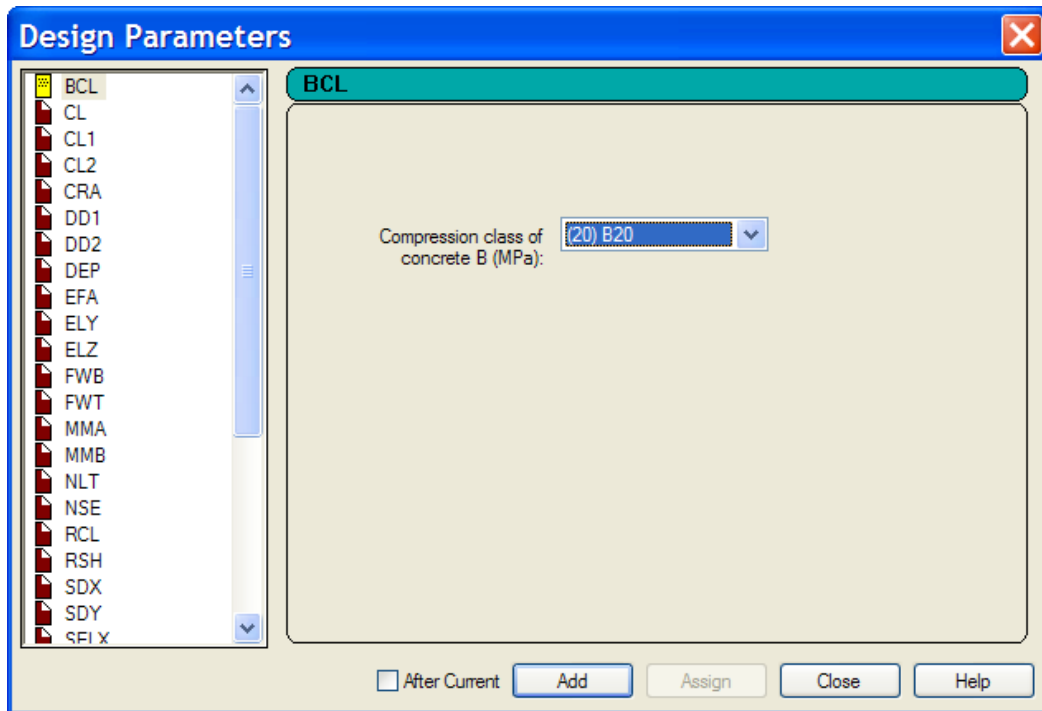


2. Select SNiP 2.03 01-84 in the Current Code drop-down list.
3. Click the Define Parameters button below the model outline panel.
4. The RCL, BCL, and RSH parameters have the new definitions available. Refer to the International Codes Manual for additional information.

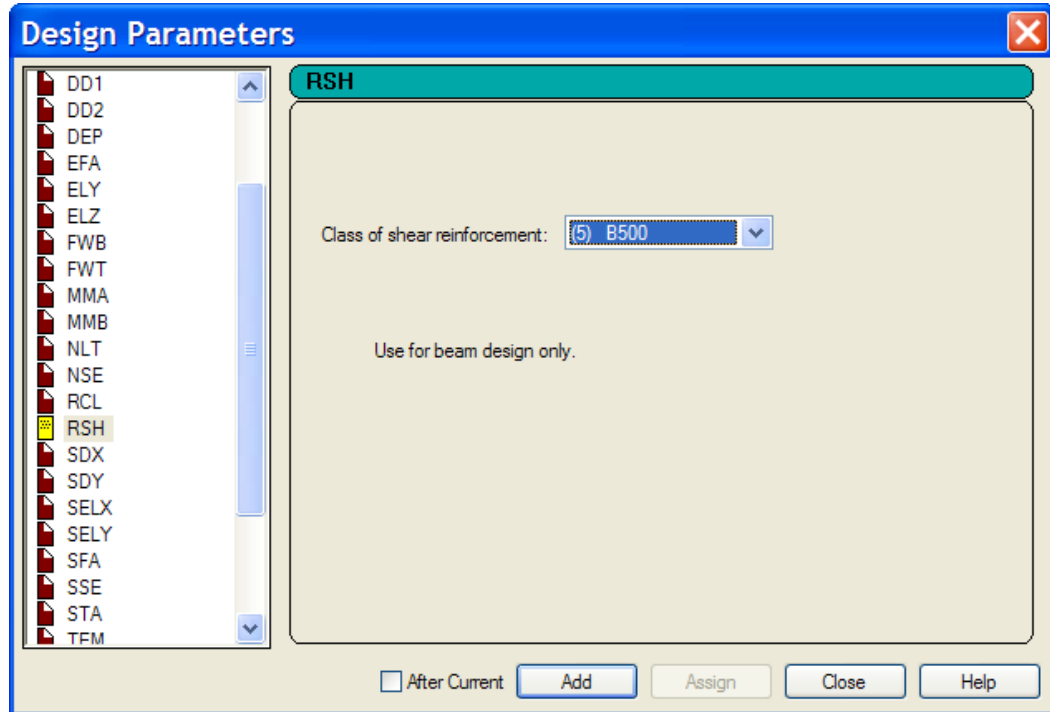
Reinforcement Class for longitudinal reinforcement (RCL):



Compression class of concrete B (BCL):

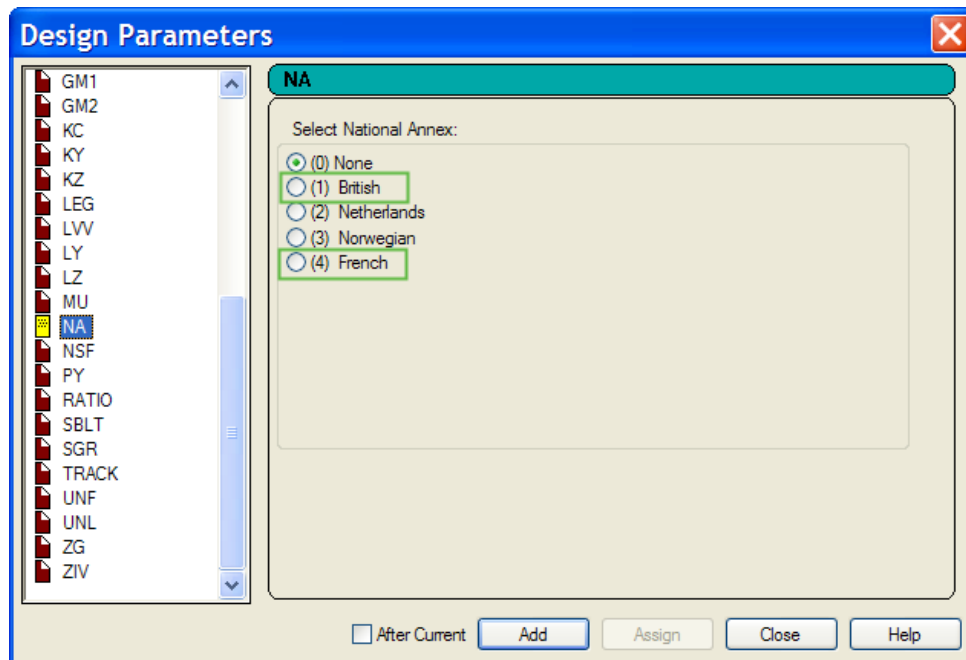


Reinforcement class for shear reinforcement (RSH):



### AD.2007-06.2.3 Eurocode 3 National Annex

Two country's National Annex to Eurocode 3 have been incorporated into the Steel design module in STAAD.Pro: United Kingdom and France. As with the other National Annexes to EC-3, this implementation will make use of the **NA** parameter.



**Warning:** The **GB1** parameter (which, in fact was common to the base EC-3 and was a reminiscent of the previous DD ENV implementation of EC-3) has been removed. Hence any legacy STAAD files that have the **GB1** Parameter defined will need to be revised to take out this parameter as it is no longer valid as per the latest EN1993.

### **AD.2007-06.2.3.1 United Kingdom National Annex to Eurocode 3 (EN 1993-1-1:2005)**

The UK National Annex document referred to is “NA to BS EN 1993-1-1:2005”.

#### **> To Initiate a EC3-UK NA Steel Design**

1. In the Modeling mode, click the Design > Steel tab.
2. In the Current Code drop-down menu, select EN 1993-1-1:2005.
3. Click the Define Parameters button. The Design Parameters dialog opens.
4. Select the NA parameter in the list box.
5. Select the option for (1) United Kingdom.
6. Click Add.

This will insert the following command into the STAAD input file:

```
CODE EN 1993-1-1:2005
NA 1
```

For additional information, please refer to the International Design Codes manual, sections 5B.(B) "Steel Design to Eurocode 3" and 5B.(C) "EC3 National Annexes."

### **AD.2007-06.2.3.2 French National Annex to Eurocode 3 (EN 1993-1-1:2005)**

The French National Annex document referred to is “Annexe Nationale a la NF EN 1993-1-1:2005”.

#### **> To Initiate a EC3-French NA Steel Design**

1. In the Modeling mode, click the Design > Steel tab.
2. In the Current Code drop-down menu, select EN 1993-1-1:2005.
3. Click the Define Parameters button. The Design Parameters dialog opens.
4. Select the NA parameter in the list box.
5. Select the option for (4) France.
6. Click Add.

This will insert the following command into the STAAD input file:

CODE EN 1993-1-1:2005

NA 4

For additional information, please refer to the International Design Codes manual, sections 5B.(B) "Steel Design to Eurocode 3" and 5B.(C) "EC3 National Annexes."

## AD.2007-06.2.4 Chinese Static Seismic Loading

### Purpose

A simplified base shear method of the seismic load generation for the Chinese code has been added to STAAD.Pro V8i.

This set of commands may be used to define and generate static equivalent seismic loads as per Chinese specifications GB50011-2001. This load uses a static equivalent approach, similar to that found in the UBC. Depending on this definition, equivalent lateral loads will be generated in the horizontal direction(s).

### Description

The seismic load generator can be used to generate lateral loads in the X and Z directions for Y up and the X and Y directions for Z up; where Y up or Z up is the vertical axis parallel to the direction of gravity loads (see the SET Z UP command).

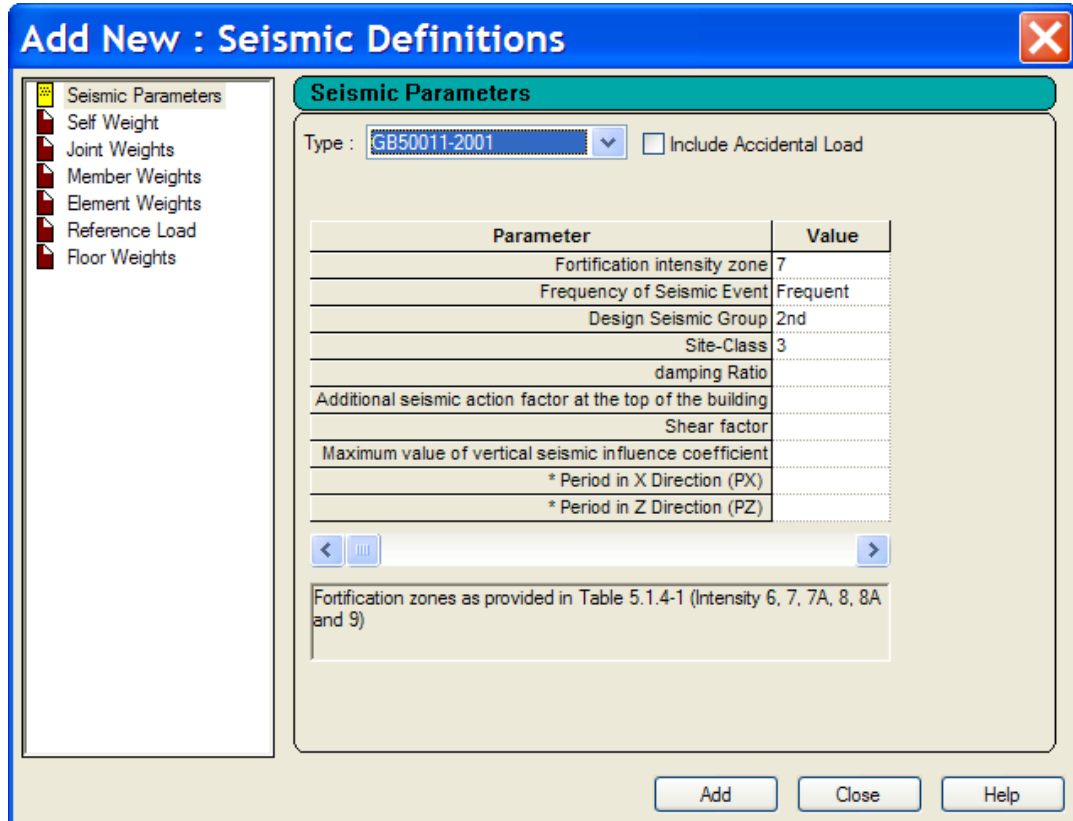
**Note:** All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

This method of seismic load generation is limited in use to buildings not taller than 40 meters, with deformations predominantly due to shear, and a rather uniform distribution of mass and stiffness in elevation. Alternately, for buildings modeled as a single-mass system, a simplified method such as this base shear method, may be used.

### > Adding GB50011-2001 Seismic Loading

1. Click on the General | Load & Definitions page in the Modeling mode.
2. Select **Definitions > Seismic Definitions**.
3. Click the **Add** button.

The Add New: Seismic Definitions dialog opens.



4. Select GB50011-2001 from the Type drop-down list.
5. Specify if to Include Accidental Load if required.
6. Specify a value for each parameter in the table below (see [general format](#) for definitions).
7. Click the **Add** button.

## Methodology

### Gravity Loads for Design

In the computation of seismic action, the representative value of gravity load of the building shall be taken as the sum of characteristic values of the weight of the structure and members plus the combination values of variable loads on the structure. The combination coefficients for different variable loads shall be taken from the following table.

Type of Variable	land Combination coefficient
Snow load	0.5
Dust load on roof	0.5
Live load on roof	Not considering

Type of Variable	land Combination coefficient	
Live load on the floor, calculated according to actual state	1.0	
Live load on the floor, calculated according to equivalent uniform state	Library, archives	0.8
	Other civil buildings	0.5
Gravity for hanging object of crane	Hard hooks	0.3
	Soft hooks	Not considering

### Seismic influence coefficient

This shall be determined for building structures according to the Intensity, Site-class, Design seismic group, and natural period and damping ratio of the structure. The maximum value of horizontal seismic influence coefficient shall be taken from Table 2.2; the characteristic period shall be taken as Table 2.3 according to Site-class and Design seismic group, that shall be increased 0.05s for rarely earthquake of Intensity 8 and 9.

Earthquake influence	Intensity 6	Intensity 7	Intensity 8	Intensity 9
Frequent earthquake	0.04	0.08 (0.12)	0.16(0.24)	0.32
Rarely earthquake	-	0.50(0.72)	0.90(1.20)	1.40

**Note:** The values in parenthesis are separately used for where the design basic seismic acceleration is 0.15g and 0.30g.

Earthquake Group	Site class			
	I	II	III	IV
1	0.25	0.35	0.45	0.65
2	0.30	0.40	0.55	0.75
3	0.35	0.45	0.65	0.90

### Calculation of seismic influence coefficient

The design base shear is computed in accordance with the equations shown below.

The damping adjusting and forming parameters on the building seismic influence coefficient curve (Fig.2.1) shall comply with the following requirements:

- A. The damping ratio of building structures shall select 0.05 except otherwise provided, the damping adjusting coefficient of the seismic influence coefficient curve shall select 1.0, and the coefficient of shape shall conform to the following provisions:
  - a. Linear increase section, whose period (T) is less than 0.1 s;
  - b. Horizontal section, whose period form 0.1 is thought to characteristic period, shall select the maximum value ( $\alpha_{max}$ );
  - c. Curvilinear decrease section, whose period from characteristic period thought to 5 times of the characteristic period, the power index ( $\gamma$ ) shall choose 0.9.
  - d. Linear decrease section, whose period from 5 times characteristic period thought to 6s, the adjusting factor of slope ( $\eta_1$ ) shall choose 0.02.

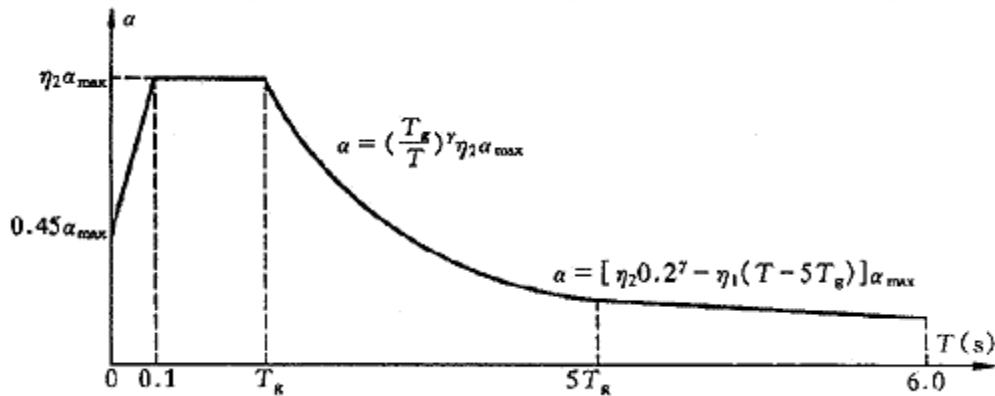


Figure 2.1 Seismic influence coefficient curve

- B. When the damping adjusting and forming parameters on the seismic influence coefficient curve shall comply with the following requirements:
  - a. The power index of the curvilinear decrease section shall be determined according to the following equation E2.1

$$r = 0.9 + \frac{0.05 - \zeta}{0.5 + 5\zeta} \quad \text{E2.1}$$

Where:

$\gamma$ - the power index of the curvilinear decrease section;

$\zeta$ - the damping ratio.

- b. The adjusting factor of slope for the linear decrease section shall be determined from following equation:

$$\eta_1 = 0.02 + (0.05 - \zeta) / 8 \quad \text{E2.2}$$

Where:

$\eta_1$ - the adjusting factor of slope for the linear decrease section, when it is less than 0, shall equal 0.

- c. The damping adjustment factor shall be determined according to the following equation:

$$\eta_2 = 1 + \frac{0.05 - \zeta}{0.06 + 1.7\zeta} \quad \text{E2.3}$$

Where:

$\eta_2$ - the damping adjustment factor, when it is smaller than 0.55 shall equal 0.55.

### Calculation of horizontal seismic action

When the base shear force method is used, only one degree of freedom may be considered for each story; the characteristic value of horizontal seismic action of the structure shall be determined by the following equations (Fig. 2.2):

$$F_{Ek} = \alpha_1 G_{eq} \quad \text{E2.4}$$

$$F_i = \frac{G_i H_i}{\sum_{j=1}^n G_j H_j} F_{Ek} (1 - \delta_n) \quad (i=1, 2, \dots, n) \quad \text{E2.5}$$

$$\Delta F_n = \delta_n F_{Ek} \quad \text{E2.6}$$

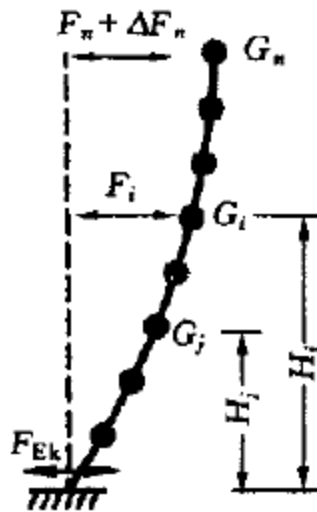


Figure 2.2 Calculation of horizontal seismic action

Where:

$F_{Ek}$ - characteristic value of the total horizontal seismic action of the structure.

$\alpha_1$ - horizontal seismic influence coefficient corresponding to the fundamental period of the structure, which shall be determined by using Clause 2.3. For multistory masonry buildings and multi-story brick buildings with bottom-frames or inner-frames, the maximum value of horizontal seismic influence coefficient should be taken.

$G_{eq}$ - equivalent total gravity load of a structure. When the structure is modeled as a single-mass system, the representative value of the total gravity load shall be used; and when the structure is modeled as a multi-mass system, the 85% of the representative value of the total gravity load may be used.

$F_i$ — characteristic value of horizontal seismic action applied on mass i-th .

$G_i, G_j$ — representative values of gravity load concentrated at the masses of i-th and j-th respectively, which shall be determined by Clause 2.1.

$H_i, H_j$  - calculated height of i-th and j-th from the base of the building respectively.

$\delta_n$ — additional seismic action factors at the top of the building; for multi-story reinforced concrete buildings, it may be taken using Table 2.4; for multi-story brick buildings with inner-frames, a value of 0.2 may be used; no need to consider for other buildings

$\Delta F_n$  - additional horizontal seismic action applied at top of the building.

Table 2.4 Additional seismic action factors at top of the building

Table 2.4 - Additional seismic action factors at top of the building

<b>Tg (s)</b>	<b>T1 &gt; 1.4Tg</b>	<b>T1 ≤ 1.4Tg</b>
≤0.35	0.08T1+0.07	0.0
<0.35 ≈ 0.55	0.08T1+0.01	
>0.55	0.08T1-0.02	

**Note:** T, is the fundamental period of the structure.

The horizontal seismic shear force at each floor level of the structure shall comply with the requirement of the following equation:

$$V_{Eki} > \lambda \sum_{j=i}^n G_j \quad E2.7$$

Where:

$V_{Eki}$ - the floor i-th shear corresponding to horizontal seismic action characteristic value.

$\lambda$ - Shear factor, it shall not be less than values in Table 2.5; for the weak location of vertical irregular structure, these values shall be multiplied by the amplifying factor of 1.15.

$G_j$ - the representative value of gravity load in floor j-th of the structure.

Table 2.5 Minimum seismic shear factor value of the floor level

Structures	Intensity 7	Intensity 8	Intensity 9
structures with obvious torsion effect or fundamental period is less than 3.5s	0.16 (0.024)	0.032 (0.048)	0.064
Structures with fundamental period greater than 5.0s	0.012 (0.018)	0.024 (0.032)	0.040

**Note:**

1. The values may be selected through interpolation method for structures whose fundamental period is between 3.5s and 5s.
2. Values in the brackets are used at the regions with basic seismic acceleration as 0.15g and 0.30g respectively.

**Calculation of vertical seismic action:**

For tall buildings for Intensity 9, the characteristic value of vertical seismic action shall be determined by the following equations (figure 2.3). The effects of vertical seismic action at floor level may be distributed in proportion of representative value of gravity load acting on the members, which should multiply with the amplified factor 1.5:

$$F_{Evk} = \alpha_{vmax} G_{eq} \quad E2.8$$

$$F_{vi} = \frac{G_i H_i}{\sum G_j H_j} F_{Evk} \quad E2.9$$

Where:

$F_{Evk}$  - characteristic value of the total vertical seismic actions of the structure.

$F_v$  - characteristic value of vertical seismic action at the level of mass i-th.

$\alpha_{vmax}$ - maximum value of vertical seismic influence coefficient, which may be taken as 65% of the maximum value of the horizontal seismic influence coefficient.

Geq- equivalent total gravity load of the structure, which may be taken as 75% of the representative value of the total gravity load of the structure.

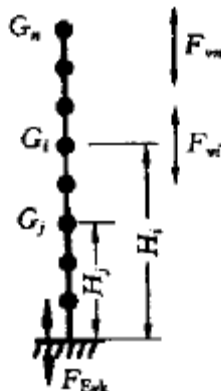


Figure 2.3 Sketch for the computation of vertical seismic action

### Complementarities:

- Structures having the oblique direction lateral-force-resisting members and the oblique angle to major orthogonal axes is greater than 150, the horizontal seismic action along the direction of each lateral-force-resisting member shall be considered respectively. So we could consider this though the item, the action of the oblique member could be multiplied by this factor as design force.
- Eccentricity: similar to UBC code. The eccentricity value of gravity center on each floor should be  $e_i = \pm 0.05L_i$ ,  
where:  
 $e_i$  - Eccentricity value of gravity center on i-th floor.  
 $L_i$  - maximum width of calculated story of the building.
- Structures having obviously asymmetric mass and stiffness distribution, the torsion effects caused by both two orthogonal horizontal direction seismic action shall be considered; and other structures, it is permitted that a simplified method, such as adjusting the seismic effects method, to consider their seismic torsion effects.

So we may use the to decide which whether torsion effects should be considered.

### General Format

The following general format should be used to generate loads in a particular direction.

DEFINE GB (ACCIDENTAL) LOAD

INTENSITY { 6 | 7 | 7A | 8 | 8A | 9 } { FREQUENT | RARE } GROUP { 1 | 2 | 3 }  
 } SCLASS { 1 | 2 | 3 | 4 } (DAMP  $f_1$ ) (DELN  $f_2$ ) (SF  $f_3$ ) (AV  $f_4$ ) (PX  $f_5$ ) (PZ  $f_6$ )

Where:

- INTENSITY is the Fortification Intensity (ref. table 5.1.4-1)
- Frequency of seismic action, as specified by either FREQUENT or RARE (ref. table 5.1.2-2)
- GROUP is Design Seismic Group (ref. table 5.1.4-2)
- SCLASS is Site-Class (ref. table 5.1.4-2)
- $f_1$  = damping ratio (default = 0.05 for 5% damping)
- $f_2 = \delta_n$ , Additional seismic action factor at the top of the building (default as calculated from Table 5.2.1)
- $f_3$  = Shear Factor  $\lambda$ , Minimum seismic shear factor of the floor (default as calculated from Table 5.2.5)
- $f_4$  = maximum value of vertical seismic influence coefficient  $\alpha_{(v,max)}$  (default=0.0) (ref. section 5.3)

As a fraction of total vertical load is to be considered such as 0.75G<sub>eq</sub>, specify the product of the factor on maximum horizontal seismic influence factor and factor of total gravity load as f<sub>4</sub>. For instance,

- $\alpha_{(v,max)} = 0.65\alpha_{max}$

And

- G<sub>v,eq</sub> = 0.75G<sub>eq</sub>
- Specify f<sub>4</sub> as (0.65\*0.75) i.e. equal to 0.4875
- f<sub>5</sub> and f<sub>6</sub> = optional time period along two horizontal direction (X and Y, respectively).

To apply the load in any load case, following command would be used

**LOAD CASE 1**

**GB LOAD X factor**

## Example

Example output:

```

*****
*****
*          *
* EQUIV. SEISMIC LOADS AS PER SEISMIC DESIGN CODE FOR BUILDINGS
*
*                                     (GB50011-2001) OF CHINA ALONG X
*
* T CALCULATED = 0.252 SEC.          T USER PROVIDED = 1.200 SEC.
*

```

```

* T USED = 1.200 SEC.
*
* MAX. HORIZONTAL SEISMIC INFLUENCE COEFFICIENT = 0.240
*
* CHARACTERISTIC PERIOD = 0.750 SEC.
*
* DAMPING RATIO = 0.030    POWER INDEX (GAMMA) = 0.931
*
* DAMPING ADJUSTMENT FACTOR (ETA2) = 1.180
*
* ADJUSTING FACTOR (ETA1) = 0.022
*
* HORIZONTAL SEISMIC INFLUENCE COEFFICIENT (ALPHA1) = 0.183
(18.288%)
*
* MINIMUM SHEAR FACTOR AS PER SEC. 5.2.5 (LAMBDA) = 0.050 (
5.000%)
*
* TOTAL HORIZONTAL SEISMIC ACTION =
*
*           = 0.183 X      285.529 =      52.218 KIP
*
* DESIGN BASE SHEAR = 0.750 X      52.218
*
*           =      39.164 KIP
*
* ADDITIONAL SEISMIC ACTION FACTOR (DELTAN) = 0.020
*
* VERTICAL SEISMIC INFLUENCE COEFFICIENT (ALPHA,VMAX) = -0.108
*
* TOTAL VERTICAL SEISMIC ACTION =
*
*           = -0.108 X      285.529 =      -30.837 KIP
*
* TOTAL DESIGN VERTICAL LOAD = 0.750 X      -30.837
*
*           =      -23.128 KIP
*

```

```

*
*
*****
*****
CHECK FOR MINIMUM LATERAL FORCE AT EACH FLOOR [GB50011-
2001:5.2.5]
LOAD - 1 FACTOR - 0.750
FLOOR          LATERAL          GRAVITY          LAMBDA  LAMBDA
ADJUSTMENT
HEIGHT (KIP )  LOAD (KIP )      LOAD (KIP )      (%)      MIN
(%)  FACTOR
-----
-----
30.000         23.406           79.045           29.61    5.00
1.00
20.000         19.208           180.888          10.62    5.00
1.00
10.000         9.604            282.731          3.40     5.00
1.47

JOINT          LATERAL          TORSIONAL          VERTICAL
LOAD - 1
LOAD (KIP )    MOMENT (KIP -FEET)  LOAD (KIP )      FACTOR -
0.750
-----
-----
17  FX         0.541  MY         0.000           FY         -
0.221
18  FX         0.663  MY         0.000           FY         -
0.271
19  FX         0.663  MY         0.000           FY         -
0.271
20  FX         0.541  MY         0.000           FY         -
0.221
21  FX         0.663  MY         0.000           FY         -
0.271
    
```

22	FX	0.785	MY	0.000	FY	-
0.321						
23	FX	0.785	MY	0.000	FY	-
0.321						
24	FX	0.663	MY	0.000	FY	-
0.271						
25	FX	0.663	MY	0.000	FY	-
0.271						
26	FX	0.785	MY	0.000	FY	-
0.321						
27	FX	0.785	MY	0.000	FY	-
0.321						
28	FX	0.663	MY	0.000	FY	-
0.271						
29	FX	0.541	MY	0.000	FY	-
0.221						
30	FX	0.663	MY	0.000	FY	-
0.271						
31	FX	0.663	MY	0.000	FY	-
0.271						
32	FX	0.541	MY	0.000	FY	-
0.221						
-----						
TOTAL =		10.602		0.000		-0.221
AT LEVEL		10.000 FEET				

## AD.2007-06.3 Features Affecting the RAM Connection Design Mode

### RAM Connection

Several new features have been added and existing features have been modified in the RAM Connection Design Mode. These are explained in the following pages.

**Note:** Full use of the RAM Connection Mode requires access to a valid RAM Connection license. If you do not possess a license, contact your Bentley account manager to have it added to your SELECT licenses. Without a valid license, only a small subset of the full range of available RAM connections can be utilized.

## AD.2007-06.3.1 RAM Connection V8i support

### Purpose

The enhancements included in Bentley's RAM Connection V8i (release 5.5) are now available in STAAD.Pro. Additionally, the connection assignment and design process has been streamlined within STAAD.Pro. Now, joints and connections can be automatically assigned for a set of selected members and connection designs can be grouped together. Additionally, when a connection is edited using the RAM Connection pad, those changes will be saved in the STAAD.Pro model design.

### Description

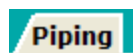
Connections are designed in the newly updated RAM Connection Mode by creating *Joints*, from the geometry, section properties and forces resulting from the analysis and assigning a design brief made up of connection templates. A suitable connection design, if one is available, will be reported once you have selected the appropriate connection templates.

In previous releases of STAAD.Pro, you were able to only select and assign a connection to individual joints. Now, any number of joints may be selected and designed. Further, connection design is performed automatically for you once appropriate templates have been selected for the selected joints. These enhancements greatly reduce the time required for connection design in models of all sizes.

**Hint:** The selected load envelope is now used for all connection designs, instead of per design brief as in previous versions of STAAD.Pro.

For additional information on using this mode, refer to the RAM Connection mode manual, found in the Additional Modules section of the STAAD.Pro Online Help.

## AD.2007-06.4 Features Affecting the Piping Mode



STAAD.Pro can utilize 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.2007-06.4.1 AutoPIPE Integration Enhancements

#### Overview

An enhancement to how pipe/structure connections are assigned has been made to the Piping module. A new Support Connection Wizard is available to allow you to add multiple supports to the entire model or a subset of a model, based on some general parameters.

## Description

Within the Piping mode, after the pipe model has been loaded, you will be able to call up a modeless connection wizard. The wizard will take you through the following steps

1. Defining the set of pipe nodes to consider.
2. Defining the set of structural beams to consider.
3. Defining the set of structural nodes to consider.
4. Setting range and tolerance parameters.
5. Previewing and accepting the determined connections.

Potential connections will be determined after step 4 and fully created at the end of step 5.

Potential connections will be determined by finding the closest beams and closest nodes to each pipe node. In the previewing stage the closest five items, along with a “to ground” option, will be available as options to you, sorted by distance.

### > Using the Support Connection Wizard

1. Run an Analysis.

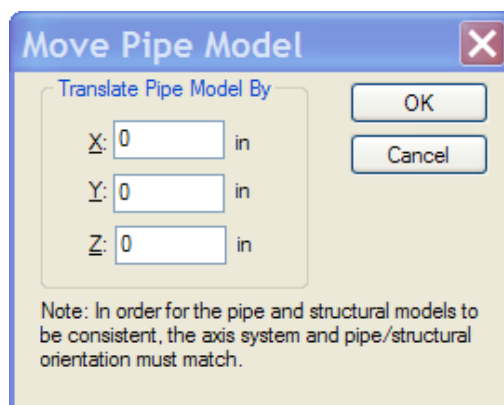
**Hint:** Piping mode is only available after the completion of a successful analysis.

2. Click on the Piping tab.
3. Start the Connection Wizard by clicking the Open File ... button in the right-hand panel. Select the Pipe Model file associated with this STAAD model.

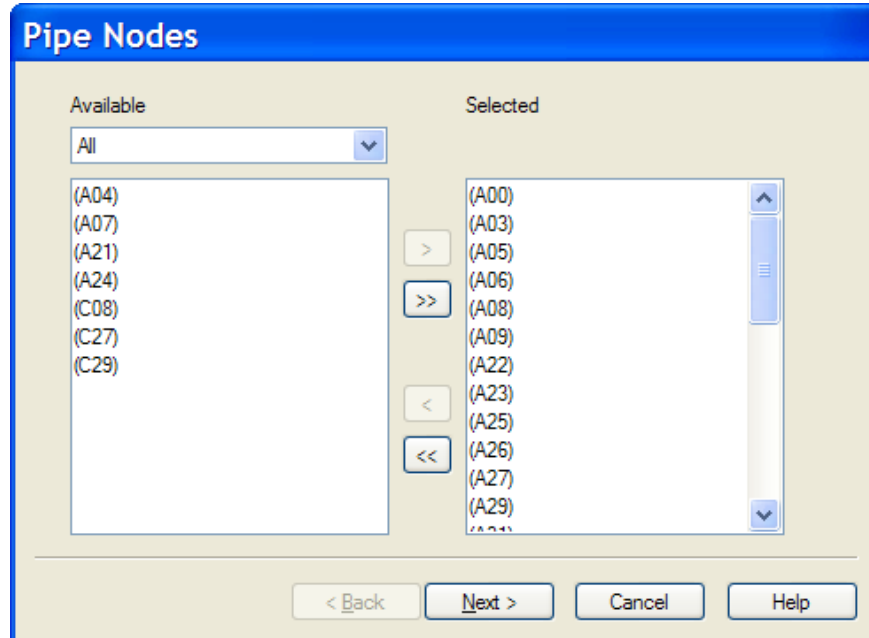
or

Select Support > Support Connection Wizard

4. The Move Pipe Model dialog opens. Enter the offset distance between pipe and structural models.

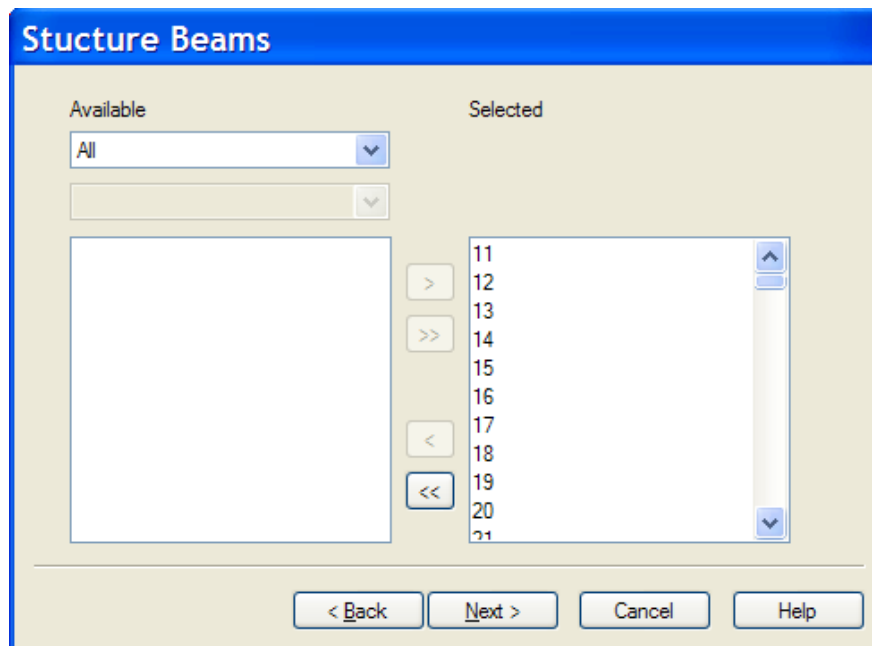


5. The Pipe Nodes dialog opens. Select which nodes will be used by adding them to the Selected list.



The filtering options are All, Connected and Unconnected (relating to whether a connection to the STAAD.Pro model has been defined). When support type information is available the filter will be expanded to include these as well.

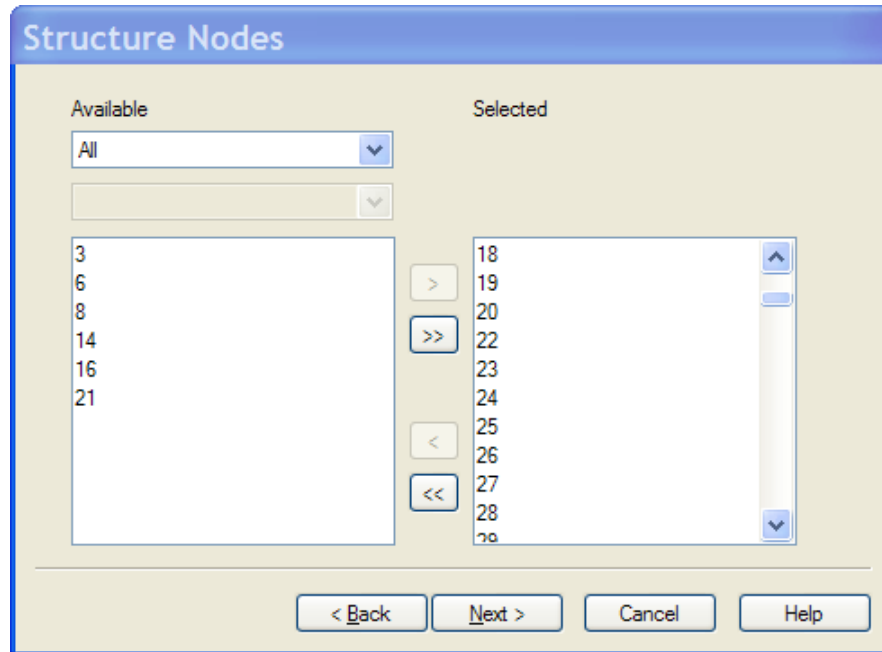
6. The Structure Beams dialog opens. Select which structure elements will be used by adding them to the Selected list.



The filtering options are implemented with two combo boxes, one for the category and one to identify the subset within that category. Available filtering options are All, Group, View and Property.

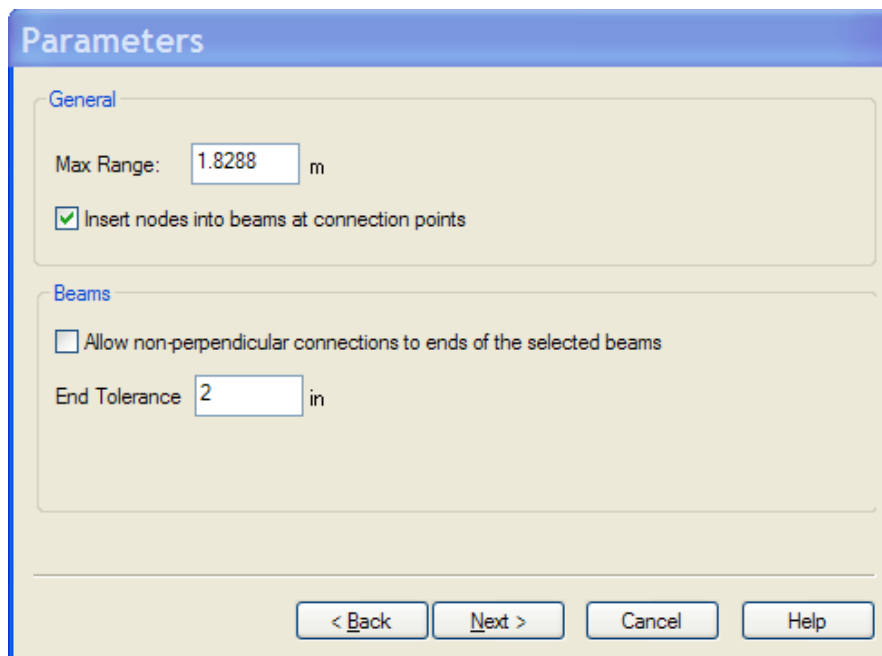
7. The Structure Node dialog opens. Select which structure nodes will be used by adding

them to the Selected list.



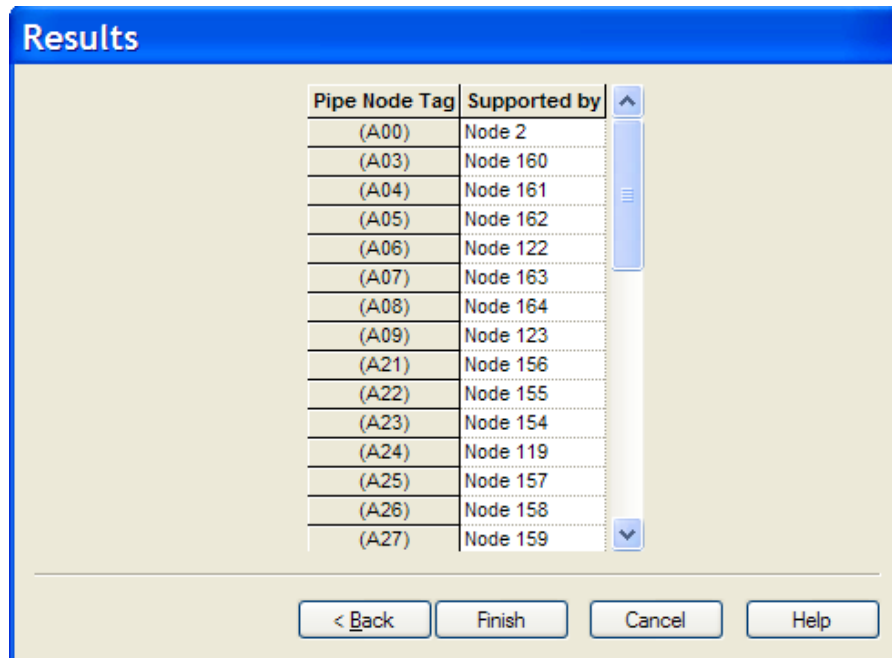
The filtering options are implemented with two combo boxes, one for the category and one to identify the subset within that category. Available filtering options are All, Group and View.

8. The Parameters dialog opens. Here you will set some parameters so the wizard can establish pipe connections.



See "Parameter Page / Connection Finder" on page 59 for additional details on these settings.

9. The results of the Wizard will be displayed in the final dialog. Review these results and if changes are necessary, you may click the < Back button. Otherwise, click the Finish button to accept.



The results will be presented in a table with columns for pipe node id and for the structural item it is to be connected to. This second column will provide a drop list allowing the user to choose either 'No connection' or one of up to 5 closest items. The items will be listed with the closest at the top. The initial state will be the closest item or 'Ground', if no good matches were found.

## Parameter Page / Connection Finder

The connection finding routine runs in two parts. First looking at beams (length of perpendicular from beam) and then looking at nodes (straight line length). The five closest found connectable points are saved to be presented on the results page. In order to provide control over the connection finder, several parameters are available to you for editing. The default values of distance-based parameters will depend on the base unit of STAAD.Pro.

### General

- Max. Range: double: default 2m / 6' : Potential connections beyond Max. Range will be discarded.
- Insert nodes into beams at connection points: default true : This parameter effects the connection of the structures rather the point finding algorithm. If set true a new node will be created at each intermediate beam point and the connection made to that rather than to the beam itself. If the algorithm finds new node points within "End Tolerance" of each other then only one new node will be added.

**Beam**

- End Tolerance : double : default 5cm /2” : To allow for differences in precision and to avoid very short beam breaks this parameter will determine at what distance from the node the perpendicular will be considered to be at the node itself.
- Allow Non-Perpendicular Connection at End Nodes: Boolean : default true: This is only really relevant if the node subset does not explicitly include the nodes at the end of members in the beam subset. If set ‘true’ the beam end nodes will be included in the node search. If set ‘false’ the end nodes of a given beam will only be considered for connections perpendicular to the beam, unless they have been explicitly added to the node subset.

This page has no effect on structure diagrams. The connection finding routine is run when advancing from this page.

**Note:** Pipe-structure links are not part of the undo system. Nodes created at the end of the wizard will be removed by an undo but the links are not changed.

**Export STAAD.Pro Structure to AutoPipe**

A piping engineer who needs to consider the steelwork as their structural supports may need to import the STAAD model into AutoPipe. A macro is now available in STAAD.Pro to facilitate this. It will create an .NTL file which is used by AutoPipe. The file will contain just the support frame data.

The Macro, called *TOAUTOPIPEPUB.VBS*, is located in the folder C:\SProV8i\STAAD\Plugins (where C:\SProV8i\ is the drive and folder where your copy of STAAD.Pro was installed).

## Section 4

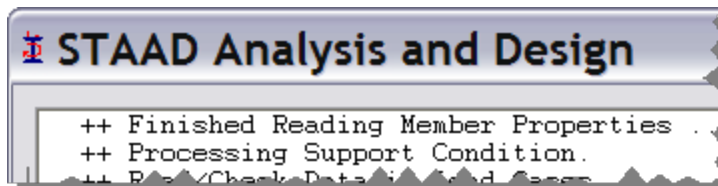
# STAAD.Pro V8i

---

### Introduction

The latest What's New document for STAAD.Pro V8i contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i (2007 build 04). This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

### AD.2007-05.1 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.2007-05.1.1 Geometric Nonlinear Analysis

### Purpose

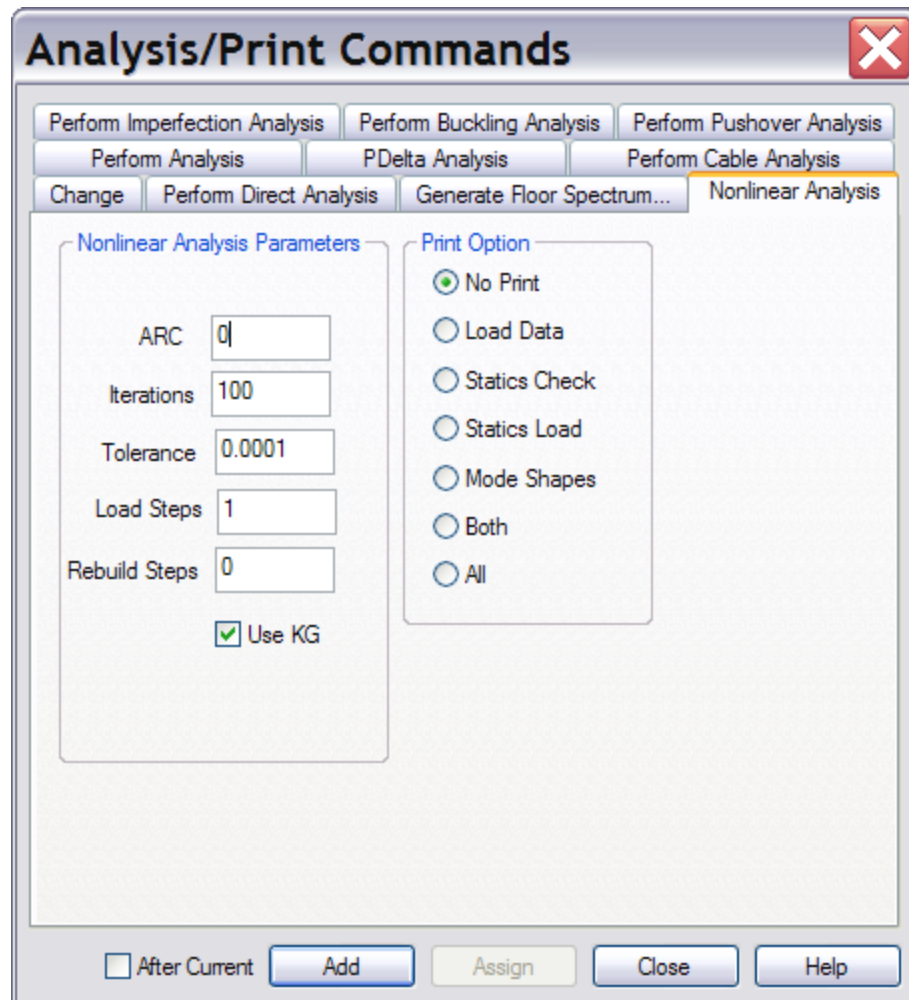
The range of analysis options has been supplemented with a new solution to account for nonlinear effects of moderate displacement and small strain. This solution holds for where the element distortion is small and small rotations are assumed.

### Description

**Note:** The nonlinear analysis command is secured with the Advanced Analysis Engine package.

To invoke a nonlinear analysis:

1. Select the Modeling | Analysis/Print page.
2. If the Analysis/Print Commands dialog does not appear, click the **Define Commands...** button.
3. Click on the Nonlinear Analysis tab.
4. Set the Nonlinear Analysis Parameters and Print Options (see table below).
5. Click the **Add** button to add the nonlinear analysis command to the model.
6. Click the **Close** button to dismiss the Analysis/ Print Commands dialog.



The first analysis step must be stable, otherwise use ARC control to prevent instability. The procedure does not use follower loads. Loads are evaluated at the joints before the first step; then those loads translate with the joint but do not rotate with the joint. Equilibrium is computed in the displaced position.

**Note:** The old NONLINEAR nn ANALYSIS command will adopt the new procedure unless a SET command is used. If a SET command is entered this will invoke the old procedure for backward compatibility.

## General Format

The format of the nonlinear analysis command is such that two data sets are required:

### 1) Initiate Command

The command is initiated with the line:

**PERFORM NONLINEAR ANALYSIS**

## 2) Specify Options

The following table describes the parameters available for nonlinear analysis:

Geometric Nonlinear Analysis parameters

Parameter Name	Default Value	Description
ARC	0.0	Displacement control. Value is the absolute displacement limit for the first analysis step. If max. displacement is greater than this limit, ARC will calculate a new step size for the first step and a new value for STEPS. Value should be in current length units.  ARC = 0 indicates no displacement control.
ITERATION	100	Max. Number of iterations to achieve equilibrium in the deformed position to the tolerance specified.
TOLERANCE	0.0001 inch	For convergence, two successive iteration results must have all displacements the same within this tolerance. Value entered is in current units.
STEPS	1	Number of load steps. Load is applied in stages if entered. One means that all of the load is applied in the first step.
REBUILD	1	Frequency of rebuilds of the tangent K matrix per load step & iteration.  0= once per load step 1= every load step & iteration

Parameter Name	Default Value	Description
KG	o	This parameter controls whether the geometric stiffness, KG, is added to the stiffness matrix, K. "KG" or "KG 1" -> Use K+KG for the stiffness matrix. (Default) "KG o" -> Do not use KG.
PRINT	none	Standard STAAD analysis print options. See Perform Analysis documentation.

Nonlinear entities such as tension/compression members, multilinear springs, gaps, etc. are not supported when using a nonlinear analysis. Additionally, nonlinear analysis does not account for post-buckling stiffness of members.

## AD.2007-05.1.2 IS 800:2007

### Purpose

The Indian Bureau of Standards has released a new version of the design code for the design of steel structures, known as IS 800:2007. This replaces the previous version of the code (also supported in STAAD.Pro), IS 800:1984, which was versified again in 1998. This is a new approach to steel design and is based on the limit state design rather than a stress based design of the old code.

The Steel Design section has been enhanced to include design per IS 800:2007. Both section checking and selection routines are supported.

### Description

The design will follow the same process as used by all other steel design codes currently available in the STAAD engine. The design of each member is controlled through a set of parameters that have been added to the GUI Steel Design Dialog under the title **IS 800:2007**.

The design engine will allow all standard section database sections and User Table sections to be designed.

The design process followed:

1. Check slenderness
2. Check classification
3. Check tension forces
4. Check compression forces
5. Check bending Forces
6. Check interaction

The results will be:

1. output to the *ANL* file
2. available in the member query
3. available in the post processing mode in the Design Results table

For the list of parameters and commands including the default values, please refer to the table below.

**Note:** The IS 800:2007 design is secured using the STAAD Indian Design Code Pack.

### To invoke the new IS 800:2007 code for steel member design

1. From the Steel Design tab, select **IS800 2007** from the current code list.
2. Click the **Define Parameters...** button to launch the Design Parameters dialog.
3. Set and add code parameter as required (see table below).
4. Click the **Close** button to dismiss the dialog.

### General Format

The format of the IS 800:2007 is as follows:

```
PARAMETERS  
CODE IS800 LSD  
KY 1.5 MEMB 3 7 TO 11  
PROFILE ISWB400 MEMB 1 2 23  
TRACK 1 ALL
```

## Design Parameters

### Indian Steel Design IS 800:2007 Parameters

Parameter Name	Default Value	Description
<u>ALPHA</u>	0.8	A Factor, based on the end-connection type, controlling the Rupture Strength of the Net Section  0.6 = For one or two bolts 0.7 = For three bolts 0.8 = For four or more bolts  (as per Section 6.3.3)
<u>ATG</u>	None (Mandatory for Block Shear check)	Minimum Gross Area in Tension from the bolt hole to the toe of the angle, end bolt line, perpendicular to the line of the force.  This parameter is applicable only when DBS = 1.0 (as per Section 6.4.1).
<u>ATN</u>	None (Mandatory for Block Shear check)	Minimum Net Area in Tension from the bolt hole to the toe of the angle, end bolt line, perpendicular to the line of the force.  This parameter is applicable only when DBS = 1.0 (as per Section 6.4.1).
<u>AVG</u>	None (Mandatory for Block Shear check)	Minimum Gross Area in shear along bolt line parallel to external force.  This parameter is applicable only when DBS = 1.0 (as per Section 6.4.1).
<u>AVN</u>	None (Mandatory for Block Shear check)	Minimum Net Area in shear along bolt line parallel to external force.  This parameter is applicable only when DBS = 1.0 (as per Section 6.4.1).
<u>BEAM</u>	1.0	0.0 = design at ends and those locations specified

Parameter Name	Default Value	Description
		by the SECTION command. 1.0 = design at ends and at every 1/12th point along member length (default).
<u>CAN</u>	0.0	Beam Type -  0.0 = non-cantilever beams for bending check and deflection check  1.0 = cantilever beam  (as per section 8.2.1.2)
<u>CMX</u>	0.9	Equivalent uniform moment factor for Lateral Torsional Buckling(as per Table 18, section 9.3.2.2)
<u>CMY</u> <u>CMZ</u>	0.9	Cm value in local Y & Z axes  (as per Section 9.3.2.2)
<u>DBS</u>	0.0	Check for Design against Block Shear:  0.0 = Design against Block Shear will NOT be performed.  1.0 = Design against Block Shear will be performed.  If DBS = 1.0, Non-Zero Positive values of <b>AVG</b> , <b>AVN</b> , <b>ATG</b> , and <b>ATN</b> must be supplied to calculate Block Shear Strength, Tdb.
<u>DFE</u>	None (Mandatory for deflection check)	"Deflection Length" / Maximum allowable local deflection.
<u>DJ1</u>	Start Joint of member	Joint No. denoting starting point for calculation of "Deflection Length".

<b>Parameter Name</b>	<b>Default Value</b>	<b>Description</b>
<u>DJ2</u>	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length".
<u>DMAX</u>	1000 in.	Maximum allowable depth.
<u>DMIN</u>	0.0 in.	Minimum allowable depth.
<u>FU</u>	420 MPA	Ultimate Tensile Strength of Steel in current units.
<u>FYLD</u>	250 MPA	Yield Strength of Steel in current units.
<u>KX</u>	1.0	Effective Length Factor for Lateral Torsional Buckling (as per Table-15, Section 8.3.1)
<u>KY</u>	1.0	K value in local Y-axis. Usually, the Minor Axis.
<u>KZ</u>	1.0	K value in local Z-axis. Usually, the Major Axis.
<u>LAT</u>	0.0	0.0 = Beam is laterally unsupported 1.0 = Beam is laterally supported (as per Section 8.2.1 and 8.2.2 respectively)
<u>LX</u>	Member Length	Effective Length for Lateral Torsional Buckling (as per Table-15, Section 8.3.1)
<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 Z-axis (Major).
<u>MAIN</u>	180	Allowable Slenderness Limit for Compression Member (as per Section 3.8)
<u>NSF</u>	1.0	Net Section Factor for Tension Member.
<u>TMAIN</u>	400	Allowable Slenderness Limit for Tension Member (as per Section 3.8)

Parameter Name	Default Value	Description
<u>PROFILE</u>	None	Used in member selection.  See Section 5.47.1 of the Technical Reference Manual for details.
<u>PSI</u>	1.0	Ratio of the Moments at the ends of the laterally unsupported length of the beam  0.8 = where Factored Applied Moment and Tension can vary independently  1.0 = For any other case.  (as per Section 9.3.2.1)
<u>RATIO</u>	1.0	Permissible ratio of the actual to allowable stresses.
<u>TRACK</u>	0	Controls the levels of detail to which results are reported.  0 = Minimum detail  1 = Intermediate detail level  2 = Maximum detail

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

## AD.2007-05.1.3 Eurocode 3 Includes National Annex

### Purpose

A number of countries that have signed up to the replace their current steel design standards with the Eurocode, EN 1993-1-1:2005, known commonly as Eurocode 3, have published their National Annex documents. These documents make small changes to the base document and STAAD.Pro has been updated to incorporate some of these National Annex documents. Currently, the Dutch and Norwegian National Annexes have been added to the STAAD.Pro engine.

### Description

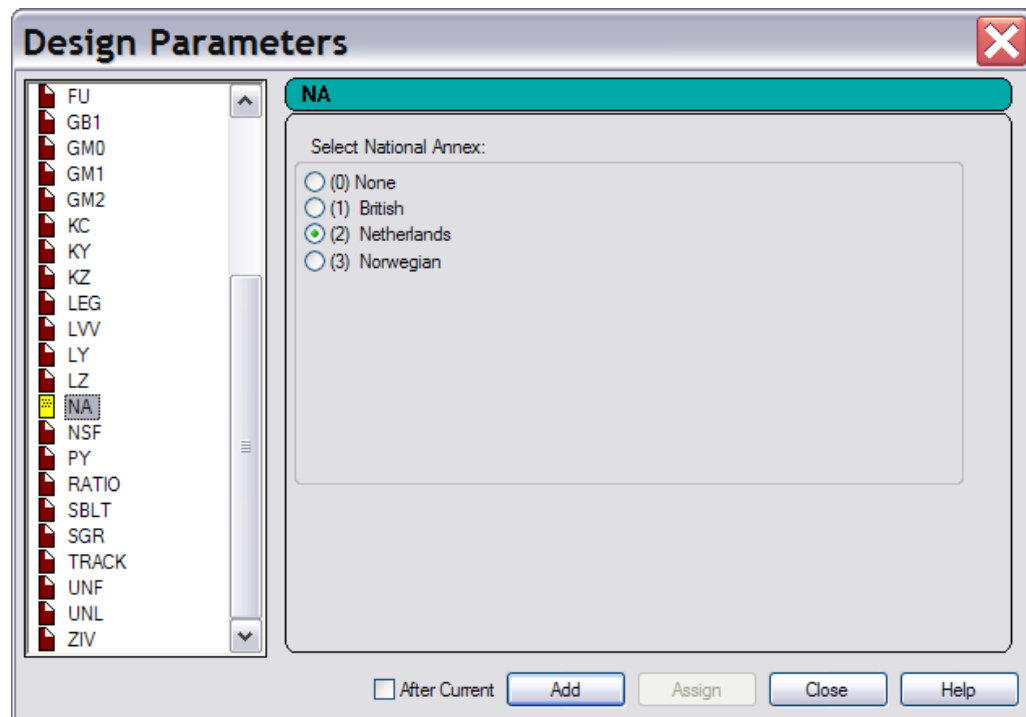
A new parameter, NA, that sets the default material gamma factors and any additional changes outlined in the country specific National Annex such as specific equations or methods.

The output file printout has been updated to indicate which National Annex (if any) has been used in a code check / select process. (For all TRACK settings)

**Note:** This Eurocode 3 design code is secured using the 'Eurocode Design' code pack.

In order to include additional check specified by a National Annex:

1. From the Modeling > Design > Steel tab, select **EN 1993-1-1:2005** from the current code list.
2. Click the Define Parameters... button to launch the Design Parameters dialog.
3. Select the parameter NA from the list.



4. Select the radio button for the National Annex you wish to use; or leave as Basic in order to use EC3 without additional checks.
5. Click the Add button to add the NA parameter to the code check.
6. Click Close to dismiss the dialog once parameter definitions are complete.

A design performed to the new Eurocode 3 National Annex is displayed in the output file (\*.ANL) with the following header, in addition to the base EC3 output:

ALL UNITS ARE - KN METE (UNLESS OTHERWISE NOTED)

MEMBER	TABLE	RESULT/ FX	CRITICAL COND/ MY	RATIO/ MZ	LOADING/ LOCATION
--------	-------	---------------	----------------------	--------------	----------------------

EC-6.2.5	0.723	1	0.0	50.0	0.0 50.0 0.0
EC-6.2.6-(Y)	0.284	1	0.0	50.0	0.0 50.0 0.0
EC-6.3.2 LTB	0.757	1	0.0	50.0	0.0 50.0 0.0

ADDITIONAL CHECKS AS PER NATIONAL ANNEX [NEN-EN 1993-1-1/NB] (units- kN,m):

EC CLAUSE	NA-CLAUSE	RATIO	LOAD	FX	VY	VZ	MZ	MY
EC-6.2.8-(Y)	NEN-6770-Eq.11.3.1	0.689	1	0.0	50.0	0.0	50.0	0.0
EC-6.2.10(Y)	NEN-6770-Eq.11.3.1	0.689	1	0.0	50.0	0.0	50.0	0.0
EC-6.2.10(C)	NEN-6770-Eq.11.3-31	0.550	1	0.0	50.0	0.0	50.0	0.0

Torsion and deflections have not been considered in the design.

**Note:** The previous, development edition of Eurocode 3 is included as the Code EN<sub>3</sub> DD.

## General Format

The format of the Eurocode 3 National Annex is as follows:

**CODE EN 1993-1-1:2005**

**NA f1**

{Code parameters: See Eurocode 3 in International Codes section}

Where: f1 represents the number designation for a specific country's National Annex:

Numerical code for Eurocode National Annex

NA Value	Country
0	None. This value represents using the base code only, with no national annex changes or additions (default).
1	British (currently inactive in STAAD.Pro)
2	Dutch
3	Norwegian

## AD.2007-05.1.4 Eurocode 8

### Purpose

Eurocode 8: Part 1 [EN 1998-1-1:2004] contains specific requirements and recommendations for building structures that are to be constructed in seismic regions. Essentially, these fundamental

requirements have been provided to ensure that the structures can sustain the seismic loads without collapse and also – where required – avoid suffering unacceptable damage and can continue to function after an exposure to a seismic event.

As with all Eurocodes, a National Annex Document should accompany the use of Eurocode 8 in each of the European nations.

## Description

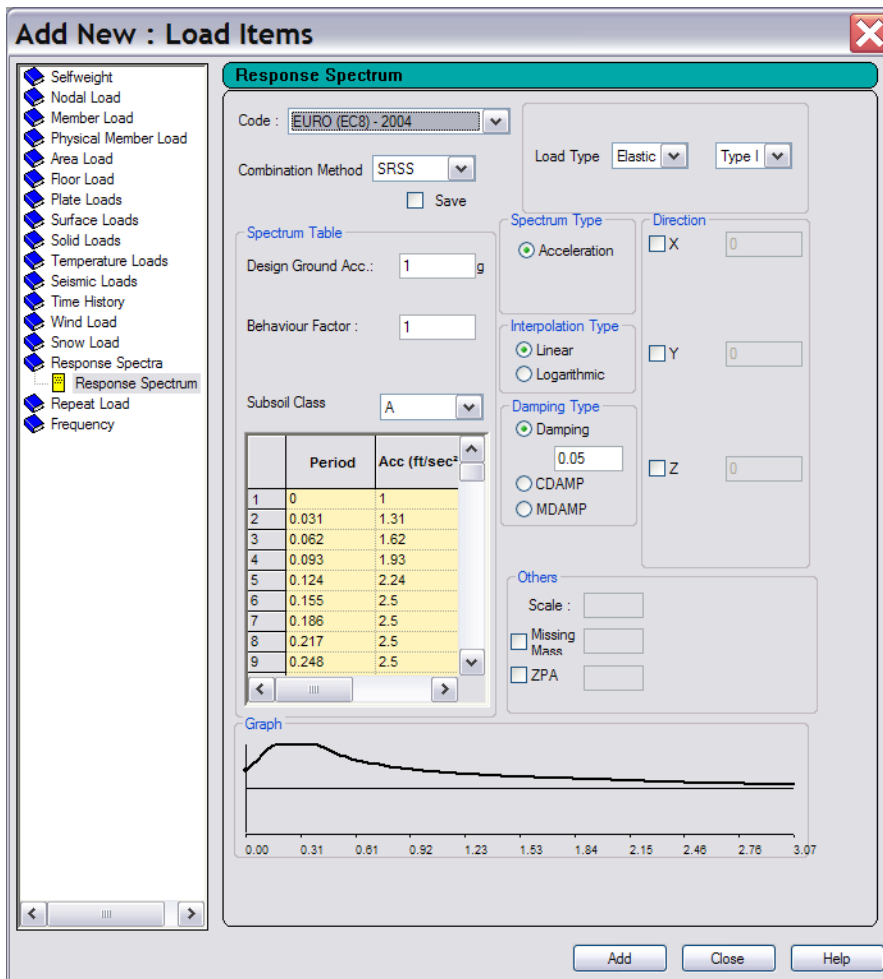
### a) Structure Modeling

To perform the geometry checks per Eurocode 8, STAAD.Pro must first calculate the center of mass for each defined floor. Specifying a response spectrum loading accomplishes this.

**Note:** The EC8 response spectrum load must be specified in the model in order to perform any further checks, design, or detailing per Eurocode 8.

From the Modeling > General > Load & Definition tab:

1. Select the load case where the EC8 seismic response spectrum definition will be added.
2. Click Add... to open the Add New Load Items box.
3. Select Response Spectra > Response Spectrum in the Load Items list.
4. Choose EURO (EC8) - 2004 from the Code list.
5. Set Response Spectrum options.
6. Click Add to add the new response spectrum to the current load case.



See also 5.32.10.1.2 Response **Spectrum** Specification per Eurocode 8.

**Note:** Floor height is determined by the program as a joint where one or more beams frame in a column.

## b) Analysis

All the analysis methods for design and evaluation of the performance of a structure mentioned in Eurocode, as listed in the following, have already been implemented in STAAD.Pro:

- Linear static (called “Lateral force method”)
- Linear modal response spectrum analysis
- Non-linear static analysis (push-over analysis)
- Non-linear dynamic analysis (Time-history analysis)

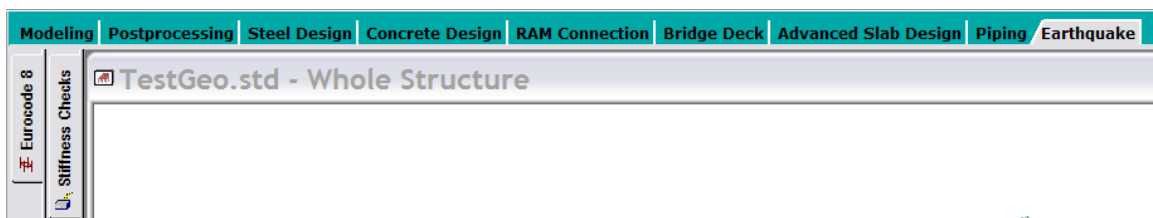
**Note:** In order to make use of the EC8 response spectrum load, a linear modal response spectrum analysis should be used. This is required to locate the center of mass which is needed to perform the associated geometry checks.

Loads are distributed by element stiffness to all members during analysis. Thus, for the purposes of Eurocode 8, all members are assumed to be "primary seismic" elements.

**Note:** Eurocode 8 requires the use of cracked section stiffness when considering concrete buildings, which however is lacking in the current analysis engine. This can be overcome by using a section reduction factor as suggested in the code. The current version of STAAD.Pro has a "section reduction factor" that can be used for this purpose.

## c) Post-Processing

### New Post-Processing Mode "Earthquake" Added



The "Earthquake" mode has been provided to allow the user to check if the structure conforms to the basic geometric recommendations made in EC8. This mode is in addition to the normal post processing mode which gives the various analysis results. These checks are intended to give the user a 'feel' for the structure and are not mandatory to proceed to the design phase.

This mode will be activated only if the analysis has been successful and if user has the right privileges to access the EC8 module.

**Note:** The program assumes y as vertical and that seismic forces act in the global x and z directions.

These Earthquake > Eurocode 8 checks have been further classified into the following tabs:

- Stiffness Checks - This set of checks is to enable you to judge whether the stiffness characteristics of the structure meet the recommendations set by EC 8. This implementation will:
  - i. Display the center of mass and center of stiffness graphically for each floor to inform you of some inherent eccentricities and their variation along the height of the building.
  - ii. Calculate the total stiffness of every story in each direction and provide you with a ratio of the stiffnesses in the two directions. This aids you to judge whether the structure conforms to the condition in EC8 that the stiffness should be similar in both directions.
  - iii. Check to see if there are any soft stories in the structure and will highlight them in the GUI. A soft story is defined as a story that has a stiffness in a particular direction that is less than 70% of the stiffness of the story above.

**Note:** The stiffness of only the vertical elements are considered while working out the total stiffness of a storey in a particular direction as the seismic loads are primarily resisted by the vertical elements in a structure.

- Plan regularity - This set of checks is to enable you to be able to classify the building as being 'regular in plan.' Checks will be performed and the results will be displayed graphically in the GUI. The program primarily checks for the main three plan regularity conditions set out in EC-8 viz.
  - i. Checks for re-entrant corners in floor slabs – The program performs checks on each floor slab and reports whether the condition set out in EC-8 for re-entrant corners has been satisfied.
  - ii. Check for slenderness of structure – The program calculates a 'slenderness ratio' for the structure based on the maximum dimensions on plan and reports whether the EC-8 criteria are satisfied.
  - iii. Checks for torsional radius: - The program calculates the torsional radius for each floor and checks against the conditions set out in EC-8.
- Elevation Regularity - This set of checks is to enable the user to be able to classify the building as a 'regular in elevation' structure. Checks will be performed and the results will be displayed graphically in the GUI.

## d) Member Design

### Perform basic design per EC2

A preliminary design per Eurocode 2 must be made for all concrete members prior to detailing and design per Eurocode 8. Once this is completed, click on the Earthquake tab in the RC Designer module. The Collapse Check Setup dialog will launch.

### Seismic Design and Detailing per EC8

The screenshot shows the 'Collapse Check Setup' dialog box with the following settings:

- Design Code: EC8 - 2004
- National Annex: UK Annex
- Ductility level: DCM
- Earthquake load case: L1: LOAD CASE 1
- Structural Type: Framed / Dual / Coupled wall system
- Check reinforcement detailing rules for ductility:
- Curvature ductility factor: 1.0

Buttons: Check Now, Cancel

1. With EC8 - 2004 selected in the Design Code list, specify the National Annex. Currently, only the UK Annex is available.
2. Specify Medium (DCM) or High (DCH) in the Ductility level list. Low ductility (DCL) does not require additional Eurocode 8 checks and therefore is not included.
3. Specify the load case containing seismic loads for use with design checks.
4. Specify the Structural Type in the list. Currently, only Framed/ Dual/Coupled Wall Systems are supported in RC Designer.
5. Select if you wish to Check reinforcement for detailing rules for ductility per EC8.
6. Specify a Curvature ductility factor to be used for column checks. If not specified, the default value is taken as unity (1).
7. Click Check Now to perform all design and detailing checks.

In order for EC8 checks to be performed, a member must have passed all EC2 checks in the initial design step. To also help identify issues in design or detailing, if one step of checks fails for a member, further checks will not be performed on that member.

The program will first check that all materials in the design brief are satisfactory. Then, moment capacity of beams and columns are evaluated and compared. EC8 stipulates that columns must have a moment capacity of 1.30 times the sum of the moment capacity of beams framing at that joint. This ensures beams are the initial failure mode.

**Note:** Deflection checks are performed for individual members only.

## AD.2007-05.1.5 AIJ Concrete Design Update

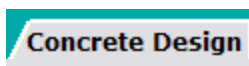
### Purpose

An old version of Japanese concrete code based on the AIJ standard for structural calculation of Reinforced Concrete Structures (1985 edition) was previously implemented in STAAD Pro batch mode design. Recently, critical errors in the column design and beam design were discovered and corrected. In addition, the AIJ concrete design has been updated to incorporate the latest AIJ standard for structural calculation of Reinforced Concrete Structures (1991 edition).

### Description

Generally, this implementation will be invisible to the user. The 1985 edition of the AIJ has been completely replaced by the 1991 edition for the design of concrete members per the AIJ. The existing commands as described in the International Codes section of this document are applicable to the 1991 edition.

## AD.2007-05.2 Features Affecting the Concrete Design Mode



An existing feature has been modified in the RC Designer section of the program, also known as the Concrete Design mode. This is explained in the following pages.

## **AD.2007-05.2.1 RC Designer Member and Envelope Import**

### **Purpose**

The ability to create physical members in STAAD.Pro has been introduced since the original implementation of the RC Designer. Previously these members did not get carried through into the RC Designer, thus requiring the extra step of re-creating these members, and possible confusion if the same member numbers were not used.

Similarly, load case/combination envelopes were not carried through into the RC Designer, requiring extra care in identifying which load cases were being considered when examining the output.

The physical member definitions are now passed through to the RC Designer along with the other structural data from STAAD.Pro. Envelope definitions are also carried through and examined to see if they contain any dynamic loading for which it may be inappropriate to use the analysis results directly.

### **Description**

Generally, this implementation is invisible to the user. Physical member assignments made in the STAAD.Pro Modeling mode will be carried over to the RC Designer (Concrete Design mode). If existing designs are re-examined, the user may be warned if member numbers or constitution (in terms of beam elements included) differ between the STAAD.Pro Modeling mode and the RC Designer definitions, and the option is given to delete the RC Designer members with their results. For envelopes, the user is prompted if the envelope numbers or constitution differ between RC Designer and STAAD.Pro Modeling mode.

Additionally, a warning may be presented to the user if any envelope incorporates a dynamic load case. This is because the dynamic load cases in STAAD.Pro only produce positive values for moments and forces, even though it is understood that the loading actually reverses. Concrete design based on an envelope of positive forces only would be seriously under-reinforced.

## Section 5

# STAAD.Pro V8i(release 20.07.04)

---

## Introduction

The Software Release Report for STAAD.Pro V8i contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2007 build 03. This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

## AD.V8i.0 New Features Affecting the General Program



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

---

## AD.V8i.0 New Features Affecting the General Program

### AD.V8i.0.1 ProjectWise Integration

ProjectWise is an engineering project team collaboration system which is used to help teams improve quality, reduce rework, and meet project deadlines. One of the major pieces of functionality provided by ProjectWise is an Integration Server which allows data to be managed and shared across a distributed enterprise.

STAAD.Pro has been enhanced so that the model STD data file can be managed on a ProjectWise server.

#### Description

Four integration functionalities have been added. These are

- Open a STAAD model from a ProjectWise repository.
- Save a local STAAD model into a ProjectWise repository.
- Update an existing model from ProjectWise.
- Review model properties (meta-data) which has been opened from a ProjectWise repository.

Note that access to all of these functionalities is available from ProjectWise sub-menu under the general File menu described below.

#### ProjectWise repository

Installation and management of a ProjectWise server is beyond the scope of this document and should be obtained from the ProjectWise installation.

#### ProjectWise client

A local ProjectWise client should be installed which allows access to ProjectWise repositories.

#### STAAD.Pro

When STAAD.Pro is launched, the option to open and check out a STAAD.Pro STD file from a ProjectWise repository is made available from the Project Tasks on the Start Page thus:-



This is also available from the File menu while still on the Start Page prior to opening a model:-



If a suitable ProjectWise client is not installed, then the link on the Start Page is shown as unavailable with a red line through the icon thus:-



### Login

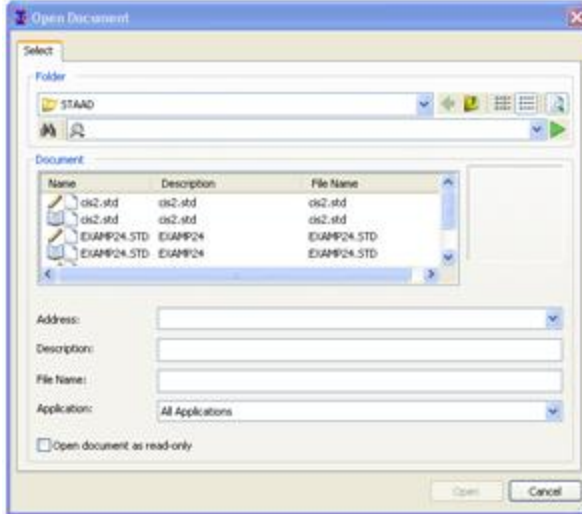
As authentication is required to access files stored on a ProjectWise repository, a login dialog allows the required details to be entered either with specific user credentials or by using the current windows login credentials thus:-



Files that are accessed from a ProjectWise server are 'Checked Out' and stored locally during the STAAD.Pro session until the file is closed and then it is returned to the server.

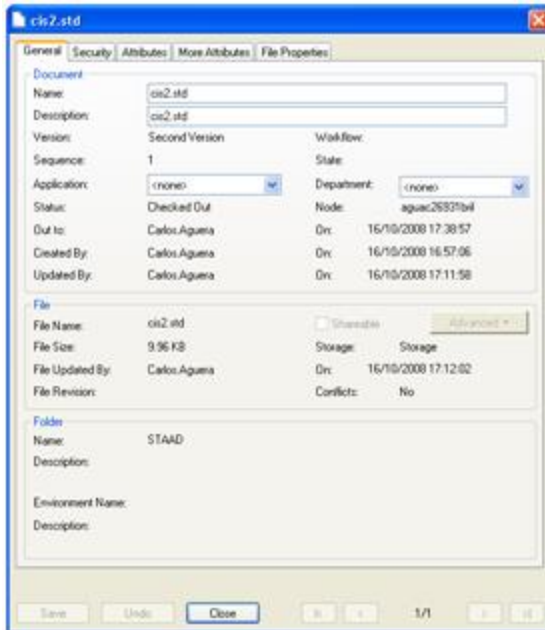
### Opening a STAAD Model from a ProjectWise repository

The first time that a successful link to a ProjectWise server is established, a location in which check out files are to be stored locally and additionally, where all the auxiliary data files are stored whilst STAAD is running is required. Afterwards and on all future occasions, the ProjectWise open dialog presented is then presented where the repository can be navigated and filtered as defined in the ProjectWise documentation.



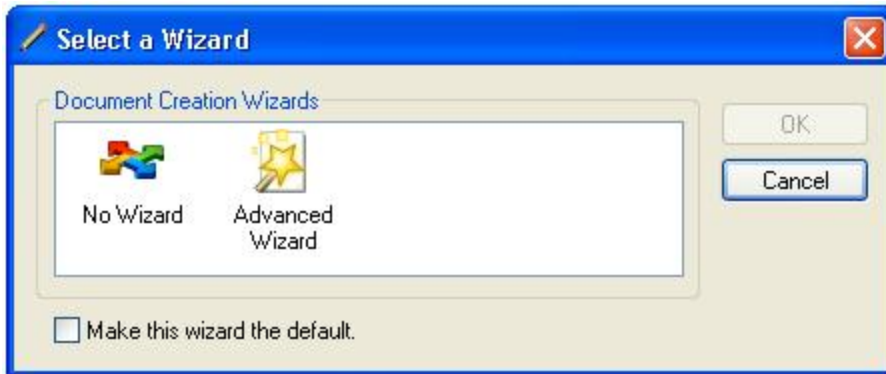
Note the significance of the icons next to the STAD filenames. These indicate the status of the file such as the current document, checked out to you, or locked as checked out to some other user. Refer to the ProjectWise documentation for a full description of each icon.

With a file checked out and loaded in STAAD.Pro, it is possible to see the ProjectWise Properties, by selecting the option from the ProjectWise toolbar or File menu:-

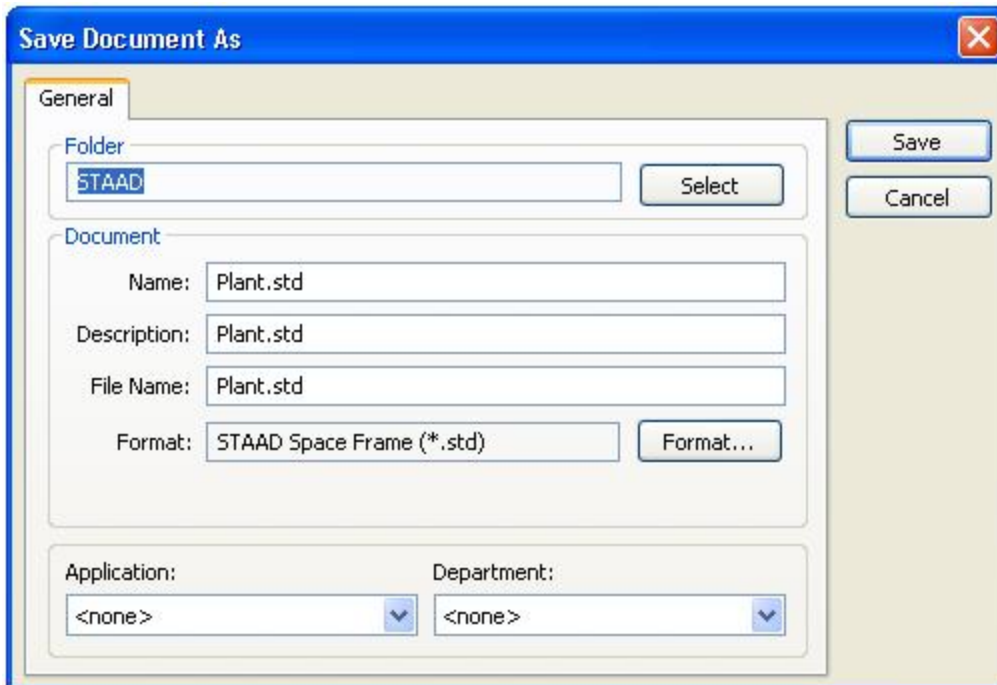


### Saving a New model onto the Server

If working on a STAAD model that has not originated from a ProjectWise server (e.g. starting a new file) and it has been decided that it needs to be added to a repository, then at any time whilst working in the STAAD.Pro environment, clicking on the toolbar option Add to ProjectWise server, or equivalent File>ProjectWise menu option will launch the following dialog:-



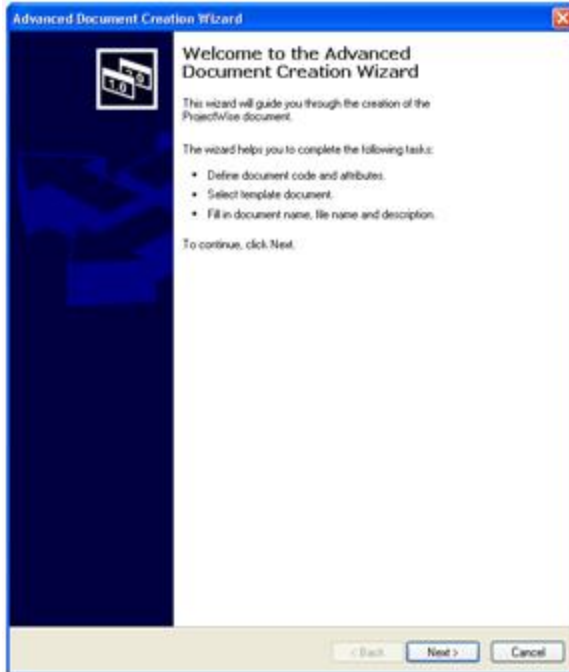
Selecting the option **No Wizard** offers the following dialog into which the file details can be entered.



Clicking on Save, adds this model to the repository, but indicates its status as checked out until the file is closed in STAAD.Pro and the model checked back in.

Alternatively, by selecting the **Advanced Wizard** option the data needed to define the ProjectWise data file is presented in the following four steps:-

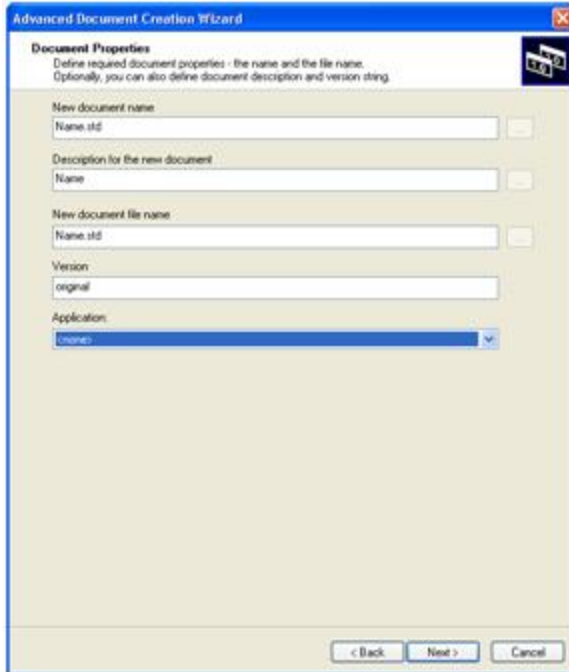
#### **Advanced Wizard**



### Select target Folder



### Document Properties



### Create the document



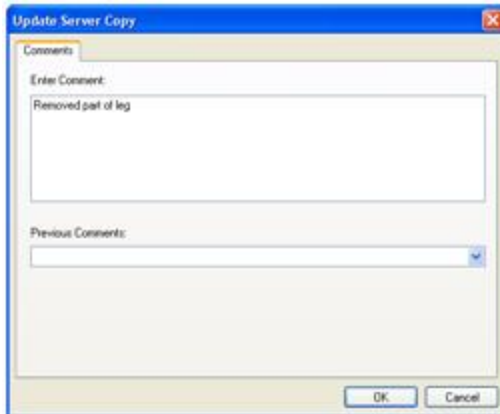
Creates the document and marks it as checked out until the file is closed and checked in to the server.

### Saving changes of a Checked Out model Back on the Server

When a model is checked out from a ProjectWise server, selecting Save or Save As, only maintains a local copy of the model. There are two methods available to update a checked out model. Firstly, during a STAAD.Pro session, it is possible at any time to save any changes back on the server by selecting the Update Server Copy icon from the ProjectWise toolbar or from the File>ProjectWise menu.

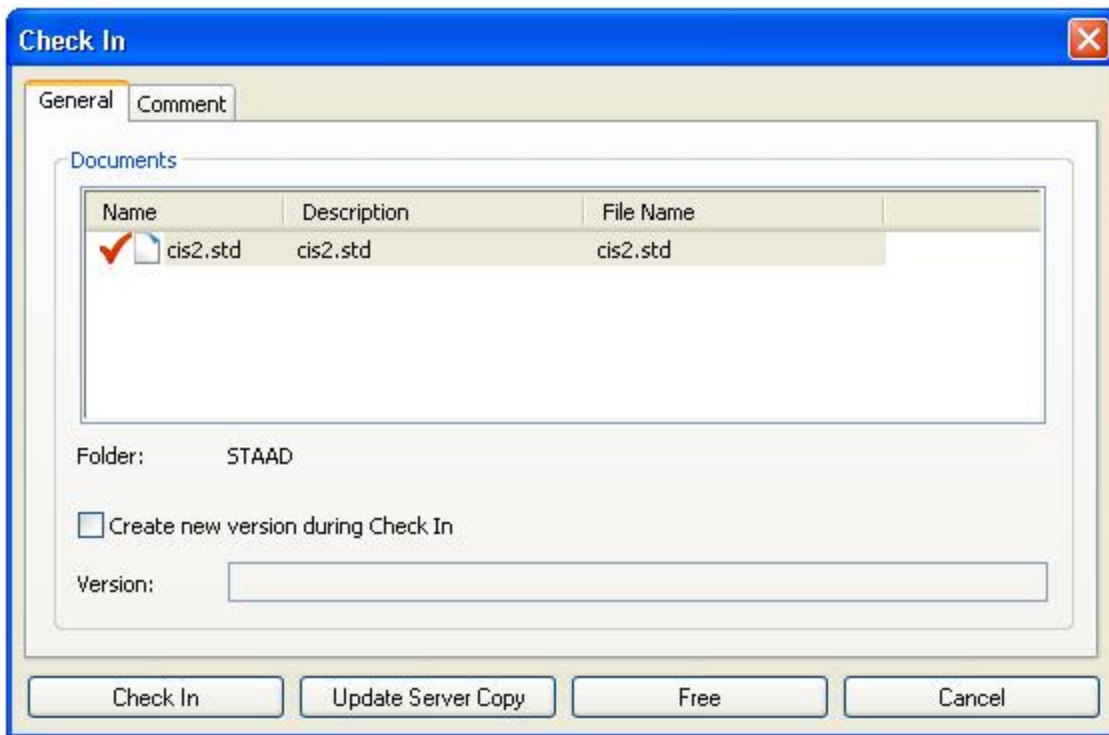
---

First save updates to the file locally. If not, then this will be prompted. Then the following dialog is displayed which allows a comment to be added to this model, thus:-



The file remains checked out and can be continued to be worked on.

The second method is automatically generated when a checked out model is closed. This launches the Check In dialog that displays the file(s) that are to be checked in and provides four actions:-



### Check In

This copies the local version of the STAAD model back to the server and releases it so that it is available for others to modify. If the option Create New Version is selected, then a new copy of the file is created on the server which becomes the current version of the model. The status of the checked out model is changed to read only on the server and can be used as a reference to a stage in the development of the model.

### Update Server Copy

---

This updates the model on the server with the current local model, but does not change its status which remains as checked out.

### Free

This changes the status of the model as checked in which allows other users to take control and check out the model, but does not update the server model. Thus local changes will be discarded

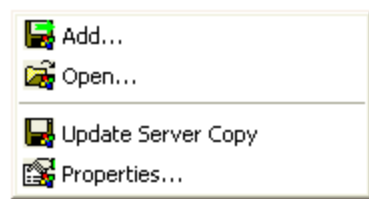
### Cancel

This cancels any change to the status of the model or the model itself on the server.

## Menu

The commands that drive the ProjectWise integration are defined in the main File menu thus:-

File>ProjectWise>



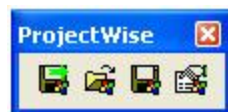
The **Add...** command is available when using a locally opened (not checked out from ProjectWise), which allows the current model to be saved into a ProjectWise repository.

The **Open...** command is available when a suitable ProjectWise client has been installed to allow access to a repository from which to check out a STD file.

The **Update Server Copy** command is available when working on a checked out file and the changes made on the model can be

## Toolbar

A new toolbar named ProjectWise has been added that duplicates the commands from the File>ProjectWise menu, thus:-



## Notes:

1. For more details on ProjectWise refer to the ProjectWise client installation documentation.
2. This functionality requires access to a version V8i or greater of ProjectWise.

## AD.V8i.0 New Features Affecting the General Program

### AD.V8i.0.2 CIS/2 Update

The STAAD.Pro tool to import and export models with the CIS/2 translator has been enhanced to work with international models and transferring models into 3D modeling such as SmartPlant 3D.

---

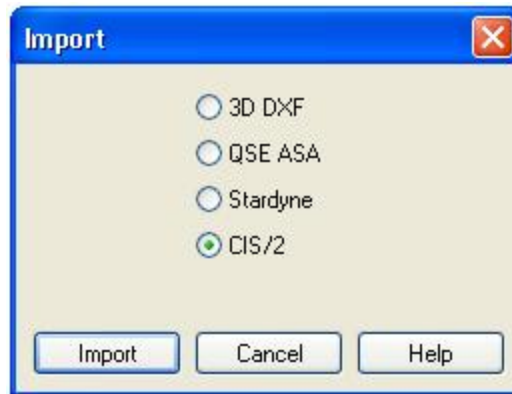
## Description

The CIS/2 (CimSteel Integration Standard, Version 2) allows for the transfer of steel models using a prescribed data standard in the STEP (Part 21) format. These files can contain different models including analysis models. The STAAD.Pro CIS/2 translator only operates on the analysis model within the file. Whilst STAAD.Pro has supported a wide range of the steel sections that this standard can support, this enhancement allows a far greater range of sections to be imported/exported with this tool.

## Import

The CIS/2 import can be initiated after starting a new model and before creating any model data, selecting the menu option,

File>Import...



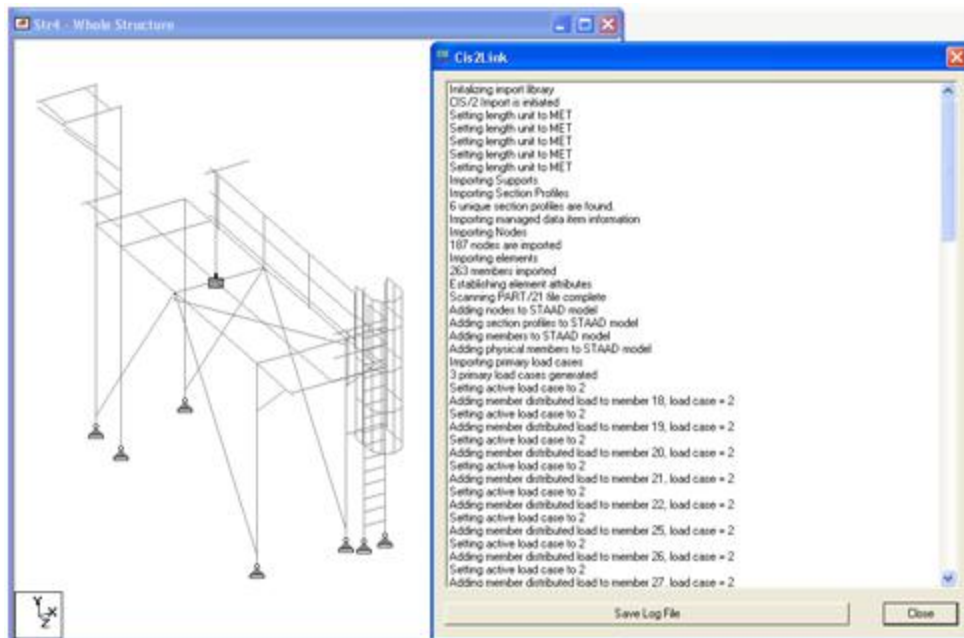
Selecting the CIS/2 option and clicking on the 'Import' button allows selection of a suitable STEP file. This file is a text file and will be similar to the following extract:

```
ISO-10303-21;
HEADER;
/* Generated by software containing ST-Developer
* from STEP Tools, Inc. (www.steptools.com)
*/
FILE_DESCRIPTION(
/* description */ ('CIS2 Export File'),
/* implementation_level */ '2;1');
FILE_NAME(
/* name */ 'model',
/* time_stamp */ '2007-01-08T17:00:33+00:00',
/* author */ (''),
/* organization */ (''),
/* preprocessor_version */ 'ST-DEVELOPER v9',
/* originating_system */ 'STAAD.Pro 2007',
/* authorisation */ '');
FILE_SCHEMA (('STRUCTURAL_FRAME_SCHEMA'));
ENDSEC;
DATA;
#10=LOAD_COMBINATION_OCCURRENCE(1.39999997615814,#13,#264);
#11=LOAD_COMBINATION_OCCURRENCE(1.39999997615814,#13,#265);
#12=LOAD_COMBINATION_OCCURRENCE(1.60000002384186,#13,#266);
#13=LOADING_COMBINATION('ULS',$,#2511);
#14=LOAD_ELEMENT_DISTRIBUTED_CURVE_LINE(#265,'Member19UDLLoad1',$,#1089,
$,$.F.,.GLOBAL_LOAD.,.TRUE_LENGTH.,#214,#214,#64);
#15=LOAD_ELEMENT_DISTRIBUTED_CURVE_LINE(#265,'Member20UDLLoad1',$,#1090,
```

Once a file has been selected, the data is ready to be imported with the CIS/2 import tool thus:-



Clicking the Import button then runs the model import where the CIS/2 file is processed and the analysis model data is extracted to form the STAAD.Pro model.



Note that as the CIS/2 file is being processed, a log file which identifies the data that has been utilized in the STAAD model is produced and displayed which can be saved as a text file for future reference.

## Export

The CIS/2 Export is available for any model that has been created as an option in the menu item File>Export...



Selecting the CIS/2 option and clicking on the 'Export' button allows the definition of a suitable STEP filename and folder to locate it. Once again the CIS/2 tool is presented, this time with an Export Model button at the bottom which when clicked creates the STP file.



Once again, when the export has completed, the user can save the log file which is produced as the model is converted into the STEP format.



## Enhancements

The STAAD.Pro CIS/2 import now recognizes sections defined from standard databases such as those defined in Japanese, British, Indian, Australian and European tables.

The import/export has been enhanced to support the ability of STAAD.Pro to create double sections, such as back to back angles and channels or double wide flange sections. Additionally, it now supports the creation of T sections which are defined in STAAD.Pro as a wide flange section that is split at mid height.

Although CIS/2 has been developed for the processing of steel models, the STAAD.Pro translator will now support the transfer of prismatic properties normally associated with concrete sections. This means that sections defined as PRISMATIC in a STAAD.Pro model will be included in exported STP file and can be imported if they exist.

## AD.V8i.1 Features Affecting the Pre-Processor (Modeling 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 Modeling Mode. These are explained in the following pages.

### AD.V8i.1 Features Affecting the Pre-Processor (Modeling Mode)

#### AD.V8i.1.1 ASME NF Steel Design Codes

The design of steel sections according to the requirements in the American Society of Mechanical Engineers (ASME) specifications, Rules for the Construction of Nuclear Power Plant Components, Section III – Subsection NF has been implemented and the steel design page has been updated to allow the design parameters to be defined and assigned.

The design requirements for the following years have been added:-

- 1974
- 1977
- 1989
- 1998

Post analysis steel design code checking requirements for the required ASME NF code can be selected by entering the Design>Steel page and setting the ASME code / year in the Current Code option in the Steel Design dialog:-



The method for selecting design parameters and assigning them to the members to change them from the default values is exactly the same as for all other steel design codes. Additionally, choosing members that are to be checked or selected for maximum utilization follows exactly the same method as for all other steel design codes.

For more information on the technical requirements of this design code, including the full set of parameters and default values, see the new section in the International Design Codes manual on the ASME NF design codes.

**Note:** In order to run a design check to any of the ASME NF design codes, then access to a STAAD Nuclear Code pack will be required.

## AD.V8i.1 Features Affecting the Pre-Processor (Modeling Mode)

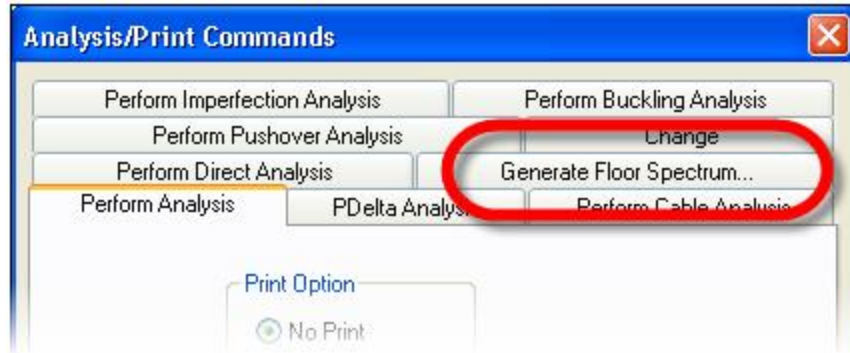
### AD.V8i.1.2 Floor Response Spectrum

A new dynamic feature has been added that allows the extraction of a response spectrum from a collection of nodes that constitute a floor when subjected to a time history loading. This information can then be used in conjunction with equipment that will be supported by these floors and is often required by the equipment manufacturers.

Users will require a license for the advanced analysis module to access this feature.

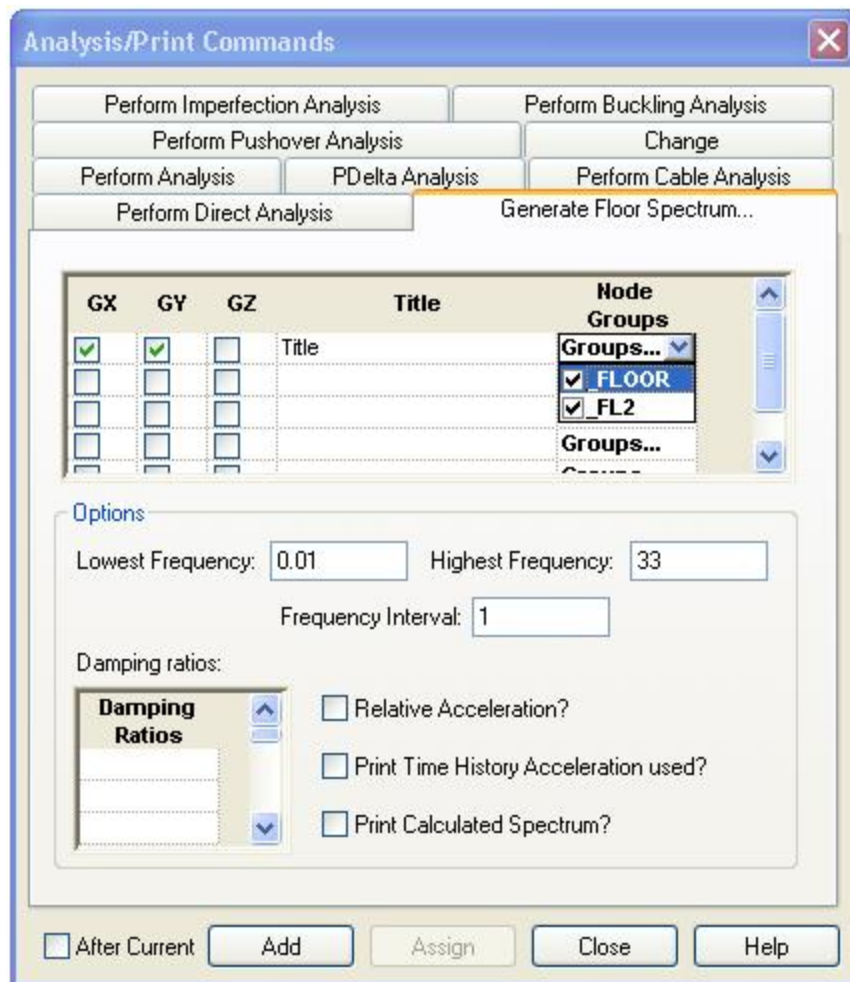
#### Description

The required commands (see section AD.V8i.2.3 Floor Response Spectrum for more information) can be entered graphically after adding the analysis command by selecting the new analysis sheet 'Generate Floor Spectrum' thus:-



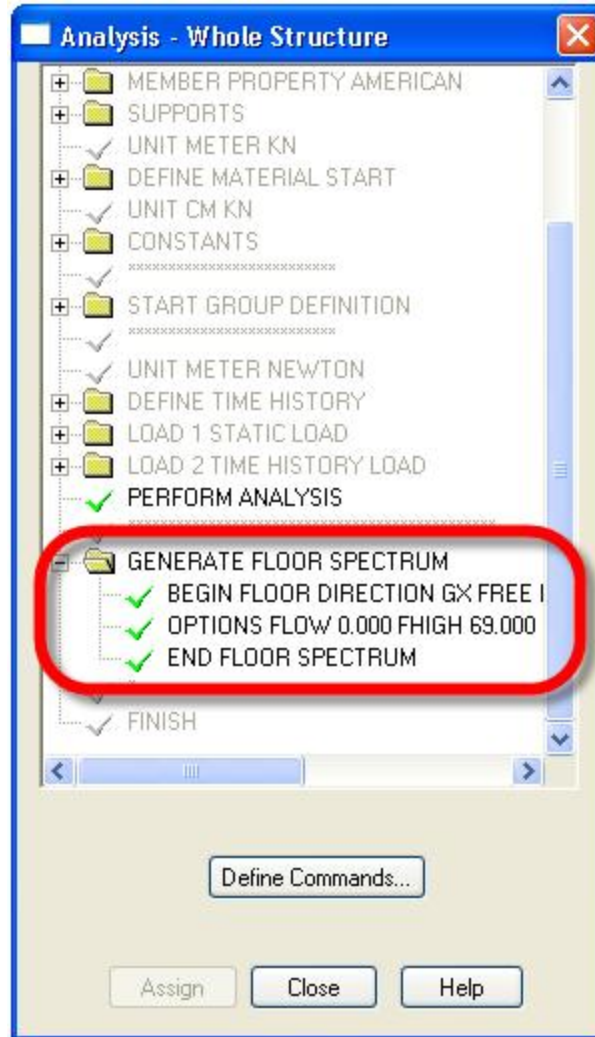
This command should follow immediately the definition of the analysis and will require defined groups of nodes which need to be defined first.

The following displays the layout of this sheet:-

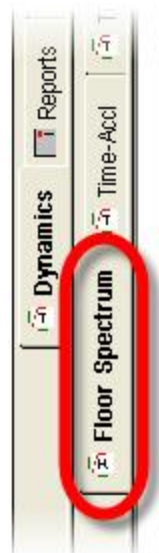


Note that the each line of settings constitutes one floor which can have one or more floor groups assigned. The resulting response spectra will be based on the collective responses of all the nodes in the selected groups.

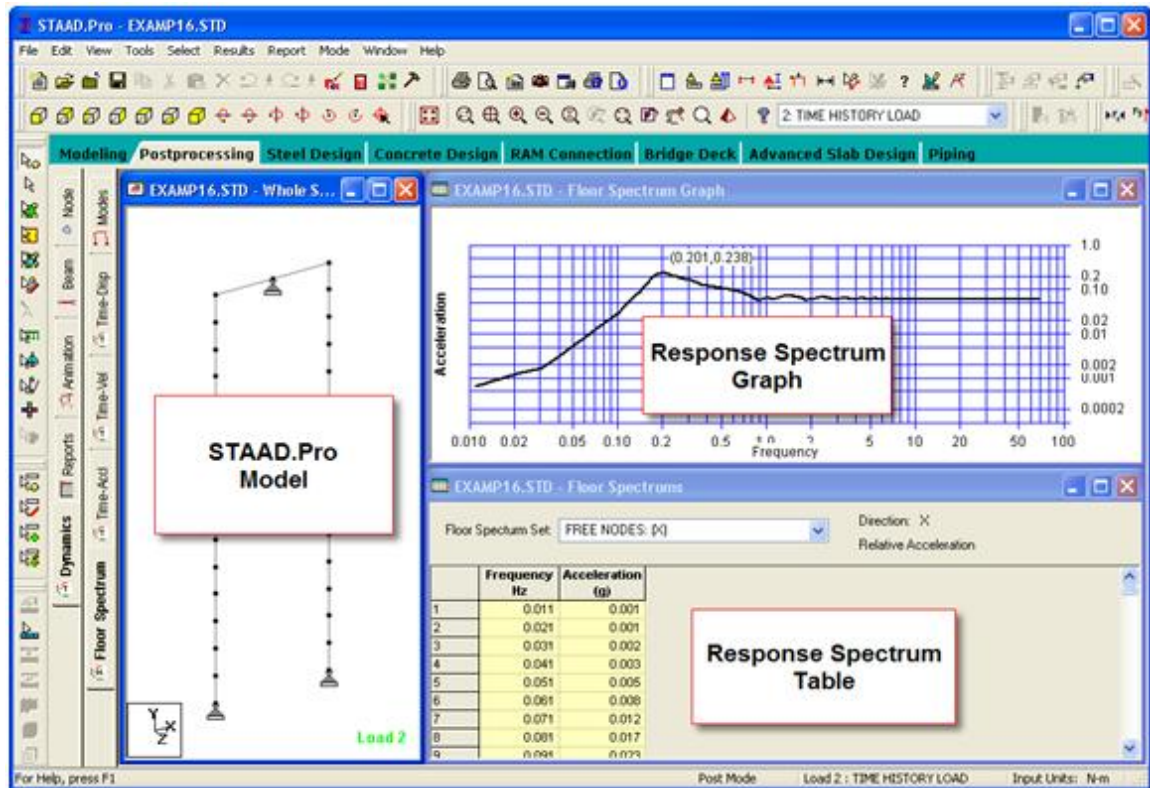
Once the required options have been set, click on the Add button to add the command set to the model which should appear in the Analysis Window thus:-



Once the command has been added and the file saved, the analysis can be run which will generate a new sub-page in the Post-Processing Mode in the Dynamics Page called Floor Spectrum:-



Entering this page, STAAD.Pro will display the floor spectrum thus:-



To change graphs to that of another selection of node groups or damping ratio, then select the required set from the drop list in the Response Spectrum Table.

The graph is initially set to display the results on a log/log graph. This can be changed to a linear graph by right clicking on the graph and selecting the 'Linear Graph' option. Additionally, the calculated points on the graph can be added by again right clicking on the graph and selecting the option 'Show Points'.

The data that has defined the graph can also be exported to a text file and used in a third party application by right clicking on the graph and selecting the option 'Save Data in Text File...'

## AD.V8i.1 Features Affecting the Pre-Processor (Modeling Mode)

### AD.V8i.1.3 Russian Wind Loading

The wind loading as defined in the design code SNI P 2.01.07-85 "Loads and Actions" added in this version of STAAD.Pro can be added graphically to the model by modifications in the Loads Page.

### Description

There are two areas where the graphical user interface has been updated:

## Wind Load Definition dialog box

The dialog has been updated to allow the entry of the required parameters for a Russian Wind Load definition thus:-

The dialog box is titled "Add New : Wind Definitions" and features a close button in the top right corner. On the left side, there is a tree view containing two items: "Intensity" (with a yellow folder icon) and "Exposures" (with a red folder icon). The main content area is titled "Intensity" and contains the following controls:

- "Select Type:" dropdown menu, currently set to "SNiP 2.01.07-85 Loads and Act".
- "SNiP Parameters" section containing:
  - "Pressure:" text input field with the value "100" and the unit "kN/m<sup>2</sup>".
  - "Terrain:" dropdown menu, currently set to "B - Urban Area".
  - "Classification:" dropdown menu, currently set to "1 - Buildings".

At the bottom of the dialog, there are three buttons: "Add", "Close", and "Help".

This will create a definition which can be added to a wind load case.

## Wind Load

**Edit :**

**Wind Load**

Select Type : 1

Direction

X (External Pressure)

Z (External Pressure)

-X (Internal Pressure)

-Z (Internal Pressure)

Factor : -1

**SNiP Parameters**

Apply Wind Load at the Corner

Select Configuration : 0

Reynolds Number (NU) 1

When Y Axis is Vertical

Define Y Range

Minimum 0 m

Maximum 0 m

Define X Range

Minimum 0 m

Maximum 0 m

Define Z Range

Minimum 0 m

Maximum 0 m

Open Structure

Change Close Help

**Note:** The first load case which has a Russian Wind Load command added to it will consider all other loads defined in it as the masses to be considered for calculating the dynamic effect which is required by this command.

For technical details of this wind loading see section [AD.V8i.2.2](#).

## AD.V8i.1 Features Affecting the Pre-Processor (Modeling Mode)

### AD V8i.1.4 Additional Standard Profile Databases

An additional Australian cold formed database has been provided to complement the cold formed sections databases currently provided.

The following database and tables have been added from OneSteel. Duragal®, Galtube® and Tubeline®

Australian Cold Formed Steel Hollow Sections

---

## Circular Hollow Sections:

- Galtube Plus®, 26.9mm to 76.1mm diameter
- Tubeline, grades C250Lo (AS1163) and 350Lo (AS1163), 21.3mm to 457.0mm

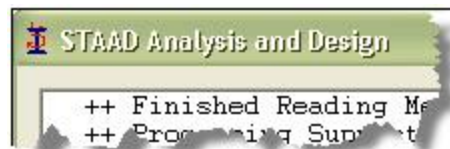
## Rectangular Hollow Sections

- Duragal®, grade C450Lo (AS1163), 50x20mm to 150x50mm
- Galtube Plus®, grade C350Lo, 50x20mm to 75x25mm
- Tubeline®, grade C350Lo (AS1163), 50x20mm to 250x150mm

## Square Hollow Sections

- Duragal®, grade C450Lo (AS1163), 20x20mm to 100x100mm
- Galtube Plus®, grade C350Lo, 20x20mm to 65x65mm
- Tubeline®, grade C350Lo (AS1163), 13x13mm to 250x250mm

## AD.V8i.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.V8i.2 Features Affecting the Analysis and Design Engine

### AD.V8i.2.1 ASME NF

The design of steel sections according to the requirements in the American Society of Mechanical Engineers (ASME) specifications, Rules for the Construction of Nuclear Power Plant Components, Section III – Subsection NF has been implemented and the steel design page has been updated to allow the design parameters to be defined and assigned.

The design requirements for the following years have been added:-

- 1974
- 1977
- 1989
- 1998

For the list of parameters and commands including the default values, please refer to the International Design Codes manual, section 16.

---

For each steel member that is checked, the following checks are performed according to the clauses for that year:-

1. Slenderness check
2. Tension
3. Compression
4. Bending (a) About major axis, (b) About major axis
5. Shear
6. Combined Stresses

## AD.V8i.2 Features Affecting the Analysis and Design Engine

### AD.V8i.2.2 Russian Wind Loading

The wind loading commands have been enhanced to allow the creation of wind loading as defined in the in Russia by the design code SNiP 2.01.07-85 “*Loads and Actions*”.

The basic quantity in the wind loading is the characteristic (normative in Russian terminology) wind pressure. The reference wind velocity pressure corresponds to a 10-minute time-averaged velocity pressure at 10-metres height in a flat terrain, based on a 5-year return period. This wind pressure is the static component of the wind load. The total wind pressure consists of static and fluctuating components. If the structure is sufficiently flexible, according to the code provisions, the dynamic structural response to the fluctuating wind component must be taken into account.

The updated wind loading commands automatically perform both the aerodynamic and structural load analysis of vibration-susceptible buildings and structures.

The wind loading commands have been updated to support the wind loading as defined in the Russian design code. This requires the creation of the following commands.

#### General Format

There are three parts to creating a Russian wind load on a STAAD Model.

#### Definition of the wind load requirements

This should occur before the definition of the first load case

**DEFINE WIND LOAD**

**TYPE j**

**SNIP PRESSURE f1 TERRAIN { A | B | C } CLASSIFICATION { 1 | 2 | 3 }**

**EXPOSURE e<sub>1</sub> { JOINT joint-list | YRANGE f<sub>2</sub> f<sub>3</sub> }**

Repeat **EXPOSURE** command up to 98 times.

Where:

j = wind load system type number (integer)

---

$f_1$  = the characteristic value of wind pressure, always positive. All values of the pressure within the range of corresponding SNIp table are valid.

**TERRAIN** is terrain roughness category:

- A. Coastal Zone
- B. Urban
- C. Large City according to the SNIp classification.

**CLASSIFICATION:**

- 1. for prismatic building structures,
- 2. for general concrete structures,
- 3. for general open framed steel structures.

$e_1, e_2 \dots e_m$  = multiplying wind pressure influence areas. For stick-type structures with compact members aligned along the vertical line (e.g, chimneys, elevated water tanks), influence areas are set to unity, and exposure factors are aerodynamic diameters of members.

*joint-list* = Joint list associated with Exposure Factor (joint numbers or "TO" or "BY") or enter only a group name.

If the command **EXPOSURE** is not specified or if a joint is not listed in an Exposure, the exposure factor for those joints is chosen as 1.0.

### Application of the wind load definition within a load case

Each load case which is to include a Russian wind load specification should include the following:

**LOAD i**

**WIND LOAD { X | Z } (f) ( OBLIQUE ) CONFIGURATION if ( NU fNU )**

**TYPE j { X RANGE  $f_1 f_2$  | Y RANGE  $f_1 f_2$  | Z RANGE  $f_1 f_2$  | LIST *member-List* | ALL }**

Where:

X or Z defines the wind direction

f = wind pressure factor. A negative value is used to define a reverse wind direction

if = configuration parameter for prismatic buildings, range 0-12;

- 0. prismatic building structure - Rectangular building- Outstanding architectural details on the left façade.
- 1. prismatic building structure - Rectangular building- Both side façades are smooth
- 2. prismatic building structure - Rectangular building- Outstanding architectural details on the right façade
- 3. prismatic building structure - Rectangular building- Outstanding architectural details on both side façades

4. prismatic building structure - Rectangular building- Triangular building
5. prismatic building structure - Rectangular building- Rhombic building
6. prismatic building structure - Rectangular building- Number of vertices of polygonal building, not more than 12
7. prismatic building structure - Less than 3 — rectangular building
8. prismatic building structure - 3 — triangular
9. prismatic building structure - 4 — rhombic
10. prismatic building structure - More than 4 — polygonal
11. framed RC structure
12. lattice steel structure

it is ignored for non-building structures, but cannot be omitted from the command

fNU = wind pressure correlation coefficient. If parameter is omitted or is exactly 1, a computed value is used instead. For rectangular buildings, the correlation coefficient is always calculated automatically, thus any specified value will be ignored.

Additionally, as part of the load force is determined from the dynamic behavior of the model, the first load case that includes a Russian Wind Load specification also needs to include all the masses that can cause the structure to vibrate from which a dynamic analysis will be used to extract the mode shapes.

**Note:** This in effect becomes a dynamic load case, thus if the effects of these forces are to be combined with the effects of static load cases, then this should be done using a LOAD COMBINATION which references the required load case results with the appropriate factors.

### Cut-Off Frequency or Mode Shape

As the analysis will requires extraction of eigen solutions to determine the dynamic effects of the wind loading, the number of modes to be used will also affect the results. Thus setting the command CUT OFF FREQUENCY or CUT OFF MODE should be considered and specified as required prior to the definition of the load cases.

If the cut-off command is omitted, six mode shapes are computed by default.

For more information on these commands, see the Technical Reference manual section 5. 30.1 Cut-Off Frequency, Mode Shapes or Time.

### Example

```
DEFINE WIND LOAD
TYPE 1
SNIP PRESSURE 0.38 TERRAIN A CLASSIFICATION 1
EXP 0.5 JOINT 1 3 5 7 9 11
```

```

*
LOAD 1 LOADTYPE WIND TITLE WIND LOAD IN THE +VE X DIRECTION
WIND LOAD X 1 CONFIG 0 NU 1 TYPE 1
* MASS MODEL REQUIRED IN FIRST WIND LOAD CASE
JOINT LOAD
3 TO 6 FZ 62.223
9 TO 12 FZ 62.223
*
LOAD 2 LOADTYPE WIND TITLE WIND LOAD IN THE -VE X DIRECTION
WIND LOAD X -1 CONFIG 0 NU 1 TYPE 1
* NO MASS MODEL OR ADDITIONAL LOADS IN THIS LOAD CASE
*
LOAD 3 LOADTYPE WIND TITLE WIND LOAD IN THE +VE Z DIRECTION
WIND LOAD Z 1 CONFIG 0 NU 1 TYPE 1
* NO MASS MODEL OR ADDITIONAL LOADS IN THIS LOAD CASE
*
LOAD 4 LOADTYPE WIND TITLE WIND LOAD IN THE -VE Z DIRECTION
WIND LOAD Z -1 CONFIG 0 NU 1 TYPE 1
* NO MASS MODEL OR ADDITIONAL LOADS IN THIS LOAD CASE
*
LOAD 10 LOADTYPE DEAD TITLE SELFWEIGHT LOAD CASE
SELF Y -1 ALL
*
LOAD COMBINATION 100 WIND PLUS SELFWEIGHT
1 1.0 10 1.0

```

## Notes

- a. This command cannot be used with models that have been defined with the SET Z UP command.
- b. For more information on wind loading see sections 5.31.3 Definition of Wind Load and 5.32.12 Generation of Loads of the Technical Reference Manual.

## AD.V8i.2 Features Affecting the Analysis and Design Engine

### AD.V8i.2.3 Floor Response Spectrum

The following commands have been added in order to allow the response spectrum of floors to be extracted from a time history analysis.

This command is used to specify the calculation of floor and/or joint spectra from time history results. The Floor Response Spectrum command must immediately follow an analysis command. That analysis can only contain a single time history load case.

Refer to Section 5.32.10.3 of the Technical Reference manual details on the **GENERATE FLOOR SPECTRUM** command.

## AD.V8i.3 Features Affecting the Post-Processing (Results Mode)

### Postprocessing

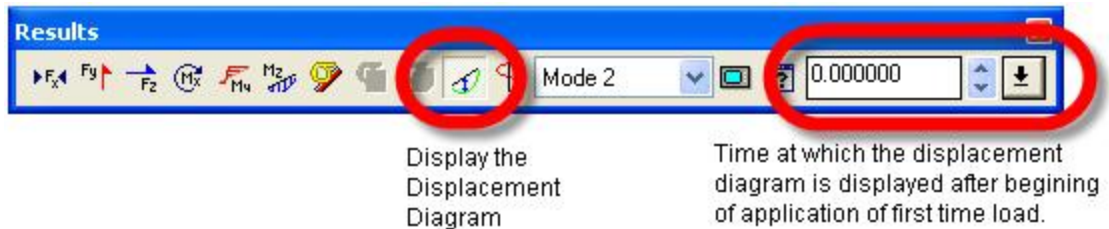
Several new features have been added and existing features have been modified in the post-processing section of the program, also known as the Results Mode. These are explained in the following pages.

## AD.V8i.3 Features Affecting the Post-Processing (Results Mode)

### AD.V8i.3.1 Time History Animation

In order to visualize the displacement that occurs on a model during the application of a time history load, a new toolbar icon has been added which allows the displacement at a specific time instance to be displayed.

The results toolbar has been updated to include a new option which is activated when the displacement diagram icon has been clicked and the current load case contains time history loading:



For a time history load case the displacement that is displayed will be defined by the time instance entered in the above edit box. Alternatively the time can be set from a sliding scale by clicking on the button with the down arrow icon which displays the following option setting:



The slider scale is based on the overall time for which the time history analysis has been performed.

The displacement is produced at the time at which the slider arrow is dragged to and the mouse button released.

If the **Apply Immediately** option is selected, then the application will attempt to render the displacement diagram dynamically as the slider is dragged up and down the scale of time. For large models this may prove to be too demanding on the graphics system and left un-checked which means that the displacement diagram will be produced at the time step at which the slider arrow is released.

Note that the displacement diagram will be set to the scale as defined in the Scales sheet in the *Structure Diagrams* dialog box.

Additionally note that it is possible to view the time v displacement of individual nodes by clicking on the Dynamics>Time-Disp in the post processing mode which is only available for models that include time history.

## **AD.V8i.3 Features Affecting the Post-Processing (Results Mode)**

### **AD.V8i.3.2 Enhanced Plate Stress Results**

In order to provide additional understanding of stress distribution in finite element models, STAAD.Pro has expanded the sets of results that can be reported for each element both in the Plate Centre Stress Table and graphically using the Plate Stress Contour.

To view the results data of plate elements, enter the Post Processing (Results) Mode, and click on the Plate Contour Page on the left menu.

#### **Plate Centre Stress Table**

The new Combined Stresses sheet in the Plate Centre Stress table provides resolved stresses for the top (positive local Z) and bottom (negative local Z axis) surface for each plate element, referred to as the Top Combined Stresses and Bottom Combined Stresses respectively.

		Top Combined Stress			Bottom Combined Stress		
Plate	L/C	Comb. SX psi	Comb. SY psi	Comb. SXY psi	Comb. SX psi	Comb. SY psi	Comb. SXY psi
1	1 ALL	142.747	142.737	101.261	-142.671	-142.681	-101.215
2	1 ALL	514.879	101.366	218.581	-514.854	-101.376	-218.579
3	1 ALL	968.391	192.264	200.708	-968.705	-192.306	200.750

Note that if the sheet title Combined Stresses is not visible on the top of the table, then click on the right arrows displayed on the left of the table header to scroll the table sheets which will display the title.

The combined stresses are calculated thus:

Top:

$$S_{X_{top}} = S_X + M_X/S$$

$$S_{Y_{top}} = S_Y + M_Y/S$$

$$S_{XY_{top}} = S_{XY} + M_{XY}/S$$

Bottom:

$$S_{X_{bottom}} = S_X + M_X/S$$

$$S_{Y_{bottom}} = S_Y + M_Y/S$$

$$S_{XY_{bottom}} = S_{XY} - M_{XY}/S$$

Where:

$$S = t^2/6 t$$

t = average plate thickness

## Plate Stress Contour

The Plate Stress Contour sheet of the Diagrams dialog has been enhanced to allow visualization of these stresses. The six new stress results are available in the Stress Type pull down menu thus:

**Diagrams**

Structure    Loads and Results    Scales    Labels

Force Limits    Animation    Design Results    Plate Stress Contour

Load Case: 1:

Stress Type

Stress type: Top Combined SX (local)

Contour Type

- Normal
- Enhanced
- Normal
- Absolute Value
- Index Based
- View Stress
- Re-Index Factor
- Show Displacement
- Contour Based
- Use Custom

Minimum:  Global Stress

Maximum:  Global Stress

No of values: 1

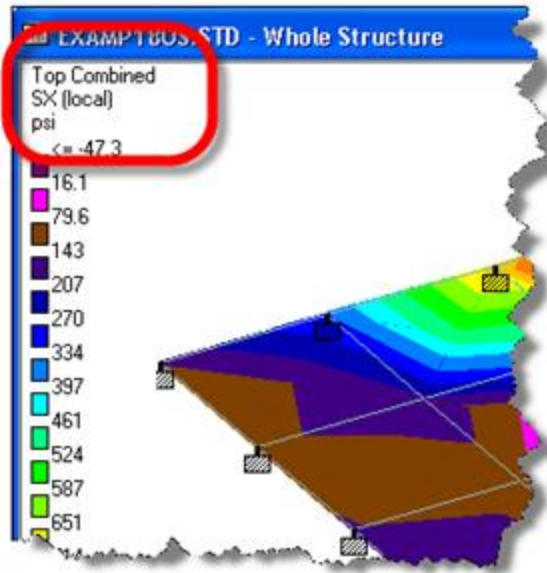
Stress Type List:

- Max Absolute
- Max Top (Principal Major Stress)
- Min Top (Principal Minor Stress)
- Tau Max Top
- Max Bottom (Principal Major Stress)
- Min Bottom (Principal Minor Stress)
- Tau Max Bottom
- Max Von Mis
- Von Mis Top
- Von Mis Bottom
- Max Tresca
- Tresca Top
- Tresca Bottom
- SX (local)
- SY (local)
- SXY (local)
- MX (local)
- MY (local)
- MXY (local)
- SQX (local)
- SQY (local)
- Global Moment
- Global Stress
- Base Pressure
- Top Combined SX (local)
- Top Combined SY (local)
- Top Combined SXY (local)
- Bottom Combined SX (local)
- Bottom Combined SY (local)
- Bottom Combined SXY (local)

	A	B
Min	-47.341	16.1423
Max	16.1423	79.6255
Avg	79.6255	143.109
Min	143.109	206.592
Max	206.592	270.075
Avg	270.075	333.559
Min	333.559	397.042
Max	397.042	460.525
Avg	460.525	524.008
Min	524.008	587.492
Max	587.492	650.975
Avg	650.975	714.458
Min	714.458	777.941
Max	777.941	841.425
Avg	841.425	904.908
Min	904.908	>= 968.391

Note: Stress contours and index values are computed on the basis of weighted average values.

OK    Cancel    Apply    Help





# Technical Support

These resources are provided to help you answer support questions:

- Service Ticket Manager — <http://www.bentley.com/serviceticketmanager> — Create and track a service ticket using Bentley Systems' online site for reporting problems or suggesting new features. You do not need to be a Bentley SELECT member to use Service Ticket Manager, however you do need to register as a user.
- Knowledge Base — <http://appsnet.bentley.com/kbase/> — Search the Bentley Systems knowledge base for solutions for common problems.
- FAQs and TechNotes — [http://communities.bentley.com/Products/Structural/Structural\\_Analysis\\_\\_\\_Design/w/Structural\\_Analysis\\_and\\_Design\\_\\_Wiki/structural-product-technotes-and-faqs.aspx](http://communities.bentley.com/Products/Structural/Structural_Analysis___Design/w/Structural_Analysis_and_Design__Wiki/structural-product-technotes-and-faqs.aspx) — Here you can find detailed resolutions and answers to the most common questions posted to us by you.
- Ask Your Peers — <http://communities.bentley.com/forums/5932/ShowForum.aspx> — Post questions in the Be Community forums to receive help and advice from fellow users.



# Index

---

**2**

2.3 Floor Response Spectrum 92

**A**

Actions 95

AD.V8i.1.3 Russian Wind Loading 95

Add 80

Add button 93

Advanced Wizard 83

Although CIS 91

Analysis Window 93

Annexe Nationale a la NF EN 1993-1-1 20054:

ANSI/AISC 38

N690-1994

ASME 91, 98

    setting 92

ASME NF 91

Australian 91, 97

Australian Cold Formed Steel Hollow Section

AutoPIPE 55

**B**

Belgium National Annex xx

**C**

Check In 86

Check In dialog 86

    launches 86

Checked Out 81

Chinese Response Spectrum 43

Chinese Static Seismic Loading 43

CimSteel Integration Standard 88

CimSteel Integration Standard, Version 234

Circular Hollow Sections 98

---

CIS 87

    Selecting 88

CIS/2 34

CIS2 Export File 88

Clicking 88

    Import button 89

Construction 91, 98

    Rules 91, 98

Containing 88

Corresponds 99

    10 99

Create New Version 86

Created as 89

Creates 90

    STP file 90

Current Code 92

Cut-Off Frequency 101

CUT OFF FREQUENCY 101

CUT OFF MODE 101

**D**

DATA 88

Databases such 91

Design 91

    entering 92

Developer 88

Document Properties 84

Drive 87

Duragal 97

During 81

Dynamics Page 94

**E**

EC3 Torsion Design 13

Eigen 101

ENDSEC	88	International Design Codes	92, 98
Entering	92	refer	98
Design	91		
Eurocode 3	xx, xxi, 15-16, 42		
European	91		
EXPORT	87		
Export Model button	90		
		<b>J</b>	
		Japanese	91
		<b>L</b>	
		Launches	86
		Check In dialog	86
		Linear Graph	95
		Link	81
		LOAD_COMBINATION_OCCURRENCE	88
		LOAD_ELEMENT_DISTRIBUTED_CURVE_LINE	88
		LOADING_COMBINATION	88
		Loads	95
		Generation	102
		Loads Page	95
		<b>M</b>	
		Meta-data	80
		Minute	99
		Mode	91
		Modeling	91
		Model STD	80
		Modeling	91
		Mode	91
		<b>N</b>	
		NA to BS EN 1993-1-1 2005	42
		New	
		Saving	82
		No Wizard	83
		NU	99
		Nuclear Power Plant Components	91, 98
		<b>O</b>	
		OneSteel	97
ENDSEC	88		
Entering	92		
Design	91		
Eurocode 3	xx, xxi, 15-16, 42		
European	91		
EXPORT	87		
Export Model button	90		
		<b>F</b>	
File menu	80		
FILE_DESCRIPTION	88		
FILE_NAME	88		
Finnish National Annex	15-16		
Floor Spectrum	92		
Folder	84		
French National Annex	42		
Functionalities	80		
		<b>G</b>	
Galtube	97		
GB50011-2001	43		
Generate Floor Spectrum	92		
Generation	102		
GLOBAL_LOAD	88		
		<b>H</b>	
HEADER	88		
		<b>I</b>	
Implementation_level	88		
IMPORT	87		
Import button	89		
Clicking	88		
Inc	88		
Include	100		
Indian	91		
Integration Server	80		



STAAD Model	80		
Opening	81		
STAD	82		
Start Page	80		
Static Seismic Loading			
Chinese	43		
STD file	80		
Steel	91		
STEP	88		
STEP (Part 21)	34		
STEP file	88		
STEP Tools	88		
STP file	90		
creates	90		
STRUCTURAL_FRAME_SCHEMA	88		
Subsection NF	91, 98		
Support Connection Wizard	56		
<b>T</b>			
Technical Reference	101		
see	101		
Technical Reference Manual	101		
TERRAIN	99		
Text File	95		
Time	99		
Time_stamp	88		
TRUE_LENGTH	88		
<b>U</b>			
UK National Annex	42		
ULS	88		
Update Server Copy	85		
Update Server Copy icon	85		
selecting	82		
		<b>V</b>	
		Version	88
		<b>W</b>	
		Whilst STAAD	88
		WIND LOAD	95
		Working	83



Bentley Systems, Incorporated  
685 Stockton Drive, Exton, PA 19341 USA  
+1 (800) 236-8539  
[www.bentley.com](http://www.bentley.com)

