
STAAD.Pro

V8i (SELECTseries 2)

Release Report 20.07.07



DAA039020-1/0002
Last updated: 21 February 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Unpublished - rights reserved under the Copyright Laws of the United States and International treaties.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

Section 1 STAAD.Pro V8i (SELECTseries 2)	1
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 Beta Features.....	31
Section 2 STAAD.Pro V8i (SELECTseries 1)	33
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
Section 3 STAAD.Pro V8i	61
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.....	76
AD.2007-05.2 Features Affecting the Concrete Design Mode.....	76
AD.2007-05.2.1 RC Designer Member and Envelope Import.....	77
Technical Support	79
Index	81

Section 1

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 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

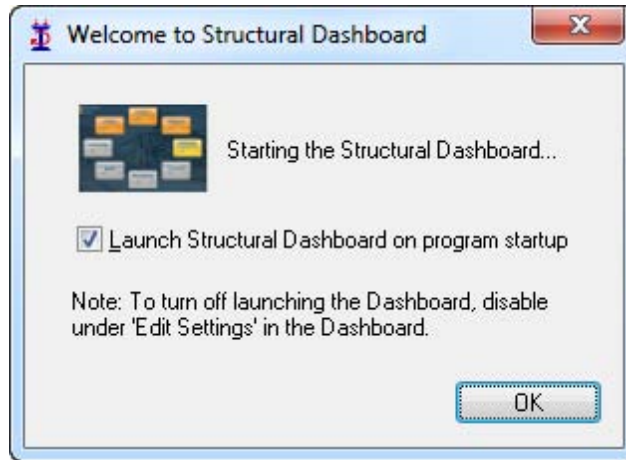
Objective

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.



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

> **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 *fi8* directs the program to output either the beginning and last 54 data points (*fi8* = 1) or the entire curve (*fi8* = 2) or a select number of beginning and last data points (*fi8* ≥ 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 *fi8* 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 *fi9* 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.

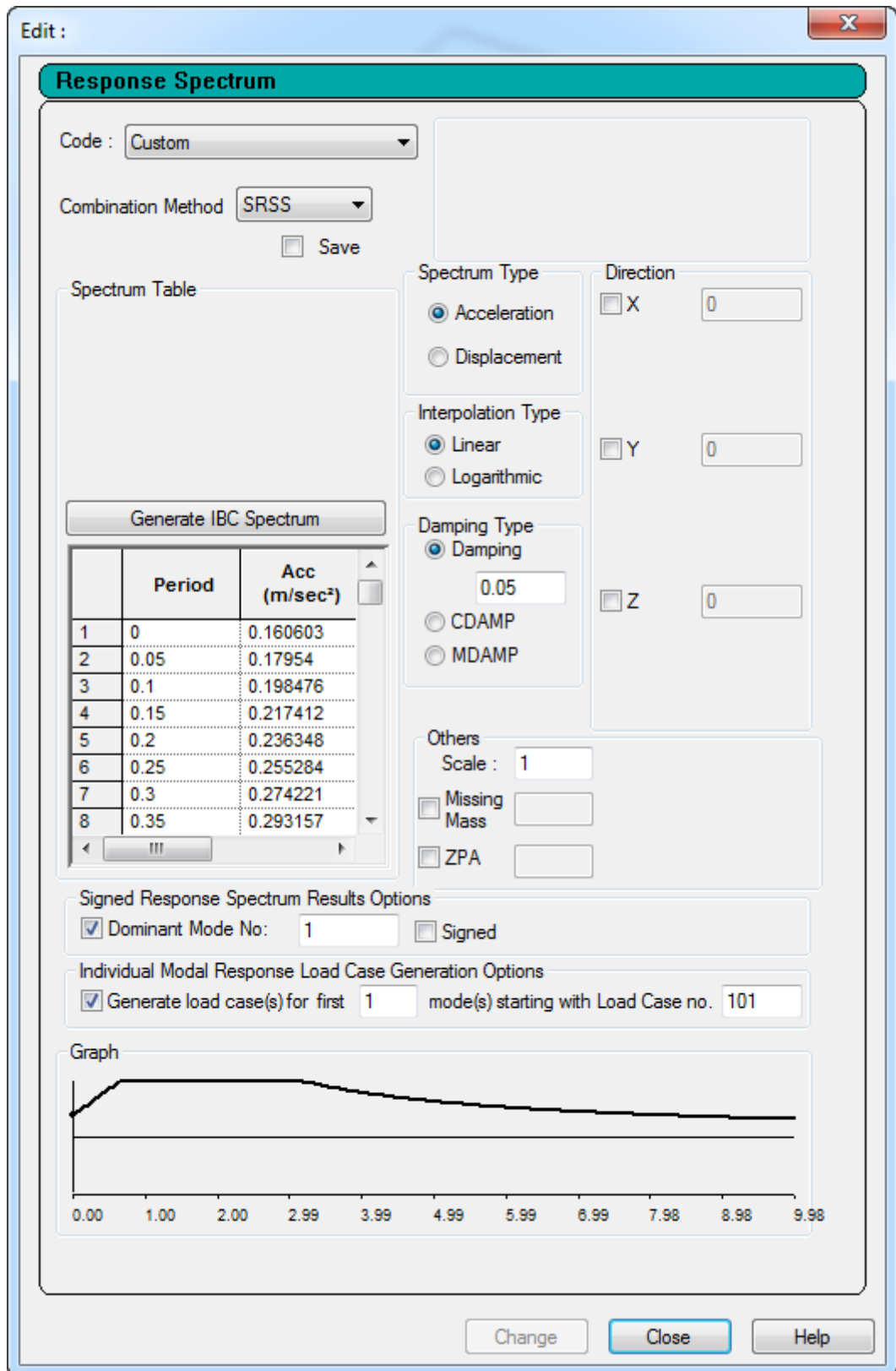


Figure - Updated Response Spectra dialog containing controls for generating signed results and IMR load cases

> 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 SNIIP 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.

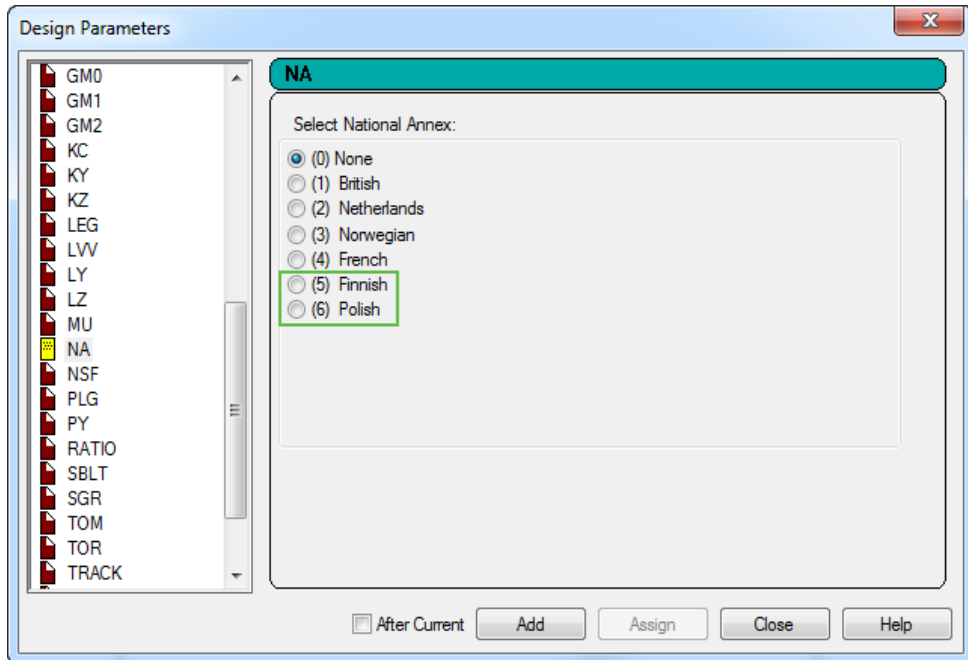
Note: If Torsion design is selected for a member which does not have any torsional moment, a warning is displayed in the output.

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 C_1 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

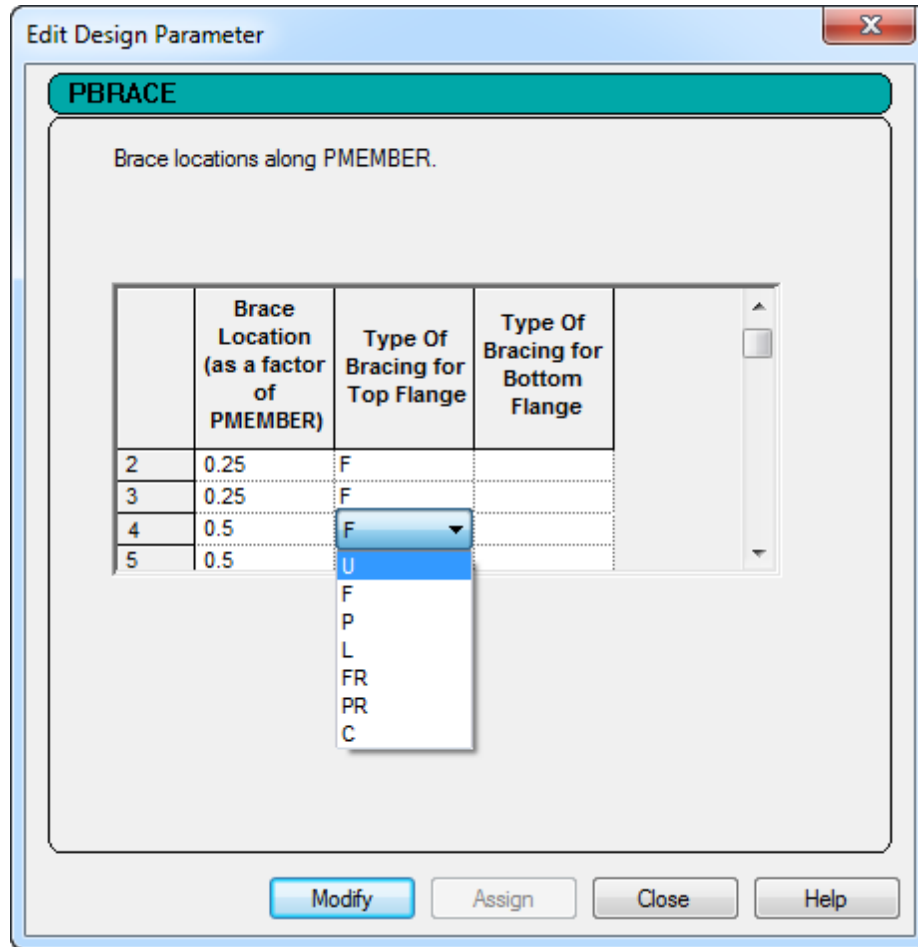
The physical member mode is initiated through the Toggle Physical Member mode tool found in the Steel Design toolbar (which docked on the left hand side of the screen by default). This "mode" is used when modeling the structure and any parameter or specification added while this tool is toggled on will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

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

and Q1.6-1b

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

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

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

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

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

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

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

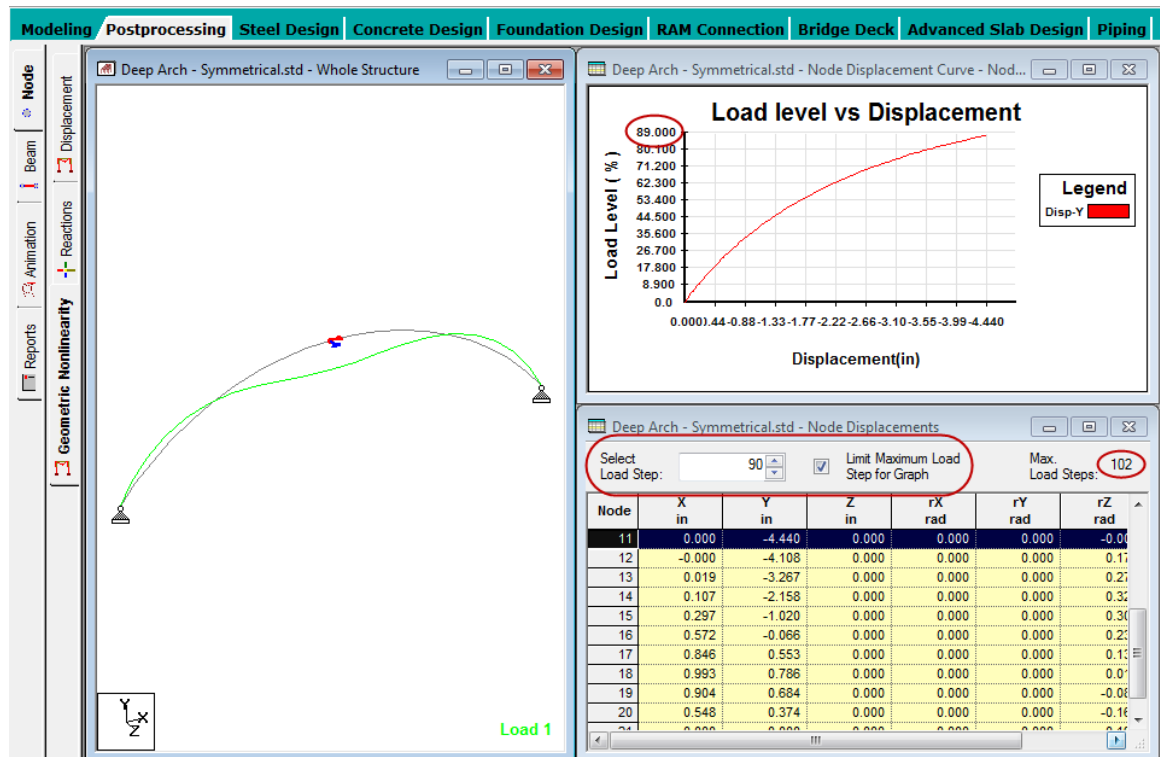
The graphical display of the results of a geometric nonlinear analysis have been improved. The Graphs are now easier to read and a new control has been added to limit the maximum Load Step displayed.

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 Beta Features

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

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

AD.2007-07.4.1* 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 2

STAAD.Pro V8i (SELECTseries 1)

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 (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

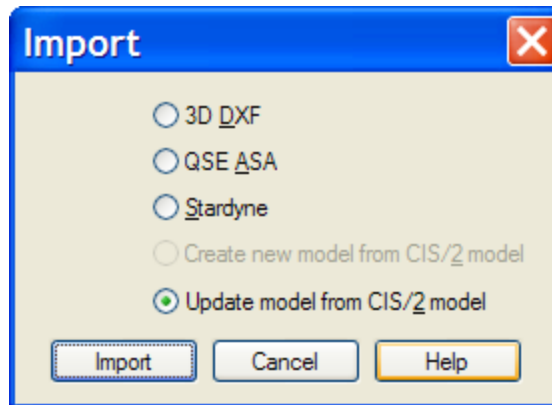
Additionally, two new VBS Macros have been included in the User Tools to aide in the verification of model integrity before and after update operations using the Import dialog. These may be found in the User Tools drop-down menu in the File toolbar or under Tools > User Tools.

> 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.



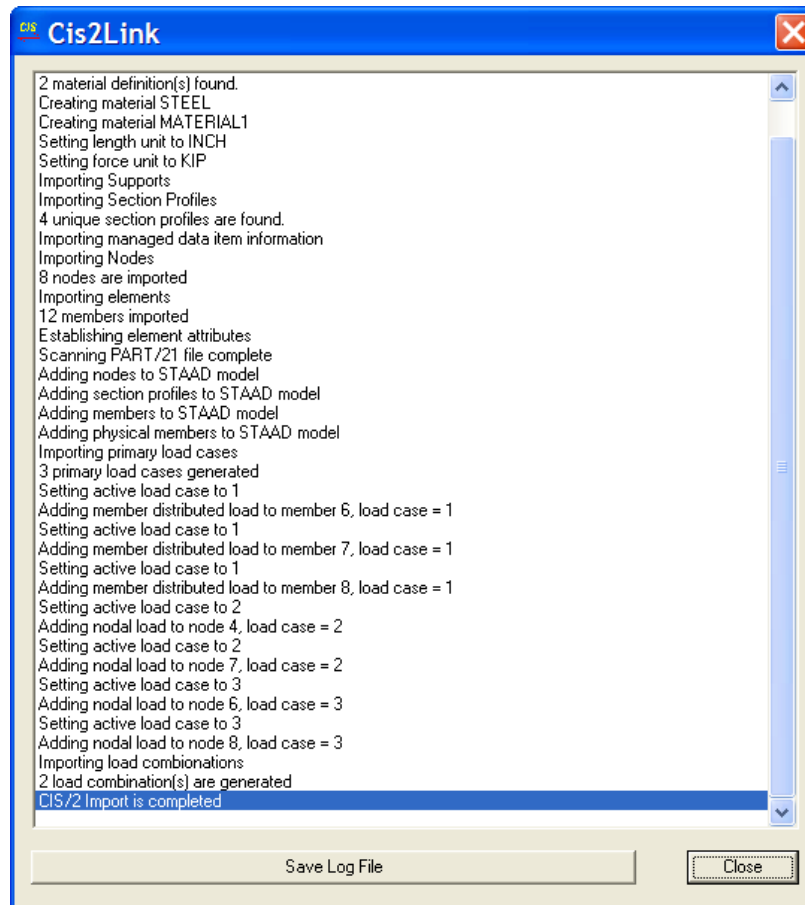
2. Select either to Create new model from CIS/2 model or Update mode from CIS/2 model.

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.

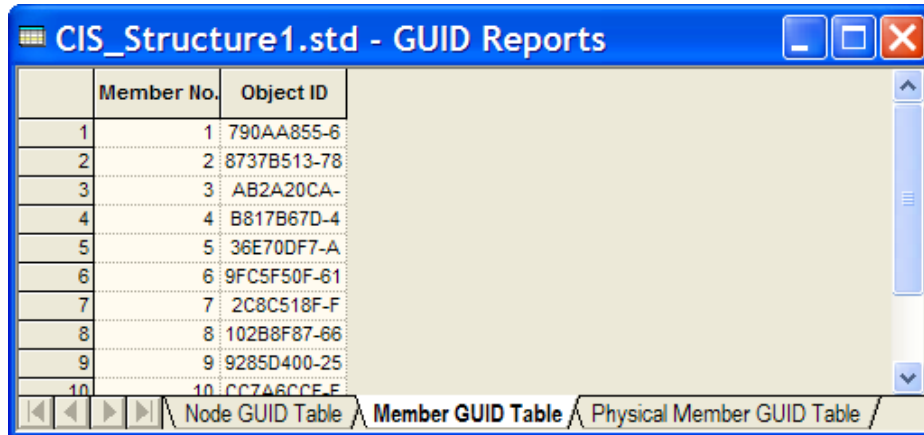
5. Click the Close button.

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.

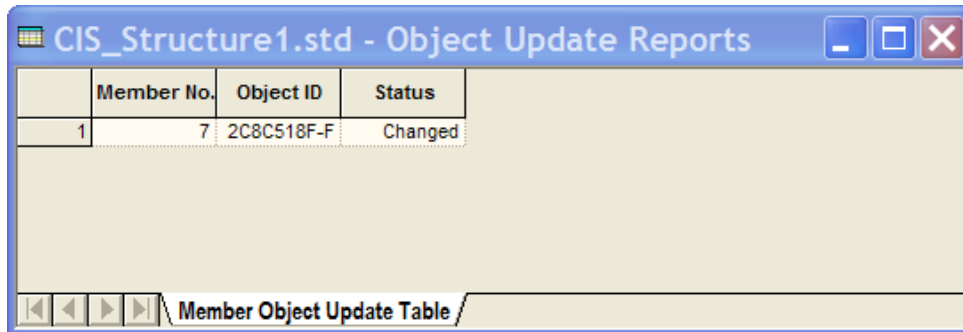


	Member No.	Object ID
1	1	790AA855-6
2	2	8737B513-78
3	3	AB2A20CA-
4	4	B817B67D-4
5	5	36E70DF7-A
6	6	9FC5F50F-61
7	7	2C8C518F-F
8	8	102B8F87-66
9	9	9285D400-25
10	10	CC7A6CCE-E

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 SNiP 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.

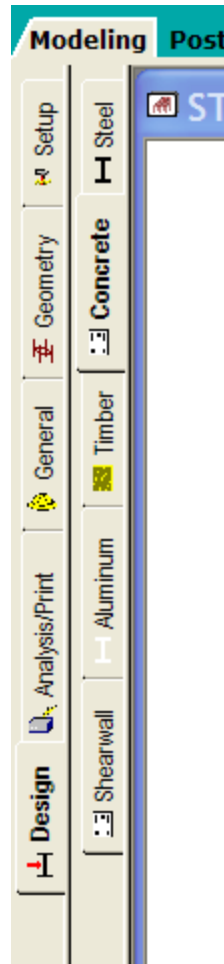
Refer to the International Codes Manual, section 12A "Russian Codes – Concrete Design per Russian Code (SNiP 2.03.01-84*)" for additional information.

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.

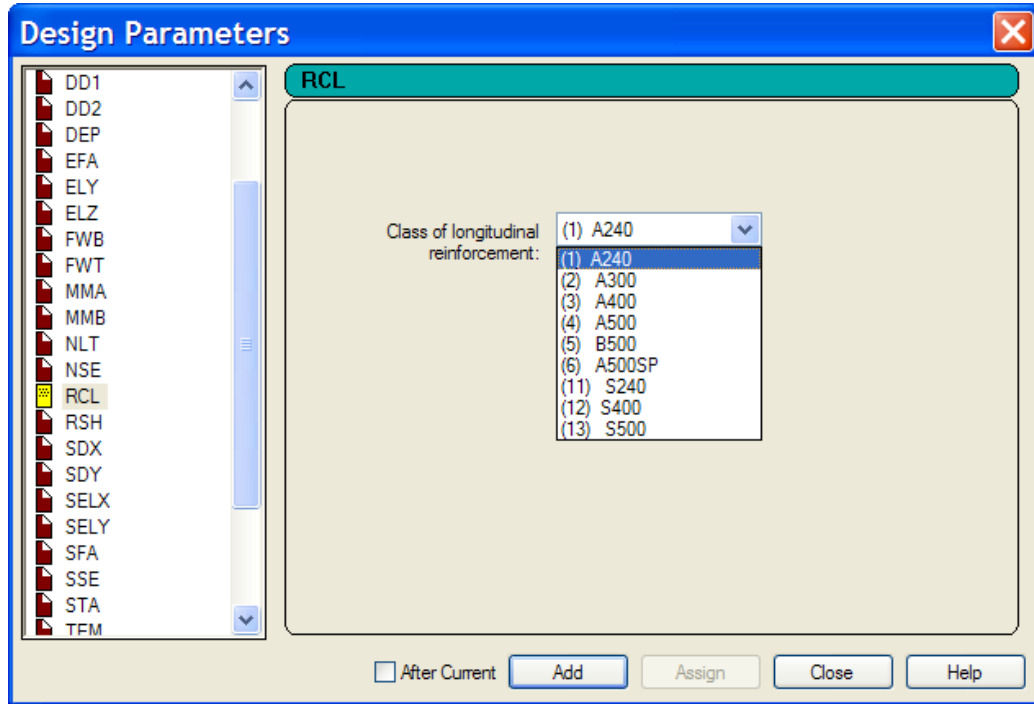
Note: STAAD.Pro supports design of concrete beams and columns to Russian SP52 code is supported in the interactive RC Designer module (Concrete Design Mode in the GUI). For more details of the RC Designer see the RC Designer manual found in the Additional Modules section of the online help. For details 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

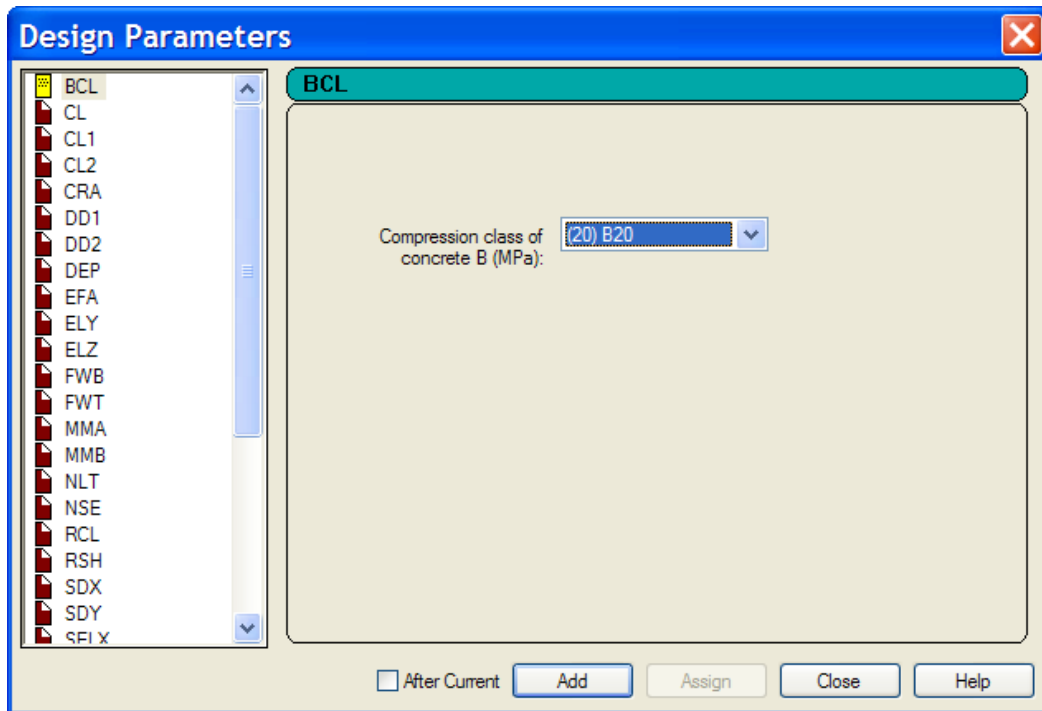
1. In the modeling mode, select the Design page and Concrete material sub-page.



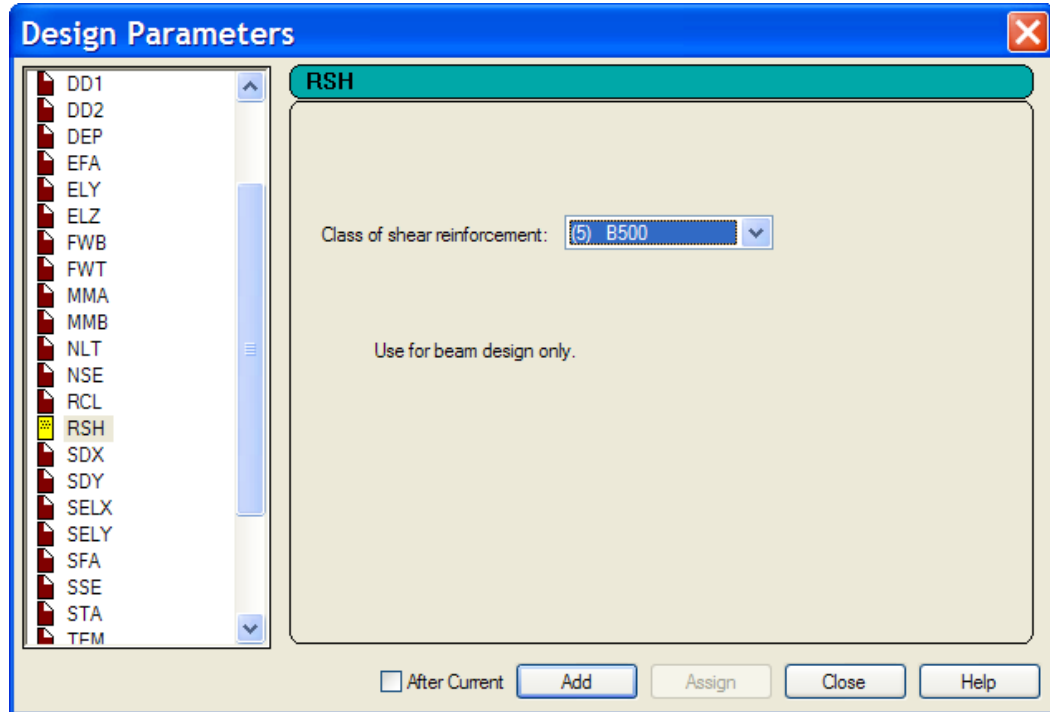
2. Select SNIp 2.03 01-84 in the Current Code drop-down list.
3. Click the Define Parameters button below the model outline panel.
4. The RCL, BCL, and RSH parameters have the new definitions available. Refer to the International Codes Manual for additional information.
Reinforcement Class for longitudinal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



Compression class of concrete B (BCL):

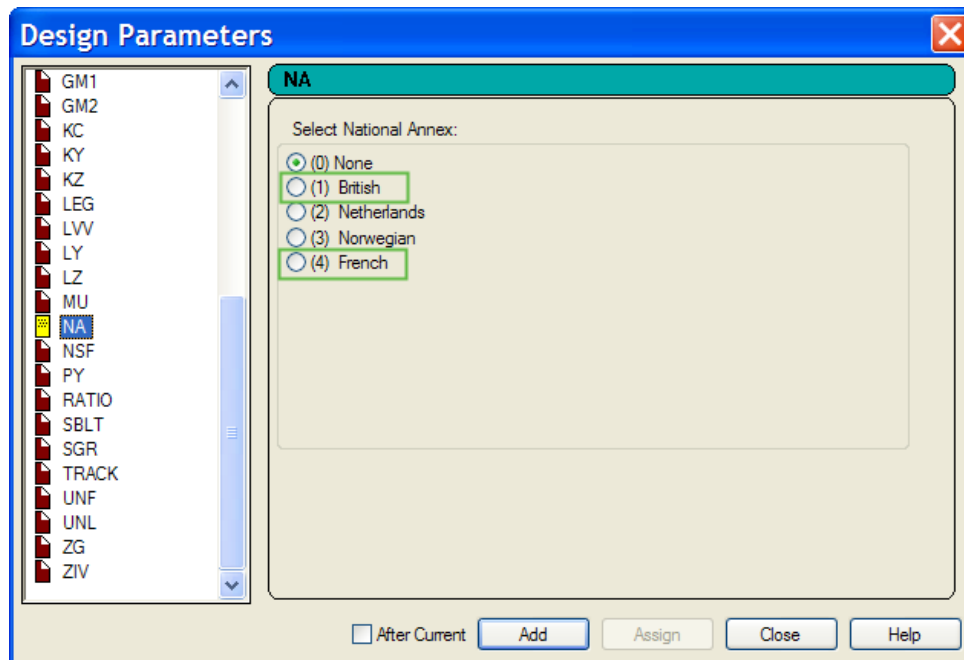


Reinforcement class for shear reinforcement (RSH):



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).

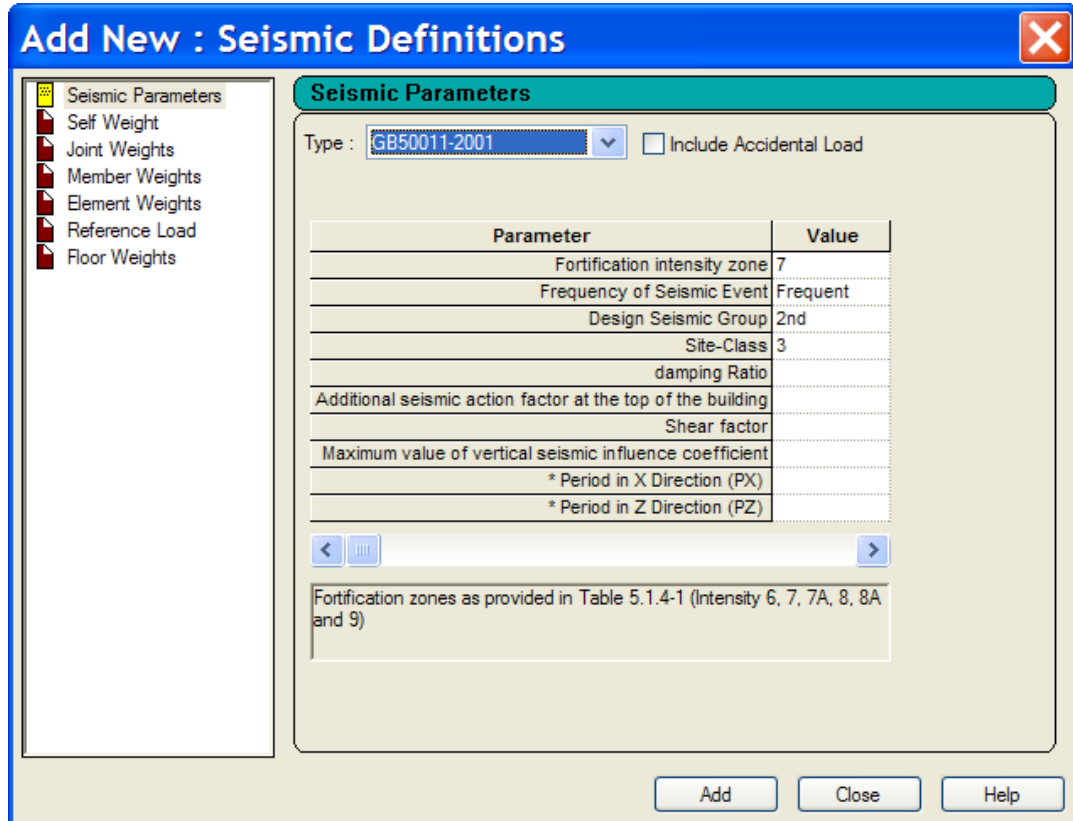
Note: All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

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
Live load on the floor, calculated according to actual state	1.0
Live load on the floor, calculated according to	Library, 0.8

Type of Variable	land Combination coefficient	
equivalent uniform state	archives	
	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 0.05 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. is thought to characteristic period, shall select the maximum value (α_{max});

- c. Curvilinear decrease section, whose period from characteristic period thought to 5 times of the characteristic period, the power index (γ) shall choose 0.9.
- d. Linear decrease section, whose period from 5 times characteristic period thought to 6s, the adjusting factor of slope (η_1) shall choose 0.02.

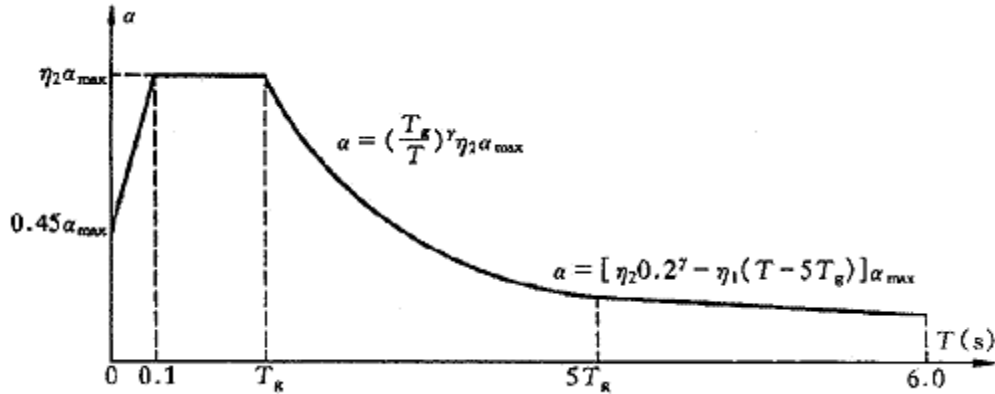


Figure 2.1 Seismic influence coefficient curve

- 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:

- γ - the power index of the curvilinear decrease section;
- ζ - 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:

- η_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:

η_2 - the damping adjustment factor, when it is smaller than 0.55 shall equal 0.55.

Calculation of horizontal seismic action

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

$$F_{Ek} = a_1 G_{eq} \quad \text{E2.4}$$

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

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

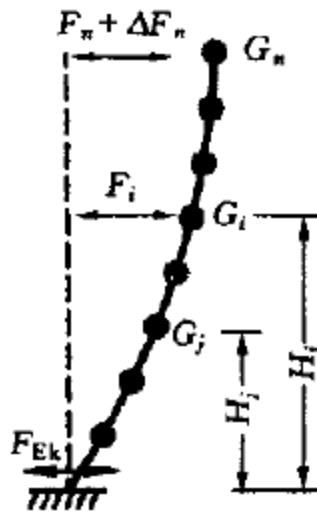


Figure 2.2 Calculation of horizontal seismic action

Where:

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

α_1 - horizontal seismic influence coefficient corresponding to the fundamental period of the structure, which shall be determined by using Clause 2.3. For multistory masonry buildings and multi-story brick buildings with bottom-frames or inner-frames, the maximum value of horizontal seismic influence coefficient should be taken.

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

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

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

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

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

ΔF_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

Tg (s)	$T_1 > 1.4T_g$	$T_1 \leq 1.4T_g$
≤ 0.35	$0.08T_1 + 0.07$	0.0
$< 0.35 \approx 0.55$	$0.08T_1 + 0.01$	
> 0.55	$0.08T_1 - 0.02$	

Note: T_1 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.

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

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

Table 2.5 Minimum seismic shear factor value of the floor level

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

Note:

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:

$$F_{Evk} = \alpha_{v \max} G_{eq} \quad \text{E2.8}$$

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

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.

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

G_{eq} - equivalent total gravity load of the structure, which may be taken as 75% of the representative value of the total gravity load of the structure.

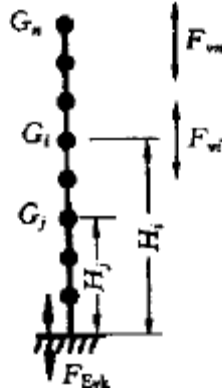


Figure 2.3 Sketch for the computation of vertical seismic action

Complementarities:

1. 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.
2. Eccentricity: similar to UBC code. The eccentricity value of gravity center on each floor should be $e_i = \pm 0.05L_i$,

where:

e_i - Eccentricity value of gravity center on i-th floor.

L_i - maximum width of calculated story of the building.

3. Structures having obviously asymmetric mass and stiffness distribution, the torsion effects caused by both two orthogonal horizontal direction seismic action shall be considered; and other structures, it is permitted that a simplified method, such as adjusting the seismic effects method, to consider their seismic torsion effects.

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

General Format

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

DEFINE GB (ACCIDENTAL) LOAD

INTENSITY { 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 λ , 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 0.75Geq, specify the product of the factor on maximum horizontal seismic influence factor and factor of total gravity load as f_4 . For instance,

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

And

- $G_{v,eq} = 0.75G_{eq}$
- Specify f_4 as $(0.65*0.75)$ i.e. equal to 0.4875
- f_5 and f_6 = optional time period along two horizontal direction (X and Y, respectively).

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 ADJUSTMENT	LATERAL LOAD (KIP)	GRAVITY LOAD (KIP)	LAMBDA (%)	LAMBDA MIN
30.000 1.00	23.406	79.045	29.61	5.00
20.000 1.00	19.208	180.888	10.62	5.00
10.000 1.47	9.604	282.731	3.40	5.00

JOINT LOAD - 1	LATERAL LOAD (KIP)	TORSIONAL MOMENT (KIP -FEET)	VERTICAL LOAD (KIP)	FACTOR -
17 0.221	0.541 MY	0.000	FY	-
18 0.271	0.663 MY	0.000	FY	-
19 0.271	0.663 MY	0.000	FY	-
20 0.221	0.541 MY	0.000	FY	-
21 0.271	0.663 MY	0.000	FY	-
22 0.321	0.785 MY	0.000	FY	-
23 0.321	0.785 MY	0.000	FY	-

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

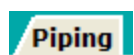
Connections are designed in the newly updated RAM Connection Mode by creating *Joints*, from the geometry, section properties and forces resulting from the analysis and assigning a design brief made up of connection templates. A suitable connection design, if one is available, will be reported once you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

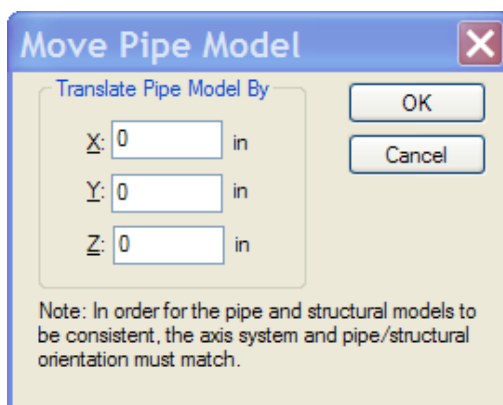
Hint: Piping mode is only available after the completion of a successful analysis.

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.

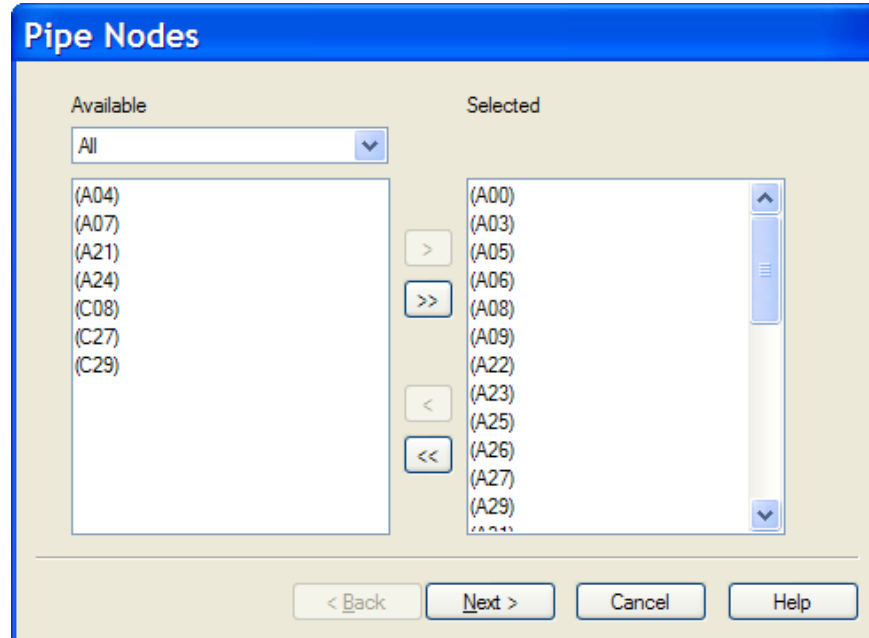
or

Select Support > Support Connection Wizard

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

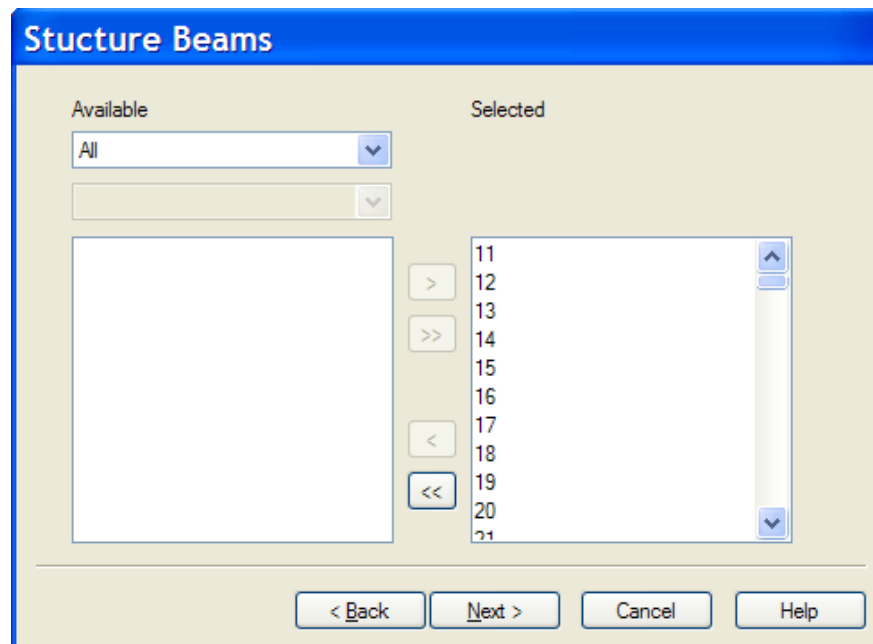


5. The Pipe Nodes dialog opens. Select which nodes will be used by adding them to the Selected list.



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.

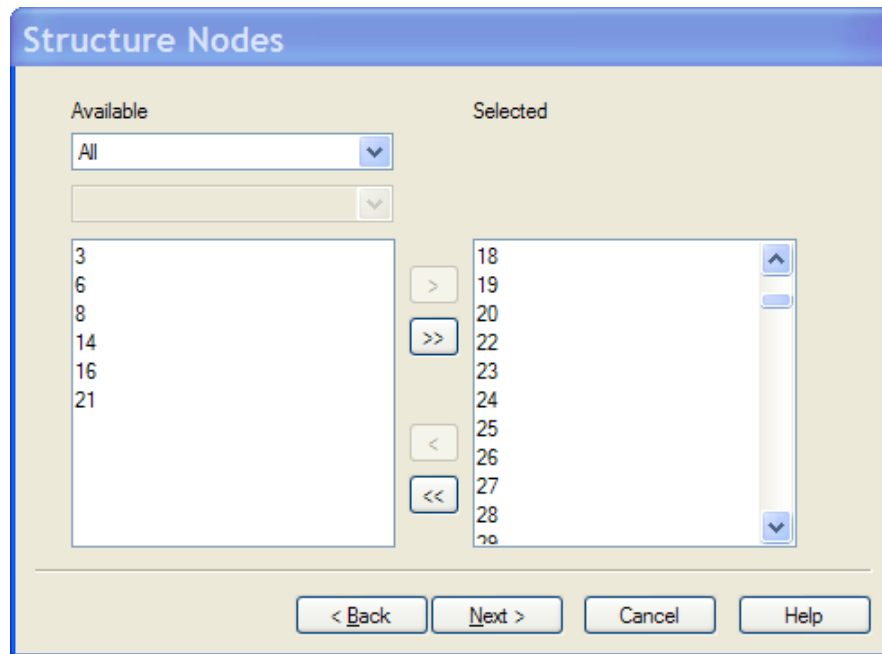
- The Structure Beams dialog opens. Select which structure elements will be used by adding them to the Selected list.



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.

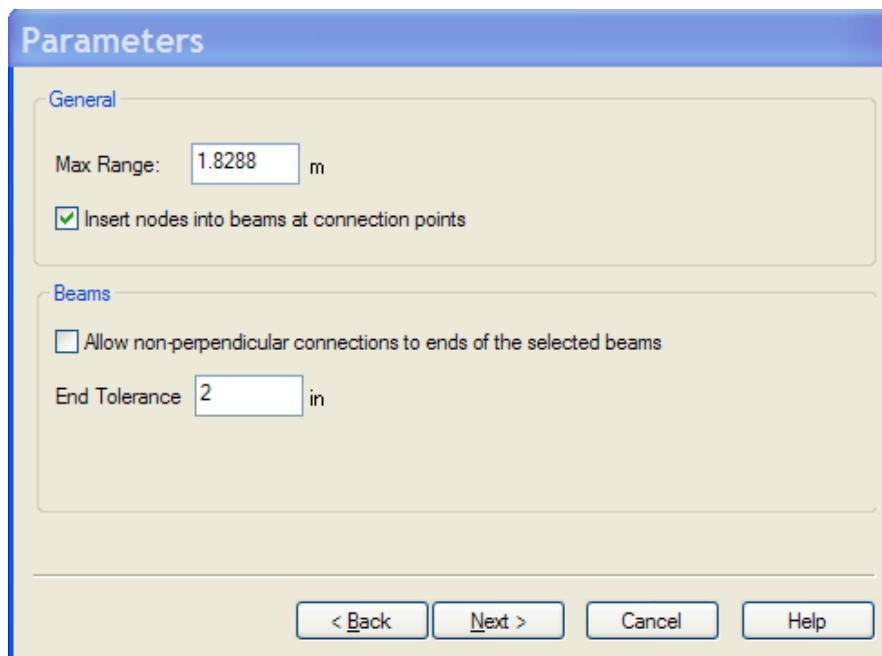
- The Structure Node dialog opens. Select which structure nodes will be used by adding

them to the Selected list.



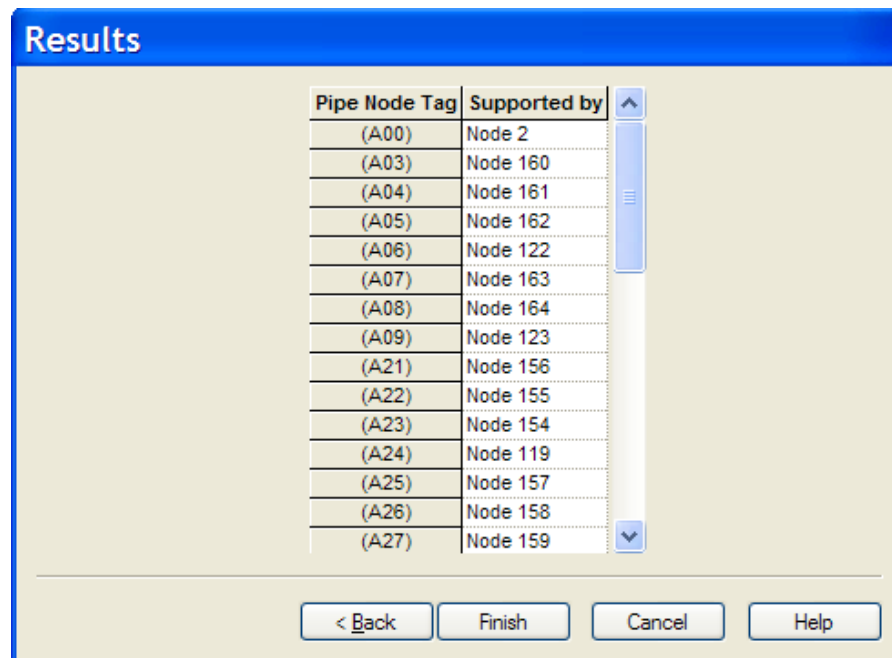
The filtering options are implemented with two combo boxes, one for the category and one to identify the subset within that category. Available filtering options are All, Group and View.

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).

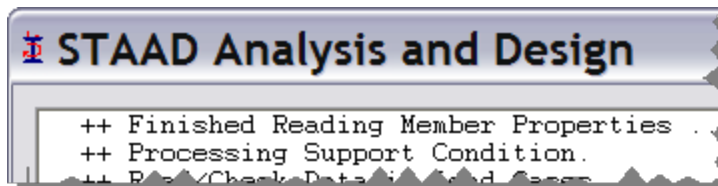
Section 3

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

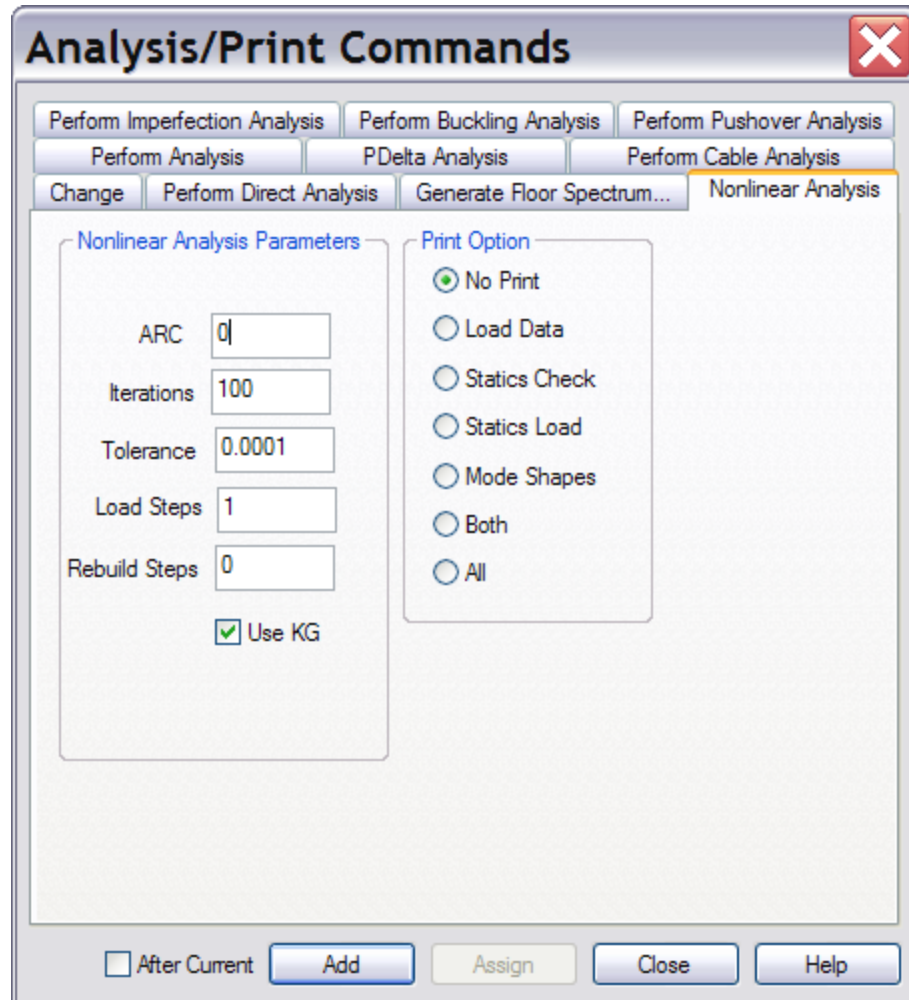
The range of analysis options has been supplemented with a new solution to account for nonlinear effects of moderate displacement and small strain. This solution holds for where the element distortion is small and small rotations are assumed.

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
KG	0	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 0" -> 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 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

Parameter Name	Default Value	Description
		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>	0.9	Cm value in local Y & Z axes
<u>CMZ</u>		(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 AVG , AVN , ATG , and ATN 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".
<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.

Parameter Name	Default Value	Description
<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)
<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

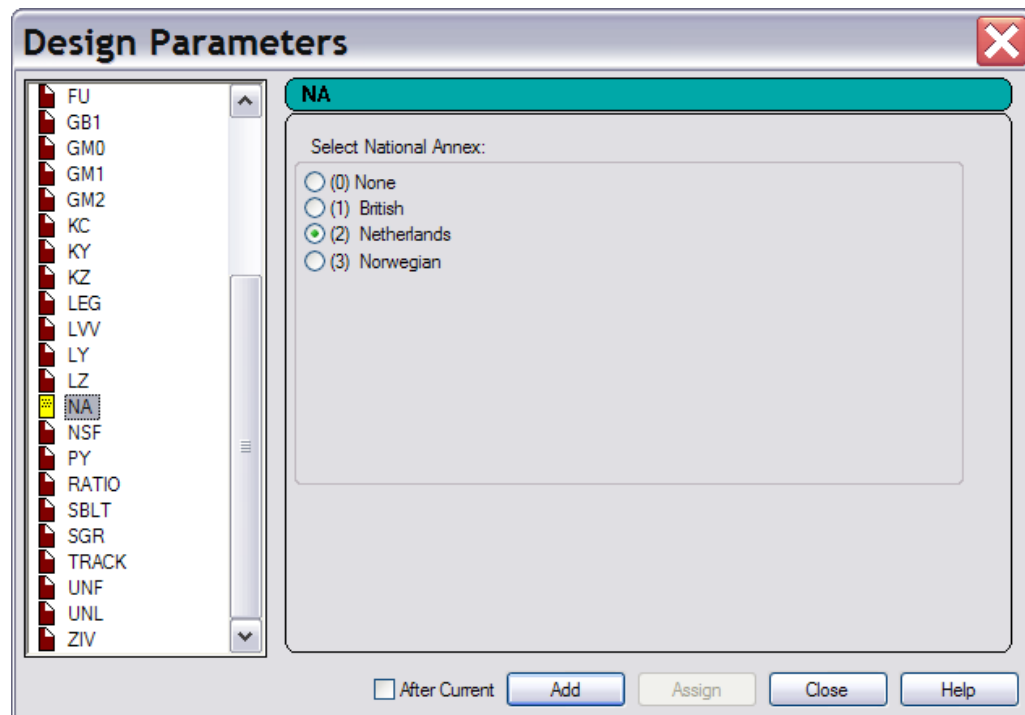
A new parameter, NA, that sets the default material gamma factors and any additional changes outlined in the country specific National Annex such as specific equations or methods.

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 ITB	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 EN3 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: fi 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

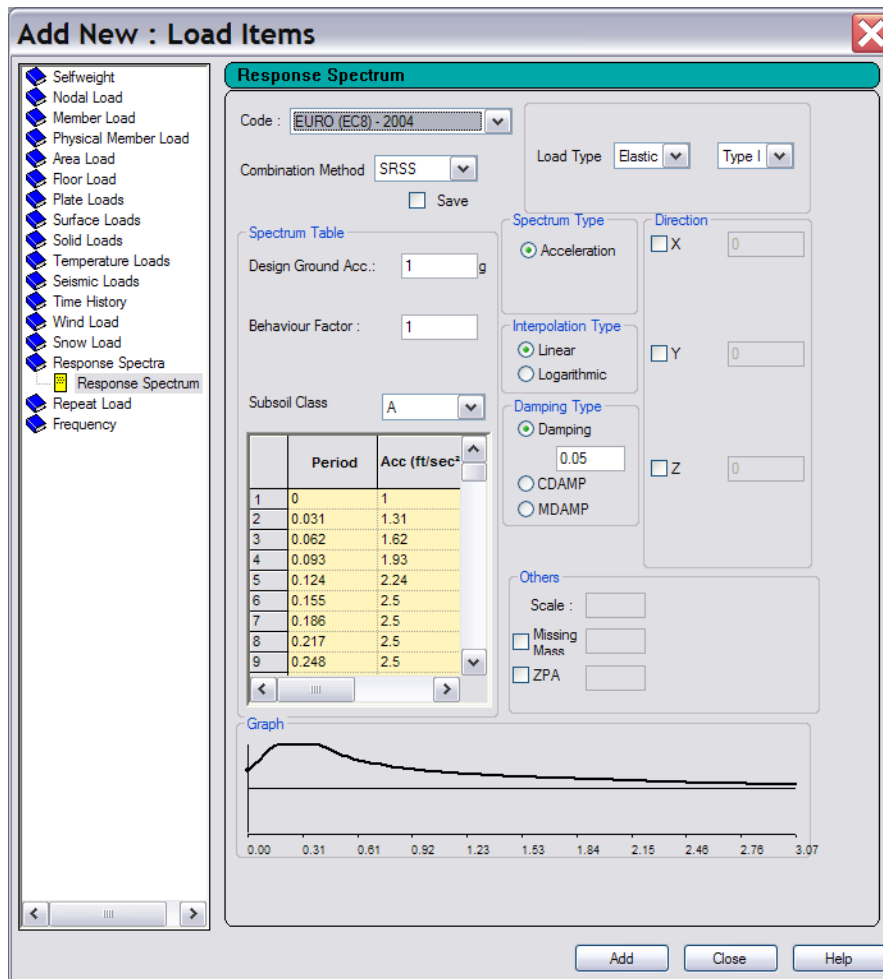
a) Structure Modeling

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)

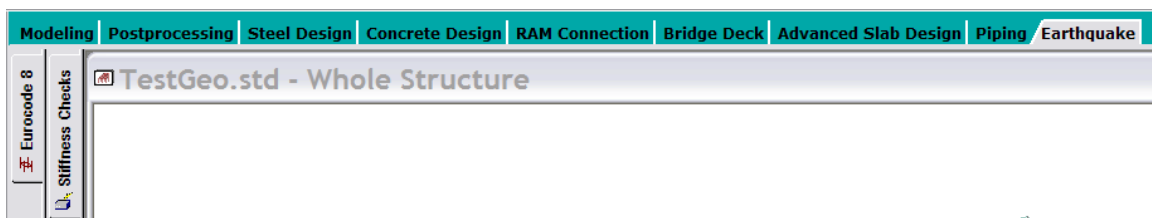
Note: In order to make use of the EC8 response spectrum load, a linear modal response spectrum analysis should be used. This is required to locate the center of mass which is needed to perform the associated geometry checks.

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

Collapse Check Setup

Design Code: EC8 - 2004 (dropdown) National Annex: UK Annex (dropdown)

Ductility level: DCM (dropdown)

Earthquake load case: L1: LOAD CASE 1 (dropdown)

Structural Type: Framed / Dual / Coupled wall system (dropdown)

Check reinforcement detailing rules for ductility

Curvature ductility factor: 1.0

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.

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 — Here you can find detailed resolutions and answers to the most common questions posted to us by you.
- Ask Your Peers — <http://communities.bentley.com/forums/5932/ShowForum.aspx> — Post questions in the Be Community forums to receive help and advice from fellow users.



Bentley Systems, Incorporated
685 Stockton Drive, Exton, PA 19341 USA
+1 (800) 236-8539
www.bentley.com

