

STAAD.Pro V8i (Build# 20.07.04.12) Release Report.....	2
Introduction.....	2
AD.V8i.0 New Features Affecting the General Program.....	3
AD.V8i.0.1 ProjectWise Integration.....	4
Purpose.....	4
Description.....	4
AD.V8i.0.2 CIS/2 Update.....	13
Purpose.....	13
Description.....	13
AD.V8i.1 Features Affecting the Pre-Processor (Modeling Mode).....	18
AD.V8i.1.1 ASME NF Steel Design Codes.....	19
Purpose.....	19
Description.....	19
AD.V8i.1.2 Floor Response Spectrum.....	20
Purpose.....	20
Description.....	20
AD.V8i.1.3 Russian Wind Loading.....	25
Purpose.....	25
Description.....	25
AD.V8i.1.4 Additional Standard Profile Databases.....	27
Purpose.....	27
Description.....	27
AD.V8i.2 Features Affecting the Analysis and Design Engine.....	28
AD.V8i.2.1 ASME NF.....	29
Purpose.....	29
Description.....	29
AD.V8i.2.2 Russian Wind Loading.....	30
Purpose.....	30
Description.....	30
AD.V8i.2.3 Floor Response Spectrum.....	34
Purpose.....	34
Description.....	34
AD.V8i.3 Features Affecting the Post-Processing (Results Mode).....	38
AD.V8i.3.1 Time History Animation.....	39
Purpose.....	39
Description.....	39
AD.V8i.3.2 Enhanced Plate Stress Results.....	41
Purpose.....	41
Description.....	41

STAAD.Pro V8i Release Report

Introduction

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

AD.V8i.0 New Features Affecting the General Program



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

AD.V8i.0.1 ProjectWise Integration

Purpose

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

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

Description

Four integration functionalities have been added. These are

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

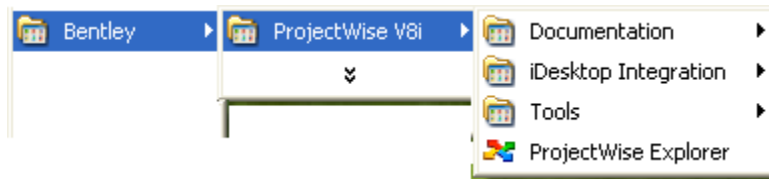
Note that access to all of these functionalities is available from ProjectWise sub-menu under the general File menu described below.

ProjectWise repository

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

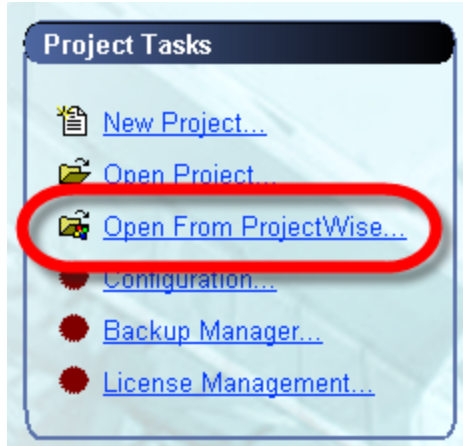
ProjectWise client

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

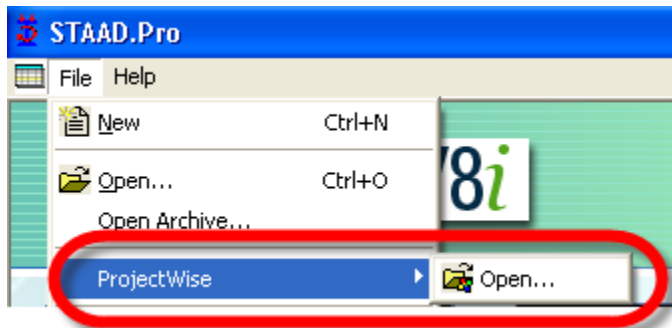


STAAD.Pro

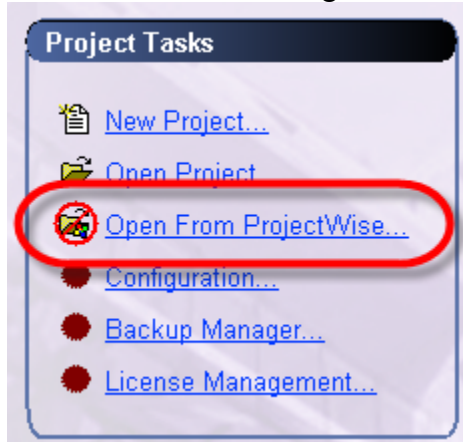
When STAAD.Pro is launched, the option to open and check out a STAAD.Pro STD file from a ProjectWise repository is made available from the Project Tasks on the Start Page thus:-



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

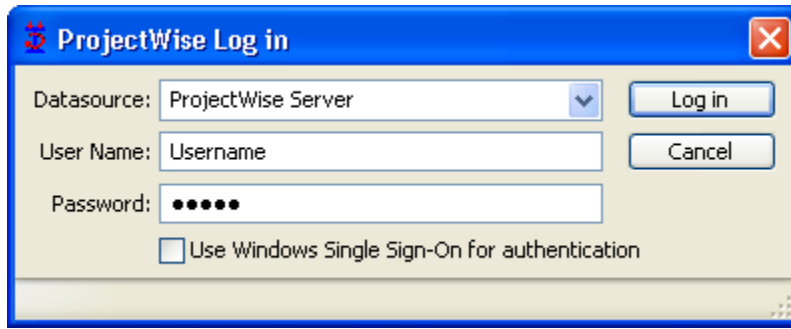


If a suitable ProjectWise client is not installed, then the link on the Start Page is shown as unavailable with a red line through the icon thus:-



Login

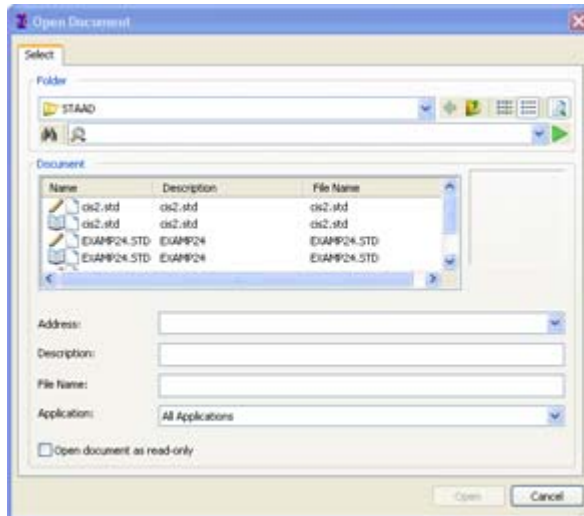
As authentication is required to access files stored on a ProjectWise repository, a login dialog allows the required details to be entered either with specific user credentials or by using the current windows login credentials thus:-



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

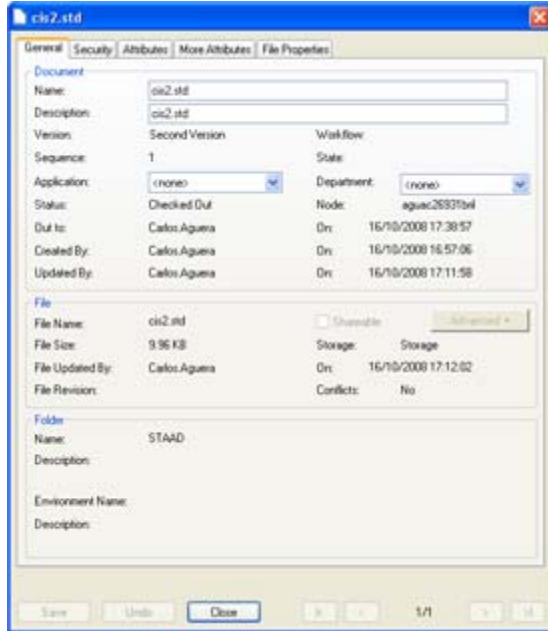
Opening a STAAD Model from a ProjectWise repository

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



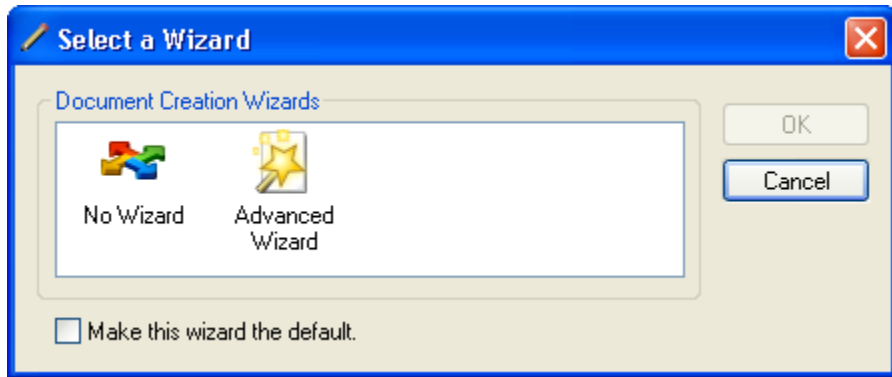
Note the significance of the icons next to the STAD filenames. These indicate the status of the file such as the current document, checked out to you, or locked as checked out to some other user. Refer to the ProjectWise documentation for a full description of each icon.

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

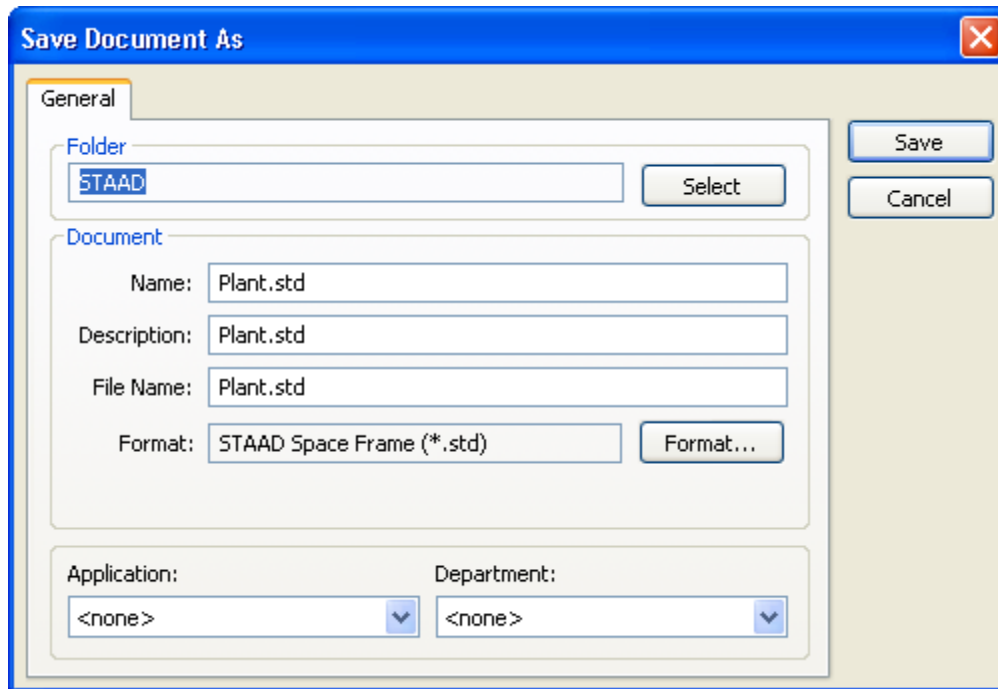


Saving a New model onto the Server

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



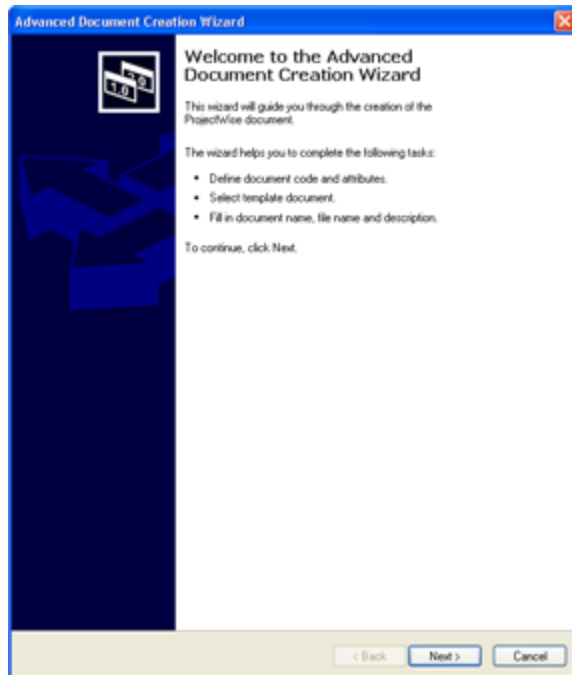
Selecting the option **No Wizard** offers the following dialog into which the file details can be entered.



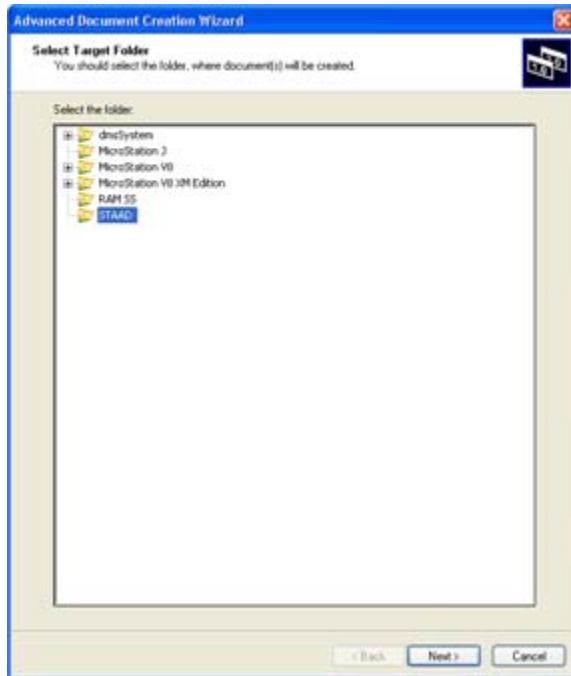
Clicking on Save, adds this model to the repository, but indicates its status as checked out until the file is closed in STAAD.Pro and the model checked back in.

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

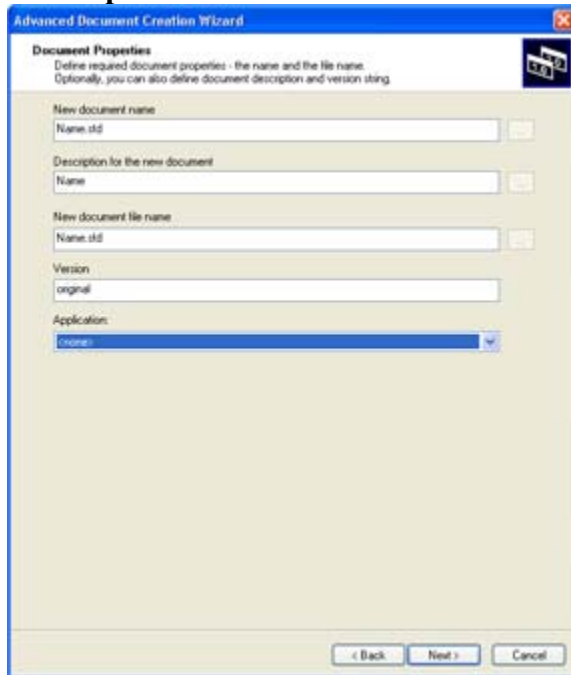
Advanced Wizard



Select target Folder:-



Document Properties:-



Create the document:-

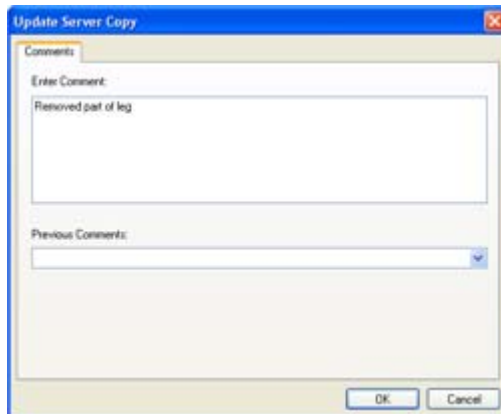


Creates the document and marks it as checked out until the file is closed and checked in to the server.

Saving changes of a Checked Out model Back on the Server

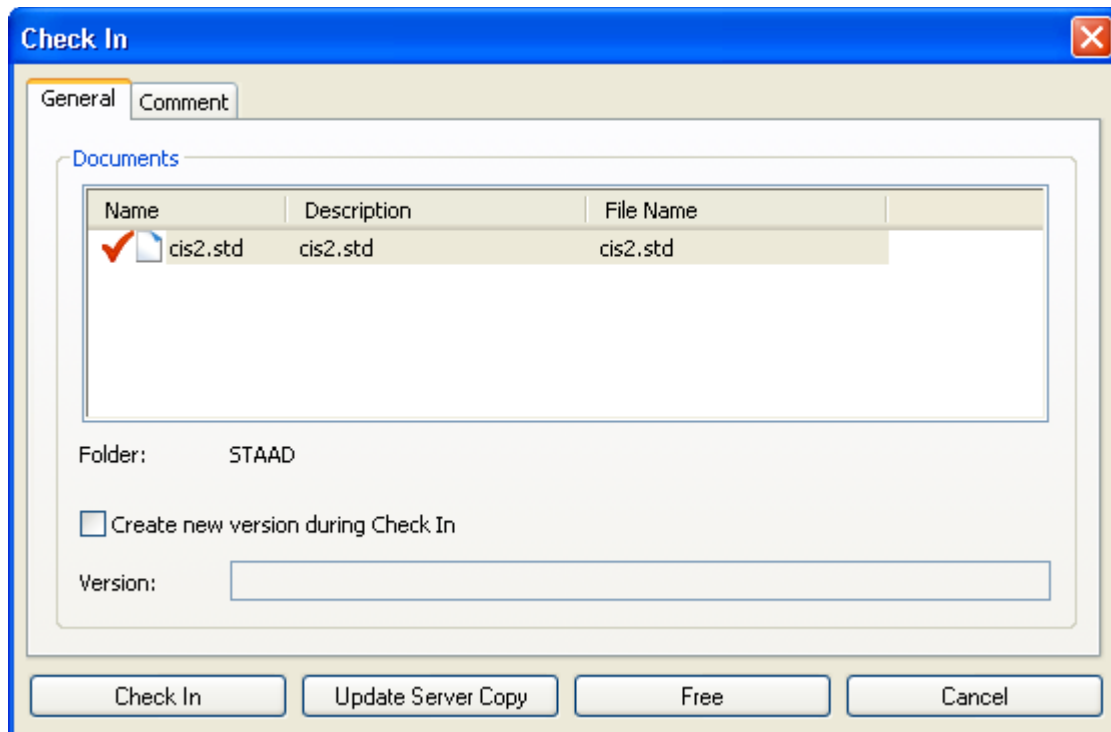
When a model is checked out from a ProjectWise server, selecting Save or Save As, only maintains a local copy of the model. There are two methods available to update a checked out model. Firstly, during a STAAD.Pro session, it is possible at any time to save any changes back on the server by selecting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

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



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

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



Check In.

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

Update Server Copy.

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

Free

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

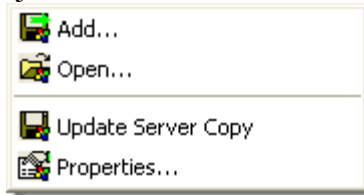
Cancel

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

Menu

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

File>ProjectWise>



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

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

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

Toolbar

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



Notes:-

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

AD.V8i.0.2 CIS/2 Update

Purpose

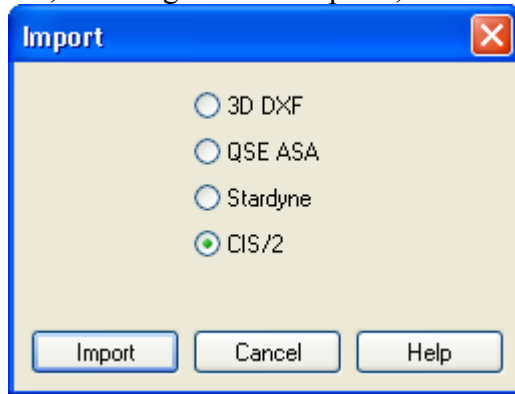
The STAAD.Pro tool to import and export models with the CIS/2 translator has been enhanced to work with international models and transferring models into 3D modeling such as SmartPlant 3D.

Description

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

IMPORT

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



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

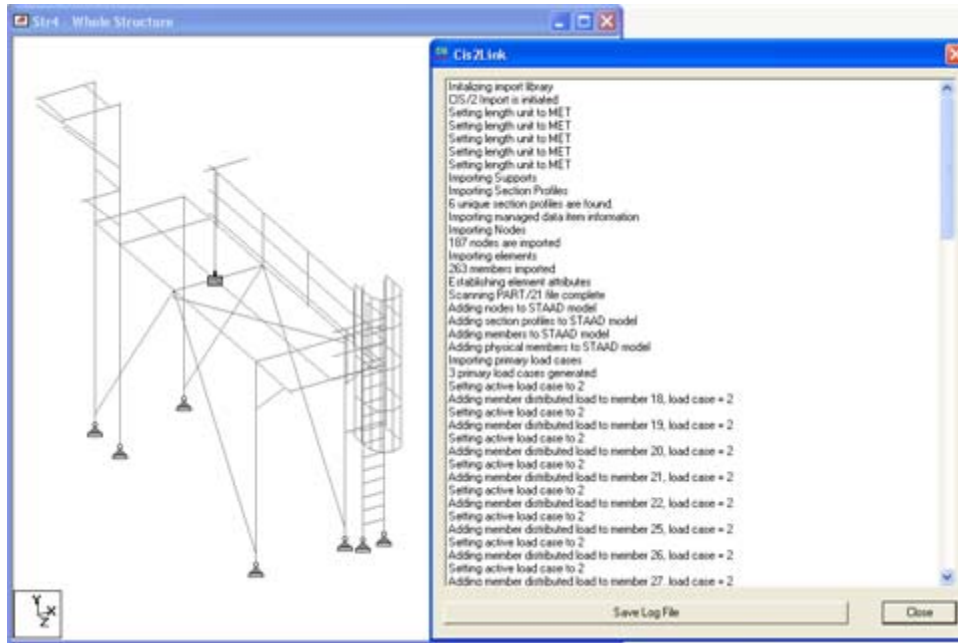
```
ISO-10303-21;
HEADER;
/* Generated by software containing ST-Developer
 * from STEP Tools, Inc. (www.steptools.com)
 */
FILE_DESCRIPTION(
/* description */ ('CIS2 Export File'),
/* implementation_level */ '2;1');
FILE_NAME(
/* name */ 'model',
/* time_stamp */ '2007-01-08T17:00:33+00:00',
/* author */ (''),
/* organization */ (''),
/* preprocessor_version */ 'ST-DEVELOPER v9',
```

```
/* originating_system */ 'STAAD.Pro 2007',  
/* authorisation */ '' );  
FILE_SCHEMA (('STRUCTURAL_FRAME_SCHEMA'));  
ENDSEC;  
DATA;  
#10=LOAD_COMBINATION_OCCURRENCE(1.39999997615814,#13,#264);  
#11=LOAD_COMBINATION_OCCURRENCE(1.39999997615814,#13,#265);  
#12=LOAD_COMBINATION_OCCURRENCE(1.60000002384186,#13,#266);  
#13=LOADING_COMBINATION('ULS',$,#2511);  
#14=LOAD_ELEMENT_DISTRIBUTED_CURVE_LINE(#265,'Member19UDLLoad1',$  
,#1089,  
$,$,.F.,.GLOBAL_LOAD.,.TRUE_LENGTH.,#214,#214,#64);  
#15=LOAD_ELEMENT_DISTRIBUTED_CURVE_LINE(#265,'Member20UDLLoad1',$  
,#1090,
```

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



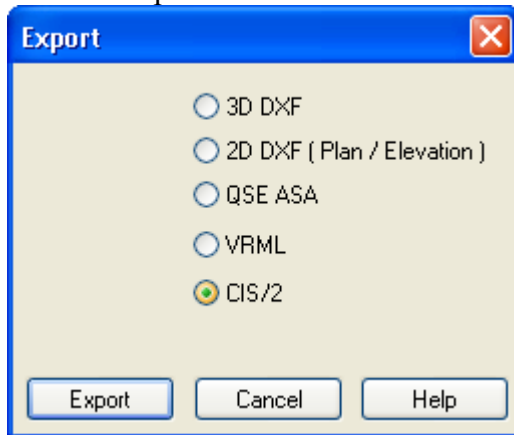
Clicking the Import button then runs the model import where the CIS/2 file is processed and the analysis model data is extracted to form the STAAD.Pro model.



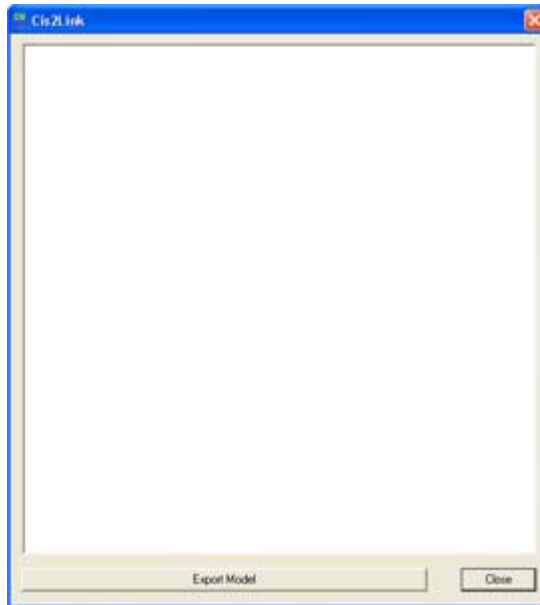
Note that as the CIS/2 file is being processed, a log file which identifies the data that has been utilized in the STAAD model is produced and displayed which can be saved as a text file for future reference.

EXPORT

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



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



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



Enhancements

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

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

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

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



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

AD.V8i.1.1 ASME NF Steel Design Codes

Purpose

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

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

- 1974
- 1977
- 1989
- 1998

Description

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



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

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

Notes

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

AD.V8i.1.2 Floor Response Spectrum

Purpose

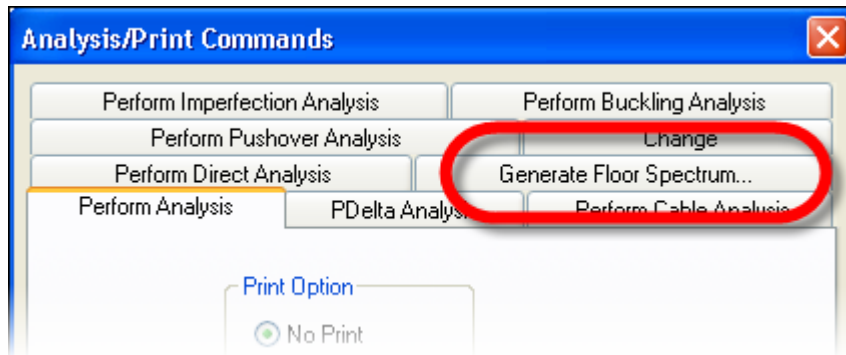
A new dynamic feature has been added that allows the extraction of a response spectrum from a collection of nodes that constitute a floor when subjected to a time history loading. This information can then be used in conjunction with equipment that will be supported by these floors and is often required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

Note :

This feature requires access to an Advanced Analysis license

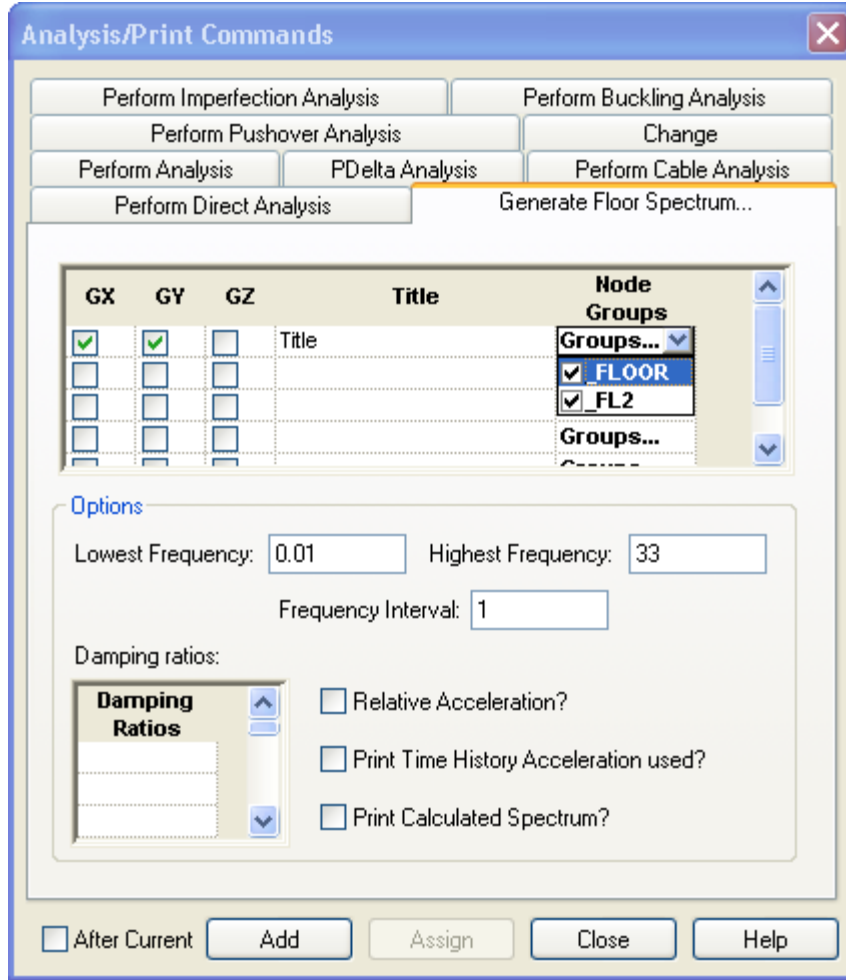
Description

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



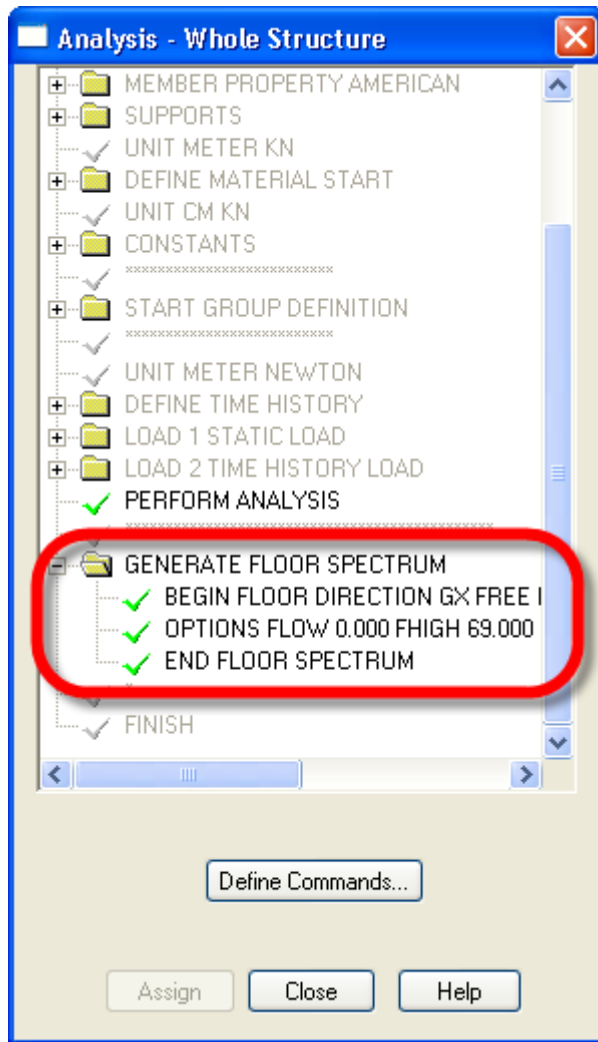
This command should follow immediately the definition of the analysis and will require defined groups of nodes which need to be defined first.

The following displays the layout of this sheet:-

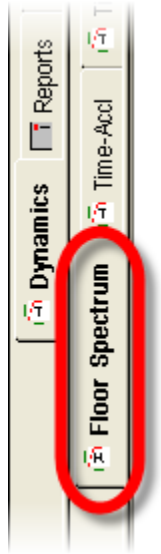


Note that the each line of settings constitutes one floor which can have one or more floor groups assigned. The resulting response spectra will be based on the collective responses of all the nodes in the selected groups.

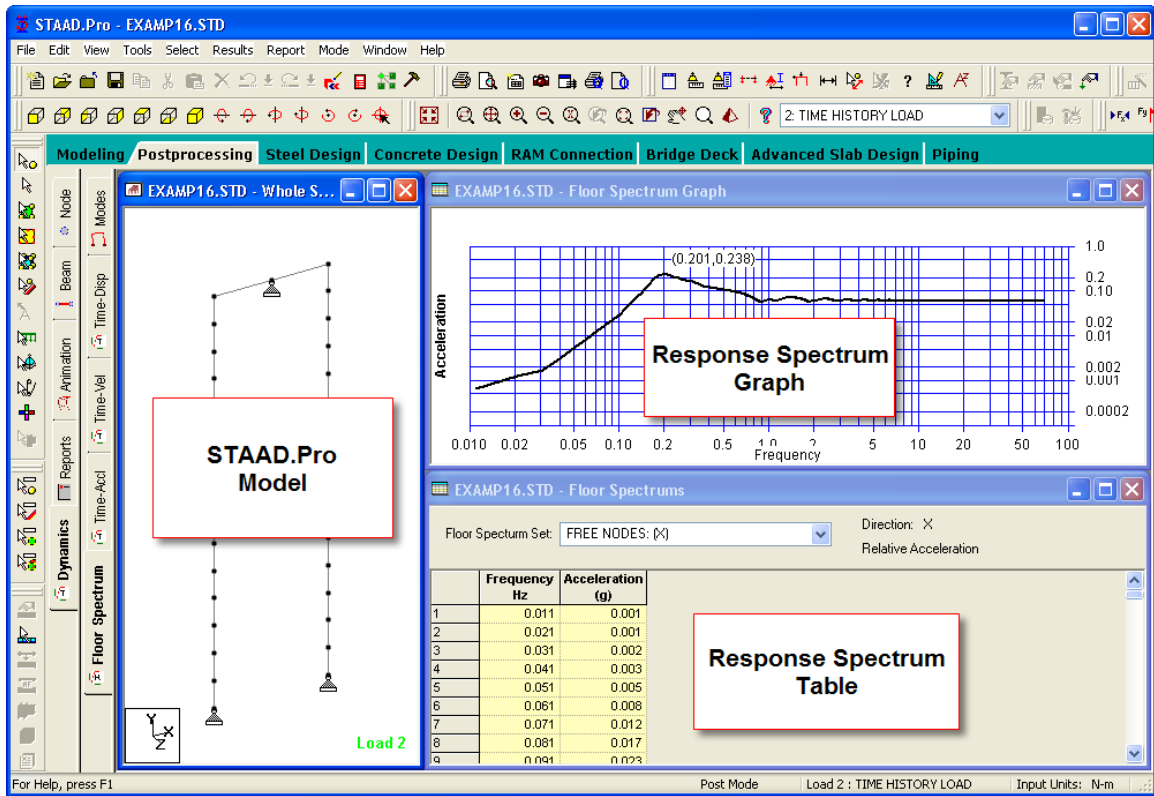
Once the required options have been set, click on the Add button to add the command set to the model which should appear in the Analysis Window thus:-



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



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



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

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

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

AD.V8i.1.3 Russian Wind Loading

Purpose

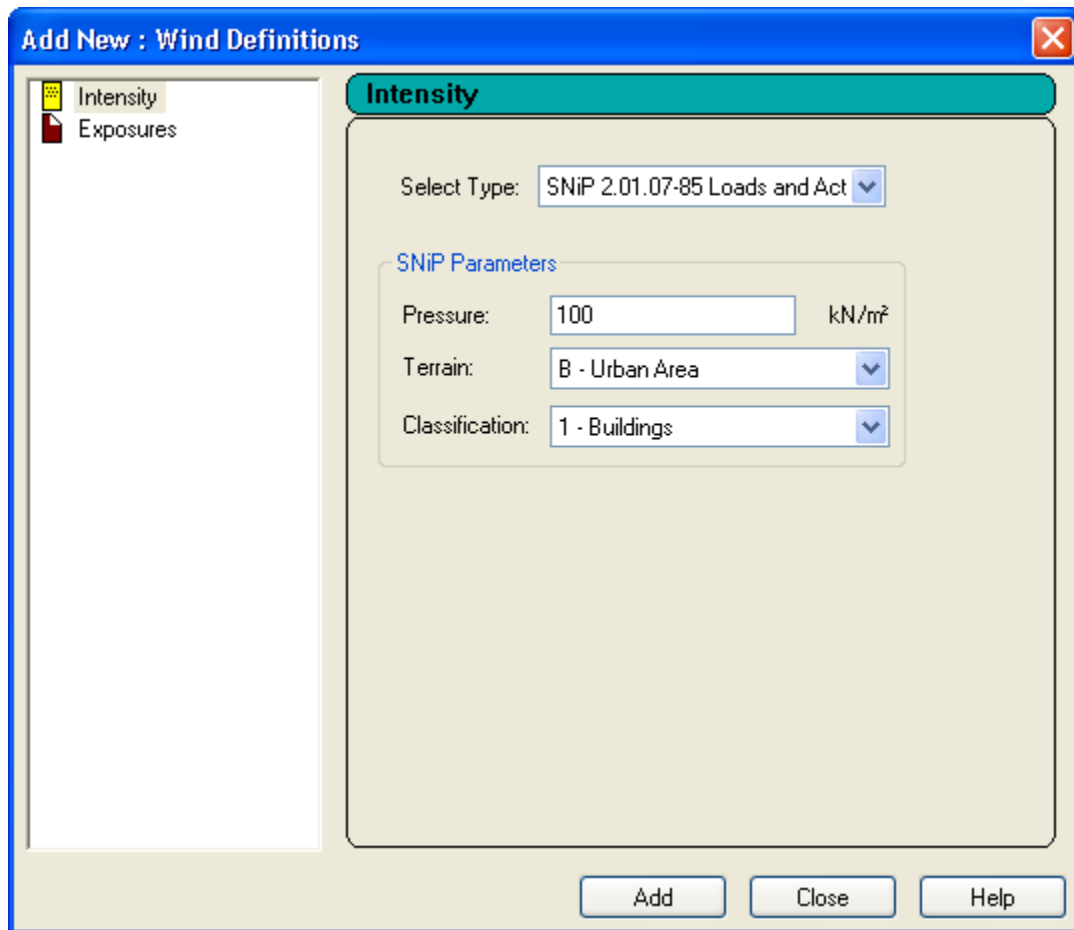
The wind loading as defined in the design code SNiP 2.01.07–85 “*Loads and Actions*” added in this version of STAAD.Pro can be added graphically to the model by modifications in the Loads Page.

Description

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

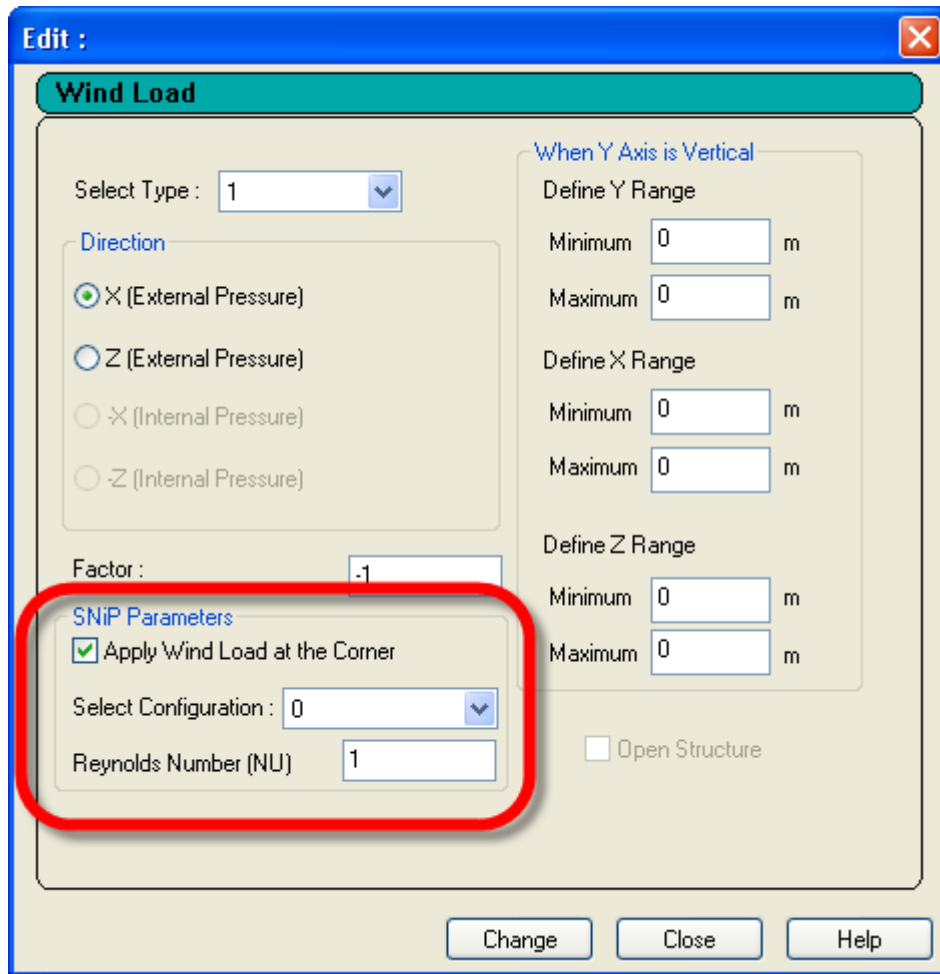
a) Wind Load Definition

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



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

b) Wind Load command



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

For technical details of this wind loading see section AD.V8i.2.2.

AD V8i.1.4 Additional Standard Profile Databases

Purpose

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

Description

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

Australian Cold Formed Steel Hollow Sections

Circular Hollow Sections:-

Galtube Plus®, 26.9mm to 76.1mm diameter

Tubeline, grades C250L0 (AS1163) and 350L0 (AS1163), 21.3mm to 457.0mm

Rectangular Hollow Sections

Duragal®, grade C450L0 (AS1163), 50x20mm to 150x50mm

Galtube Plus®, grade C350L0, 50x20mm to 75x25mm

Tubeline®, grade C350L0 (AS1163), 50x20mm to 250x150mm

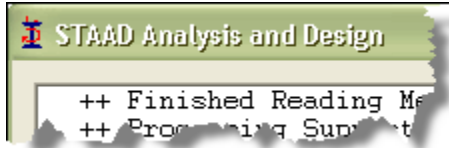
Square Hollow Sections

Duragal®, grade C450L0 (AS1163), 20x20mm to 100x100mm

Galtube Plus®, grade C350L0, 20x20mm to 65x65mm

Tubeline®, grade C350L0 (AS1163), 13x13mm to 250x250mm

AD.V8i.2 Features Affecting the Analysis and Design Engine



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

AD.V8i.2.1 ASME NF

Purpose

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

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

- 1974
- 1977
- 1989
- 1998

Description

For the list of parameters and commands including the default values, please refer to the International Design Codes manual, section...

For each steel member that is checked, the following checks are performed according to the clauses for that year:-

1. Slenderness check
2. Tension
3. Compression
4. Bending (a) About major axis, (b) About major axis
5. Shear
6. Combined Stresses

AD.V8i.2.2 Russian Wind Loading

Purpose

The wind loading commands have been enhanced to allow the creation of wind loading as defined in the in Russia by the design code SNiP 2.01.07–85 “*Loads and Actions*”.

The basic quantity in the wind loading is the characteristic (normative in Russian terminology) wind pressure. The reference wind velocity pressure corresponds to a 10-minute time-averaged velocity pressure at 10-metres height in a flat terrain, based on a 5-year return period. This wind pressure is the static component of the wind load. The total wind pressure consists of static and fluctuating components. If the structure is sufficiently flexible, according to the code provisions, the dynamic structural response to the fluctuating wind component must be taken into account.

The updated wind loading commands automatically perform both the aerodynamic and structural load analysis of vibration-susceptible buildings and structures.

Description

The wind loading commands have been updated to support the wind loading as defined in the Russian design code. This requires the creation of the following commands.

General Format:


There are three parts to creating a Russian wind load on a STAAD Model

a) Definition of the wind load requirements

This should occur before the definition of the first load case:-



Where:-

 - is the characteristic value of wind pressure, always positive. All values of the pressure within the range of corresponding SNIIP table are valid.


TERRAIN is terrain roughness category:-

A – Coastal Zone

B – Urban

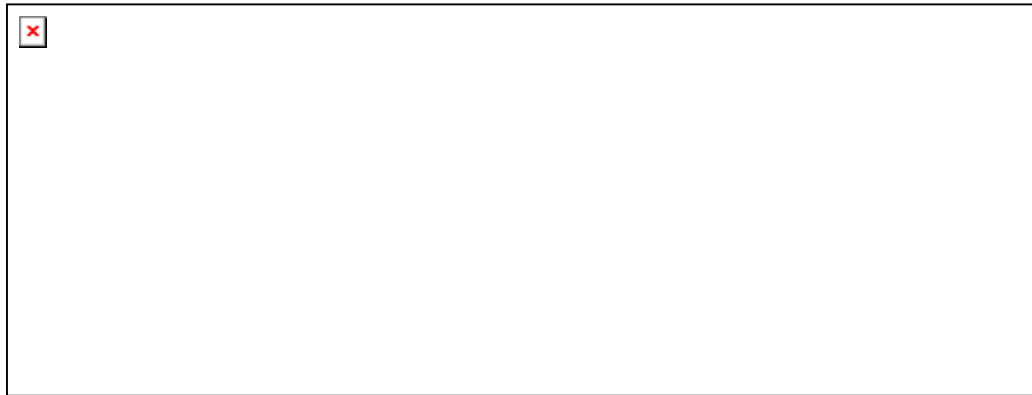
C - Large City according to the SNIIP classification.

CLASSIFICATION: 1 – for prismatic building structures,
2 – for general concrete structures,
3 – for general open framed steel structures.

EXPOSURE: factors , multiplying wind pressure influence areas. For stick-type structures with compact members aligned along the vertical line (e.g, chimneys, elevated water tanks), influence areas are set to unity, and exposure factors are aerodynamic diameters of members.

b) Application of the wind load definition within a load case.

Each load case which is to include a Russian wind load specification should include the following:-



Where:-

X, Z – define the wind direction.

f - wind pressure factor; negative value is used to define a reverse wind direction

OBLIQUE - defines wind to be applied at 45°. (note this parameter is only applicable for rectangular buildings.

if – configuration parameter for prismatic buildings, range 0-12;

0 - prismatic building structure - Rectangular building- Outstanding architectural details on the left façade.

- 1 - prismatic building structure - Rectangular building- Both side façades are smooth
- 2 - prismatic building structure - Rectangular building- Outstanding architectural details on the right façade
- 3 - prismatic building structure - Rectangular building- Outstanding architectural details on both side façades
- 4 - prismatic building structure - Rectangular building- Triangular building
- 5 - prismatic building structure - Rectangular building- Rhombic building
- 6 - prismatic building structure - Rectangular building- Number of vertices of polygonal building, not more than 12
- 7 - prismatic building structure - Less than 3 — rectangular building
- 8 - prismatic building structure - 3 — triangular
- 9 - prismatic building structure - 4 — rhombic
- 10 - prismatic building structure - More than 4 — polygonal
- 11 - framed RC structure
- 12 - lattice steel structure

(note, it is ignored for non-building structures, but cannot be omitted from the command.)

f_{NU} – wind pressure correlation coefficient. If parameter is omitted or is exactly 1, a computed value is used instead. For rectangular buildings, the correlation coefficient is always calculated automatically, thus any specified value will be ignored.

Additionally, as part of the load force is determined from the dynamic behavior of the model, the first load case that includes a Russian Wind Load specification also needs to include all the masses that can cause the structure to vibrate from which a dynamic analysis will be used to extract the mode shapes.

Note that this in effect becomes a dynamic load case, thus if the effects of these forces are to be combined with the effects of static load cases, then this should be done using a LOAD COMBINATION which references the required load case results with the appropriate factors.

c) Cut-Off Frequency or Mode Shape

As the analysis will require extraction of eigen solutions to determine the dynamic effects of the wind loading, the number of modes to be used will also affect the results. Thus setting the command CUT OFF FREQUENCY or CUT OFF MODE should be considered and specified as required prior to the definition of the load cases.

If the cut-off command is omitted, six mode shapes are computed by default.

For more information on these commands, see the Technical Reference manual section 5.30.1 Cut-Off Frequency, Mode Shapes or Time.

Example

```
DEFINE WIND LOAD
TYPE 1
SNIP PRESSURE 0.38 TERRAIN A CLASSIFICATION 1
EXP 0.5 JOINT 1 3 5 7 9 11
*
LOAD 1 LOADTYPE WIND TITLE Wind load in the +ve X direction
WIND LOAD X 1 CONFIG 0 NU 1 TYPE 1
* Mass model required in first wind load case
JOINT LOAD
3 TO 6 FZ 62.223
9 TO 12 FZ 62.223
*
LOAD 2 LOADTYPE WIND TITLE Wind load in the -ve X direction
WIND LOAD X -1 CONFIG 0 NU 1 TYPE 1
* No mass model or additional loads in this load case
*
LOAD 3 LOADTYPE WIND TITLE Wind load in the +ve Z direction
WIND LOAD Z 1 CONFIG 0 NU 1 TYPE 1
* No mass model or additional loads in this load case
*
LOAD 4 LOADTYPE WIND TITLE Wind load in the -ve Z direction
WIND LOAD Z -1 CONFIG 0 NU 1 TYPE 1
* No mass model or additional loads in this load case
*
LOAD 10 LOADTYPE DEAD TITLE Selfweight load case
SELF Y -1 ALL
*
LOAD COMBINATION 100 Wind plus selfweight
1 1.0 10 1.0
```

Notes:-

This command cannot be used with models that have been defined with the SET Z UP command.

For more information on wind loading see sections 5.31.3 Definition of Wind Load and 5.32.12 Generation of Loads of the Technical Reference Manual.

AD.V8i.2.3 Floor Response Spectrum

Purpose

The following commands have been added in order to allow the response spectrum of floors to be extracted from a time history analysis

Description

This command is used to specify the calculation of floor and/or joint spectra from time history results. The Floor Response Spectrum command must immediately follow an analysis command. That analysis can only contain a single time history load case.

General Format:

The format of the Floor Spectrum command is such that 3 sets of data are required thus:-

1) Initiate command

The command is initiated with the line:-

GENERATE FLOOR SPECTRUM

2) Specify Floor Groups

Next, identify the floors which will have spectrum curves generated either by referencing a NODE GROUP (see section 5.16) or explicitly listing the list of nodes that constitute the floor and the direction.

BEGIN FLOOR DIRECTION * $\left\{ \begin{array}{c} \text{GX} \\ \text{GY} \\ \text{GZ} \end{array} \right\}$ { TITLE }

$\left\{ \begin{array}{l} \text{joint_group} \\ \text{joint list} \end{array} \right\}$

Where:-

GX, GY and GZ specify up to 3 global directions for which acceleration vs frequency spectrums will be generated for this floor.
 Title – is an optional description of up to 50 characters, for this floor that will be displayed on the graphs in post processing.

The above can be repeated as many lines as necessary to specify all of the groups and directions that are needed.

3) Specify Options

Next enter additional parameters used in the Floor Spectrum calculations.

OPTIONS ({ **FLOW** **f1** }
 { **FHIGH** **f2** }
 { **FDELTA** **f3** }
 { **DAMP** **f4** }
 { **RELATIVE** })

This command may be continued over a number of lines by ending each line except the last with a hyphen.

Required options:

FLOW= Lowest frequency to be in the calculated spectrum. FLOW should be greater than 0.01 Hz.

FHIGH = Highest frequency to be in the calculated spectrum.

FDELTA = The spectrum will be calculated at FDELTA intervals from FLOW to FHIGH

DAMP = Up to 10 damping values may be entered. One spectrum will be generated for each damping value for each global direction requested for each floor defined. The spectrum will be based on these modal damping ratios.
 3% damping should be entered as 0.03. The default is 0.05.

RELATIVE= If there is ground motion defined and you want the spectrums to be based on the relative acceleration of the floor to the ground acceleration, then include the RELATIVE parameter.

Optional parameters.

THPRINT =0, no print;
=2, Print the time history acceleration being used in each spectrum calculation.

SPRINT = Print the calculated spectrum.

4) End the command

Finally, the command is completed with the line:-

END FLOOR SPECTRUM

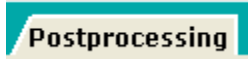
A successful analysis will result in the output file containing the base shear in the global X, Y and Z directions.

Example

```
DEFINE TIME HISTORY
TYPE 1 FORCE
0 -20 0.5 100 1 200 1.5 500 2 800 2.5 500 3 70 16 0
ARRIVAL TIME
0
DAMPING 0.075
*
LOAD 1 LOADTYPE SEISMIC TITLE Time History Load case
* Mass model required
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
JOINT LOAD
1 TO 6 FX 62.223 FY 62.223 FZ 62.223
* Time loads
TIME LOAD
2 FX 1 1
PERFORM ANALYSIS
*
GENERATE FLOOR SPECTRUM
BEGIN FLOOR DIRECTION GX GZ Ground Motion
_FL1
_FL17
BEGIN FLOOR DIRECTION GX GZ Floor 18 A/C unit 36
_FL18
OPTIONS FLO 0.5 FHI 35.0 FDEL 0.1-
DAMP 0.03 0.05 0.07 -
```

RELATIVE
END FLOOR SPECTRUM

AD.V8i.3 Features Affecting the Post-Processing (Results Mode)



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

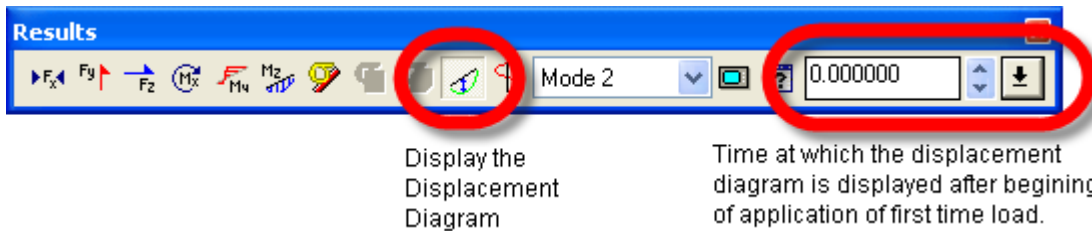
AD.V8i.3.1 Time History Animation

Purpose

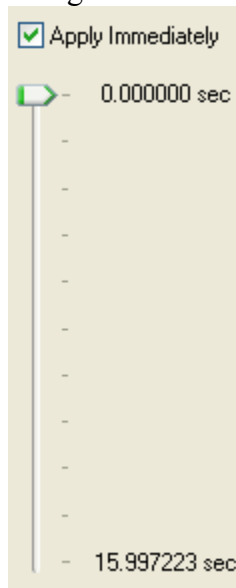
In order to visualize the displacement that occurs on a model during the application of a time history load, a new toolbar icon has been added which allows the displacement at a specific time instance to be displayed.

Description

The results toolbar has been updated to include a new option which is activated when the displacement diagram icon has been clicked and the current load case contains time history loading:-



For a time history load case the displacement that is displayed will be defined by the time instance entered in the above edit box. Alternatively the time can be set from a sliding scale by clicking on the button with the down arrow icon which displays the following option setting:-



The slider scale is based on the overall time for which the time history analysis has been performed.

The displacement is produced at the time at which the slider arrow is dragged to and the mouse button released.

If the 'Apply Immediately' option is selected, then the application will attempt to render the displacement diagram dynamically as the slider is dragged up and down the scale of time. For large models this may prove to be too demanding on the graphics system and left un-checked which means that the displacement diagram will be produced at the time step at which the slider arrow is released.

Note that the displacement diagram will be set to the scale as defined in the Scales sheet in the View>Structure Diagrams... dialog box.

Additionally note that it is possible to view the time v displacement of individual nodes by clicking on the Dynamics>Time-Disp in the post processing mode which is only available for models that include time history.

AD.V8i.3.2 Enhanced Plate Stress Results

Purpose

In order to provide additional understanding of stress distribution in finite element models, STAAD.Pro has expanded the sets of results that can be reported for each element both in the Plate Centre Stress Table and graphically using the Plate Stress Contour.

Description

To view the results data of plate elements, enter the Post Processing (Results) Mode, and click on the Plate Contour Page on the left menu.

Plate Centre Stress Table

The new Combined Stresses sheet in the Plate Centre Stress table provides resolved stresses for the top (positive local Z) and bottom (negative local Z axis) surface for each plate element, referred to as the Top Combined Stresses and Bottom Combined Stresses respectively.

Plate	L/C	Top Combined Stress			Bottom Combined Stress		
		Comb. SX psi	Comb. SY psi	Comb. SXY psi	Comb. SX psi	Comb. SY psi	Comb. SXY psi
1	1 ALL	142.747	142.737	101.261	-142.671	-142.681	-101.215
2	1 ALL	514.879	101.366	218.581	-514.854	-101.376	-218.579
3	1 ALL	968.391	192.264	200.708	-968.705	-192.306	200.750

Note that if the sheet title Combined Stresses is not visible on the top of the table, then click on the right arrows displayed on the left of the table header to scroll the table sheets which will display the title.

The combined stresses are calculated thus:-

Top

$$\begin{aligned}
 SX_{top} &= SX + MX/S \\
 SY_{top} &= SY + MY/S \\
 SXY_{top} &= SXY + MXY/S
 \end{aligned}$$

Bottom

$$\begin{aligned}
 SX_{bottom} &= SX + MX/S \\
 SY_{bottom} &= SY + MY/S \\
 SXY_{bottom} &= SXY - MXY/S
 \end{aligned}$$

where

$$S = t^2/6 t$$

$t = \text{average plate thickness}$

Plate Stress Contour

The Plate Stress Contour sheet of the Diagrams dialog has been enhanced to allow visualization of these stresses. The six new stress results are available in the Stress Type pull down menu thus:-

