Strand7®

FINITE ELEMENT ANALYSIS

What's New in Strand7 R3.1.2

DECEMBER 2022

CONTENTS

Introduction 1
Preferences 1
2D MOUSE IN VISUAL TAB 1
3D MOUSE IN VISUAL TAB 1
GRAPHICAL USER INTERFACE 2
VISUAL TAB 2
DIALOGS TAB 3
Printing 3
Fonts 3
Create Options – Number 4
Grids 5
Cell-drag Square Double-click 5
Equation Editor 5
VISUAL Tab 6
Keyboard Shortcuts 6
Variables 6
View Settings 6
Entity Display 7
Attribute Display 9
View/Multiview 9
Default Group 10
Global/UCS 11
Global/Groups 11
Global/Entity Sets 12
Tools 12
Copy/Paste 15
Utility/Make BXS 15
TEXT Tab 16
Layouts/Plate RC 17
Layouts/Cavities 18
Comparison of Stiffness Matrix Options 19
Layouts/Paths 20
Solvers 21
Parameters 21
Results 22
Settings 22
Diagram Options 22

Beam End Forces in Peek and LISTINGS....23
Beam Section Contouring in Peek....24
Beam Diagrams for Limit Envelopes....24
Graphs....26
New Applications....27
St7APIDiagnosticsTool....27
St7SetAPIPath....27
New Network Licencing Mode....27

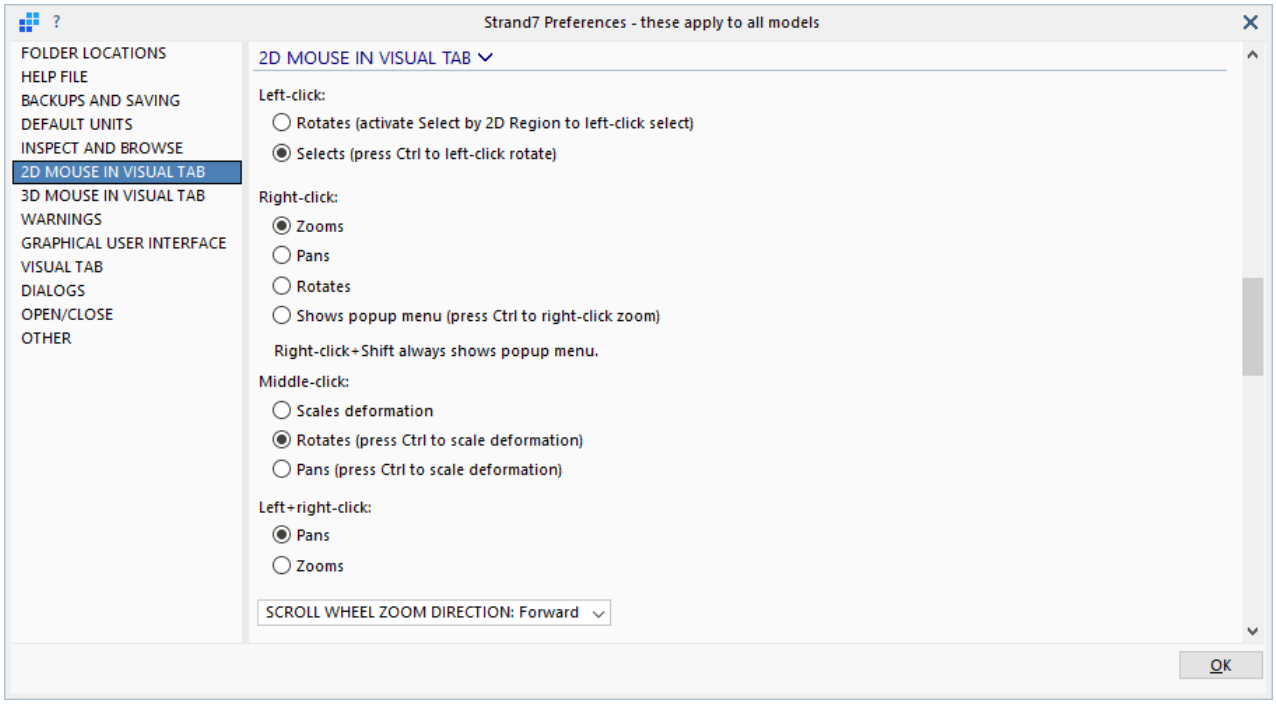
INTRODUCTION

This document summarises the main changes in Strand7 R3.1.2 compared with Strand7 R3.1.1. More details about each of the following items can be found in the Online Help.

PREFERENCES

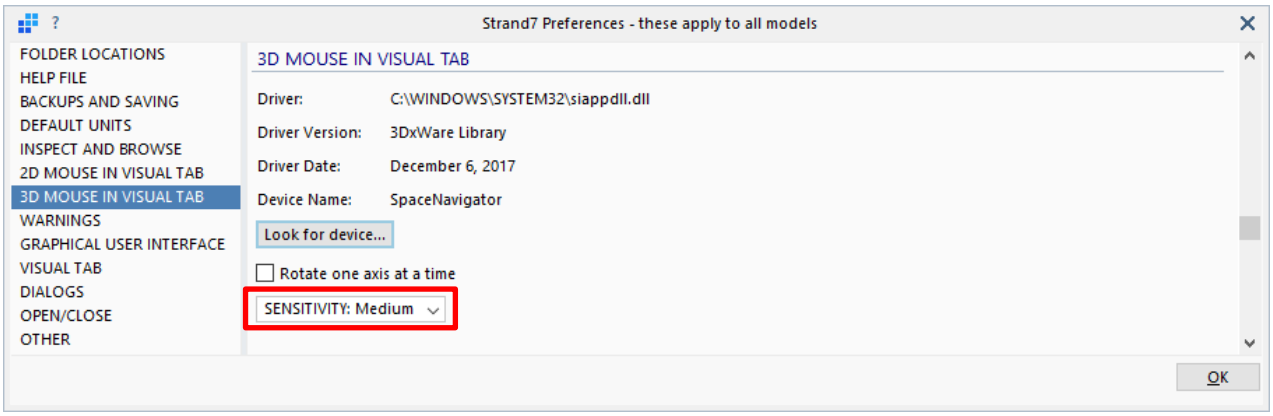
2D MOUSE IN VISUAL TAB

Greater customisation for the use of mouse clicks and movement is provided. Any of the three mouse buttons can now be assigned to rotate the view, with the others then customised for zoom and pan actions according to user preference. In addition, an option allows for the left-click action to be permanently assigned to **Select by 2D Region** (i.e., to point-and-click select). Using this setting together with (say) **Middle-click: Rotates**, point-and-click selection is always available as is view rotation, without needing to click any icons or hold the Ctrl key to activate either.



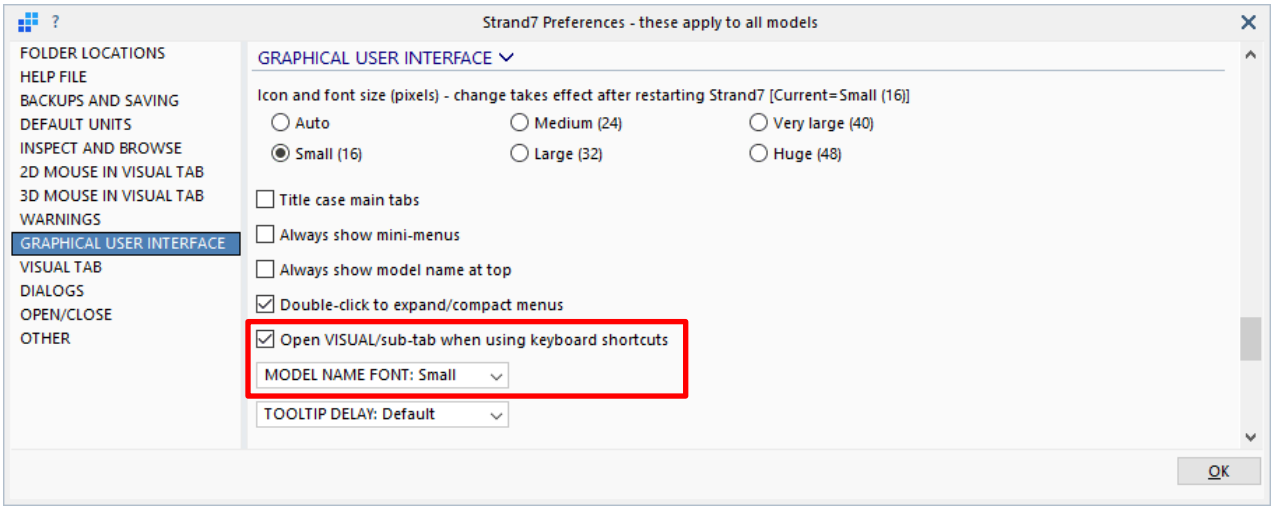
The right-click popup menu in the model window is optional, depending on the assignment of the right-click action. However, shift+right-click will now always show the popup menu, irrespective of the default action assigned to right-click.

3D MOUSE IN VISUAL TAB



- **SENSITIVITY** selects the rate of model rotation with respect to the mouse movement (Low, Medium or High).

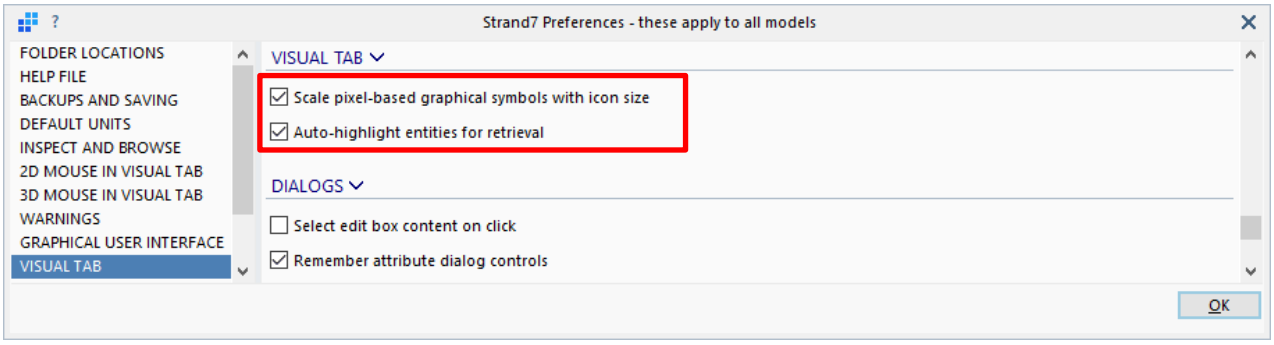
GRAPHICAL USER INTERFACE



- **Open Visual/sub-tab when using keyboard shortcuts** controls the automatic opening and closing of sub-tabs in the **VISUAL** tab when a keyboard shortcut is used. If the option is set and a keyboard shortcut is used to activate a menu from a sub-tab that is not currently open, that sub-tab is automatically opened by the keystroke. For example, if **VISUAL/Tools** is open and the keyboard shortcut to assign a plate attribute is pressed (i.e., **shift+alt+P**), the sub-tab automatically opens to **VISUAL/Attributes** and the dropdown menu for plate attributes is shown. If the option is not set, the dropdown menu for plate attributes is still shown by the keystroke, but the sub-tab does not change.
- **MODEL NAME FONT** selects from one of three font sizes (Small, Medium or Large) for the display of the model name at the top of the model window.

VISUAL TAB

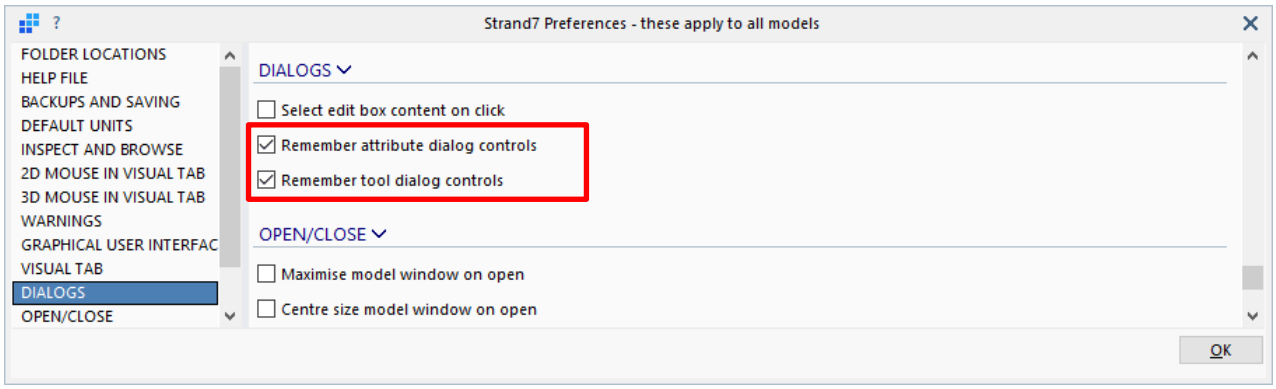
This is a new section of **Preferences** with two options.



- **Scale pixel-based graphical symbols with icon size** is used in conjunction with the **GRAPHICAL INTERFACE Icon and font size** setting. If the option is set, pixel-based symbols are proportionally scaled according to the selected icon size. For example, a 1-pixel wide line with **Small** icons becomes 1.5 pixels wide with **Medium** icons, 2.0 pixels wide with **Large** icons, and so on. This option produces better graphics when using the larger icon and font sizes on high-resolution monitors. If the option is not set, a 1-pixel wide line remains one pixel wide for all icon sizes.
- **Auto-highlight entities for retrieval** is used to highlight an entity as the mouse moves over it, in certain situations, to identify the entity that will be selected upon clicking the mouse. For example, while inserting an element via **Insert/Element**, the candidate entity (node, vertex or grid number) is highlighted as the mouse cursor passes over them in anticipation of selecting the point via the mouse click. If the option is not set, no candidate entity feedback is given as the mouse passes over an entity, but the entity will still be selected by the mouse click.

DIALOGS TAB

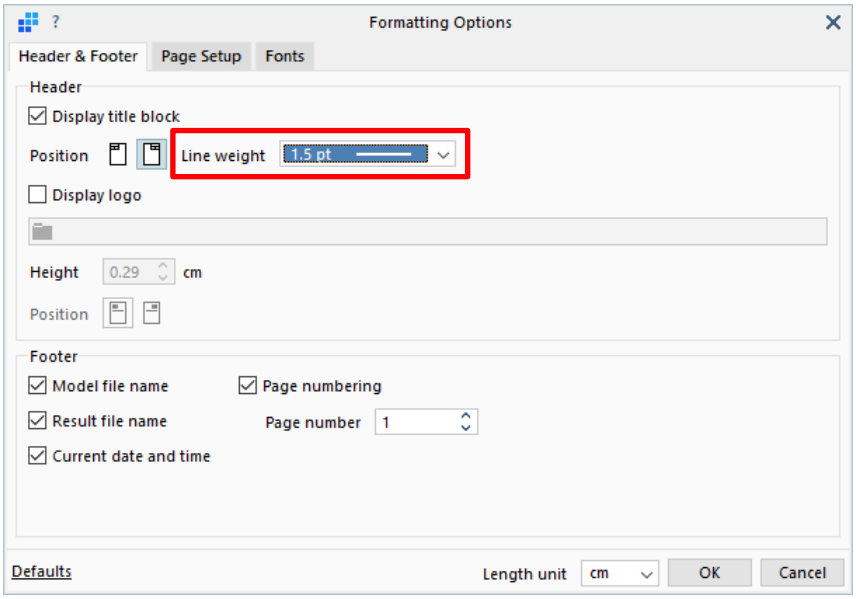
This is a new section of **Preferences** that contains the previously available **Select edit box content on click** option and adds two new options.



- **Remember attribute dialog controls** is used to store the values entered in the edit boxes of the attribute dialogs (e.g., **Attributes/Plate/Face Normal Pressure**) so that those values reappear the next time the attribute dialog is opened. If the option is not set, the edit boxes are initialised to 0.0 every time an attribute dialog is opened.
- **Remember tool dialog controls** is used for the same purpose but it applies to the tool dialogs (e.g., **Tools/Copy/by Increment**).

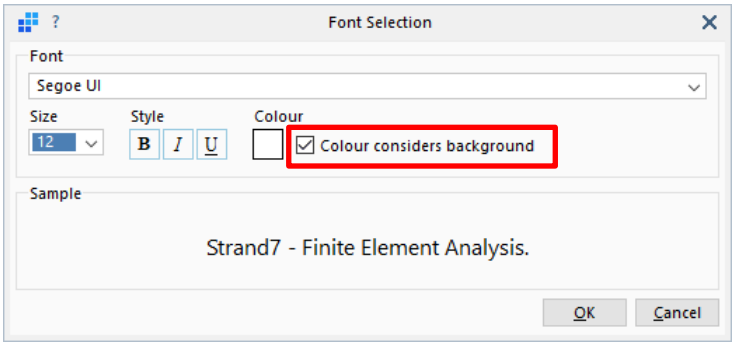
PRINTING

A Line weight option is available to independently control the line for the title block.



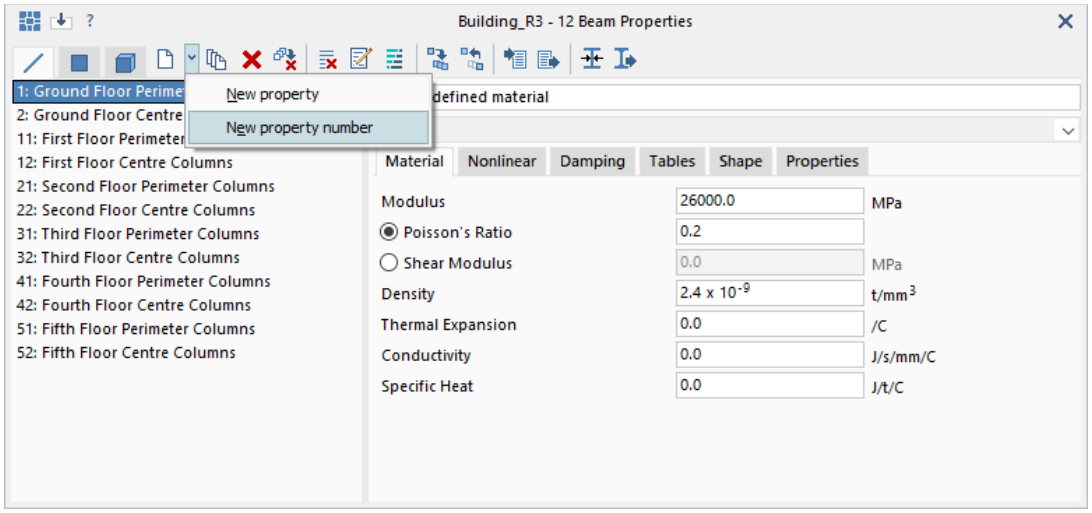
FONTS

The dialog includes the new option Colour considers background. If the option is set, the actual colour of the displayed text may vary to avoid clashes with the background colour. For example, white text on a white background is automatically changed into black text when the option is set. If the option is not set, text is displayed in the specified colour without modification, irrespective of the background colour.

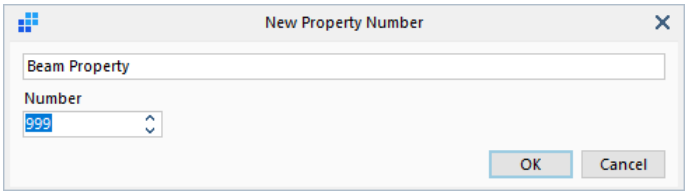


CREATE OPTIONS – NUMBER

A new option is available for specifying the number of a new data set, in place of the automatically generated number. For example, when creating a new beam property, clicking the **New property** icon will create a new property automatically numbered (the next available number after the currently focused property will be used).



To create a beam property of any number, as long as it does not already exist, the **New property number** option can be selected from the dropdown menu. The dialog below will then allow for the definition of the property name and number.



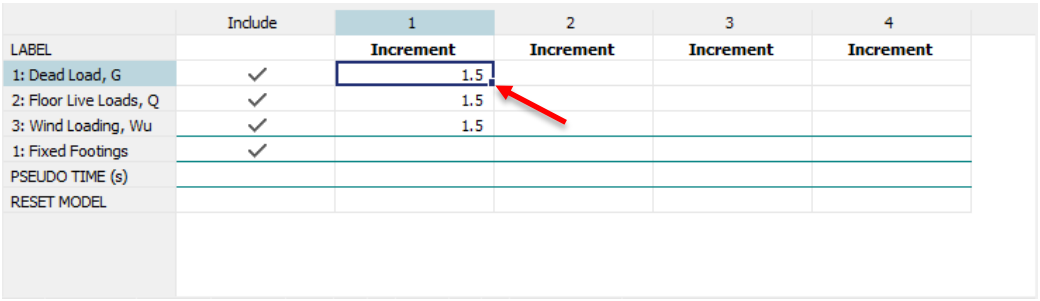
This option is also available in the following locations:

- LAYOUTS/Tables
- LAYOUTS/Plies
- LAYOUTS/Laminates
- LAYOUTS/Creep
- LAYOUTS/Plate RC
- LAYOUTS/Paths
- LAYOUTS/Cavities

GRIDS

Cell-drag Square Double-click

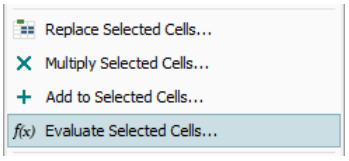
The cell-drag square at the bottom-right corner of a cell can now be double-clicked to copy that cell value all the way down the grid. This feature is available in all of the editable grids, except for **LAYOUTS/Tables**.



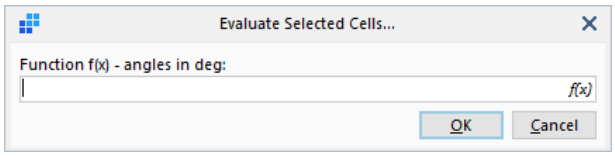
For grids that have underlines to separate different sections, such as the nonlinear static load table grid shown above, data is copied from the double-clicked cell to the next underline. In the grid shown above, double-clicking the cell-drag square for the Dead Load case will copy the data up to and including the Wind Loading case; it will not extend the data into the Fixed Footing freedom case cell.

Equation Editor

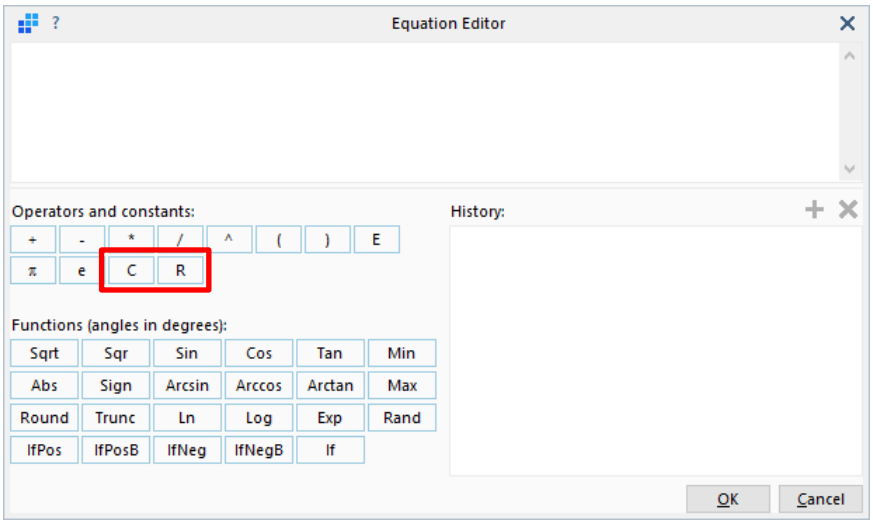
The equation editor is opened in a grid via the ***Evaluate Selected Cells...*** item of the right-click popup menu.



This displays the **Evaluate Selected Cells...** dialog, which allows for an equation to be written into the edit box, or to open the equation editor by clicking the ***f(x)*** icon.



The equation editor available in grids uses a different set of operators and constants compared with the one available in the model window. For the grid-based equation editor there are two new operators: **C** and **R**; these represent the grid column and row, respectively. This facilitates the generation of column/row specific data in the grid. It can be useful, for instance, in the nonlinear load steps grid, amongst others places.



VISUAL TAB

Keyboard Shortcuts

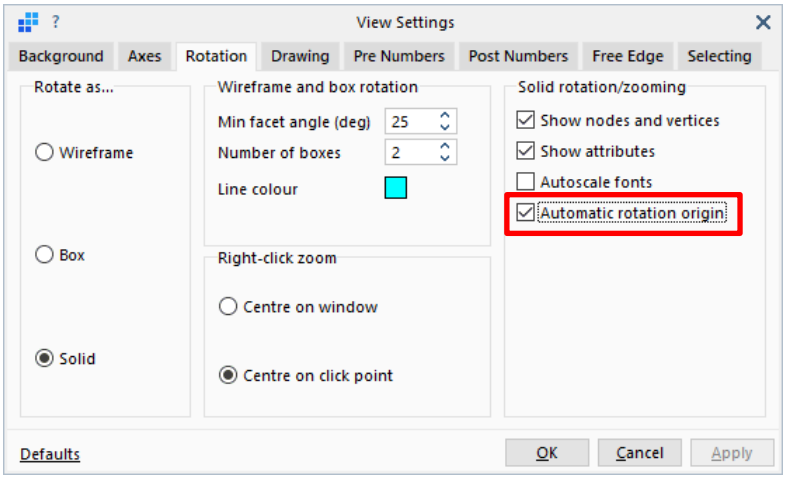
There are new keyboard shortcuts, including a full set of shortcuts for **VISUAL/Attributes**. These can be found in the Online Help.

Variables

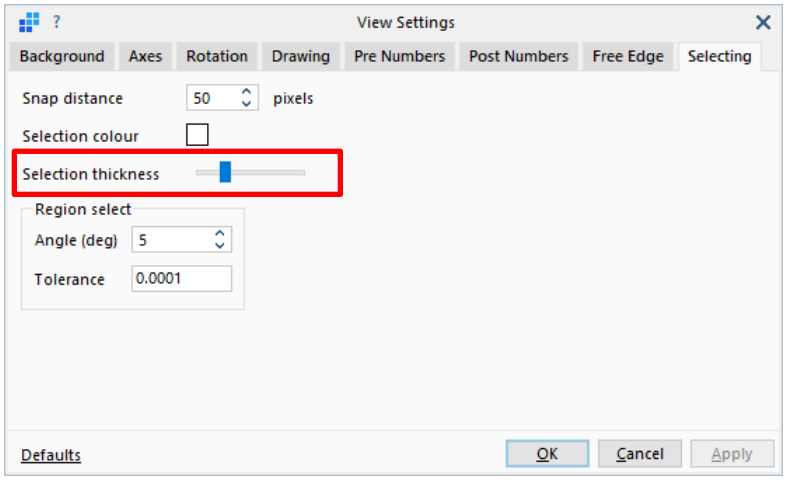
Two new variables **MT** (membrane thickness) and **BT** (bending thickness) are available for assigning plate and face attributes via equations.

View Settings

The **Rotation** tab has a new **Automatic rotation origin** option. With this option, every time you click on the model, the rotation origin is updated to that point (i.e., no need to double-click to set the rotation origin).



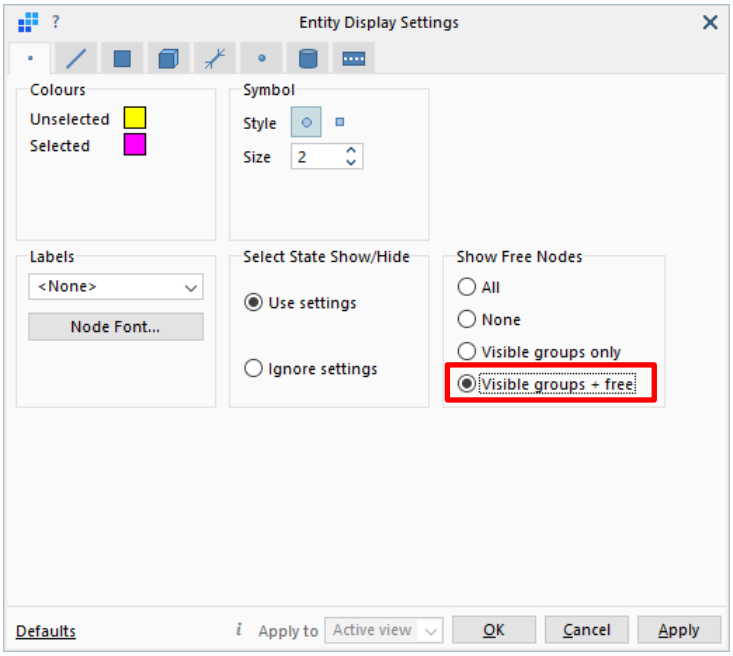
The **Selecting** tab has a new **Selection thickness** option that can be used to change the thickness of the white/grey selection outline when elements are selected.



Entity Display

Node Tab

The new **Visible groups + free** option allows for the display of nodes on active groups together with free nodes (i.e., nodes not connected to any element).



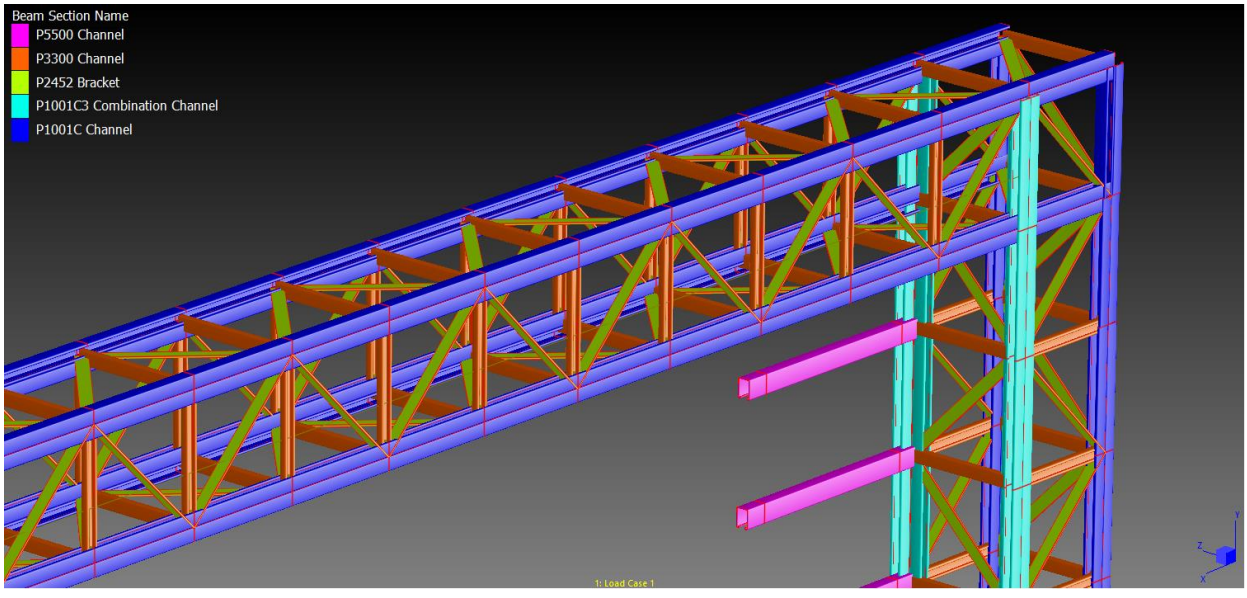
Element pre-contours

The following new pre-contour options are available:

- Beam: **Property Name, Material Name, Section Name**
- Plate: **Property Name, Material Name**
- Brick: **Property Name, Material Name**

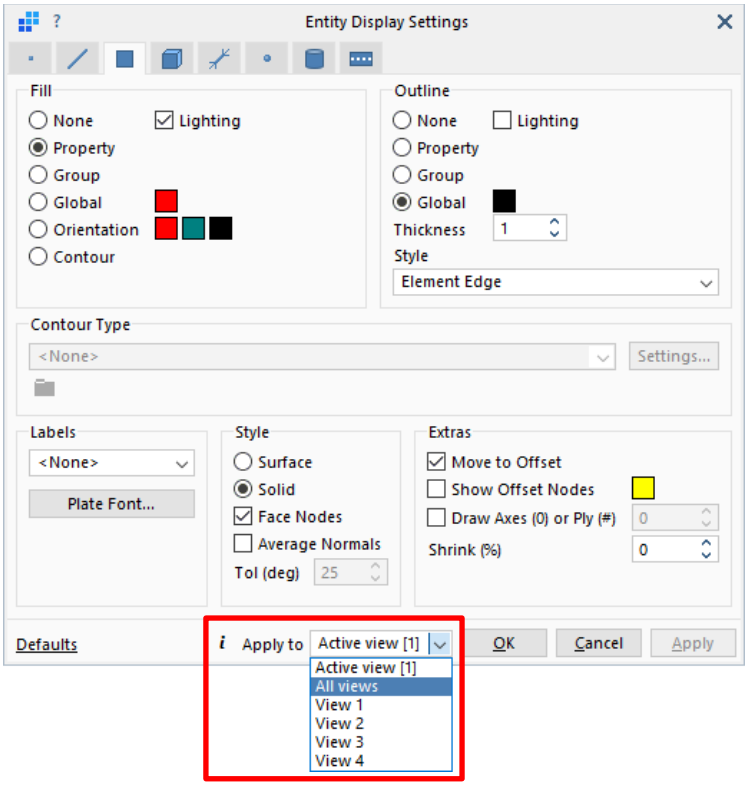
These contours produce a legend with unique labels (i.e., duplicates are removed). For example, if there are 10 properties, and all of them are of material "*Steel*", the Material Name legend will show a single colour and a single label (*Steel*) not 10 colours (all with the label *Steel*).

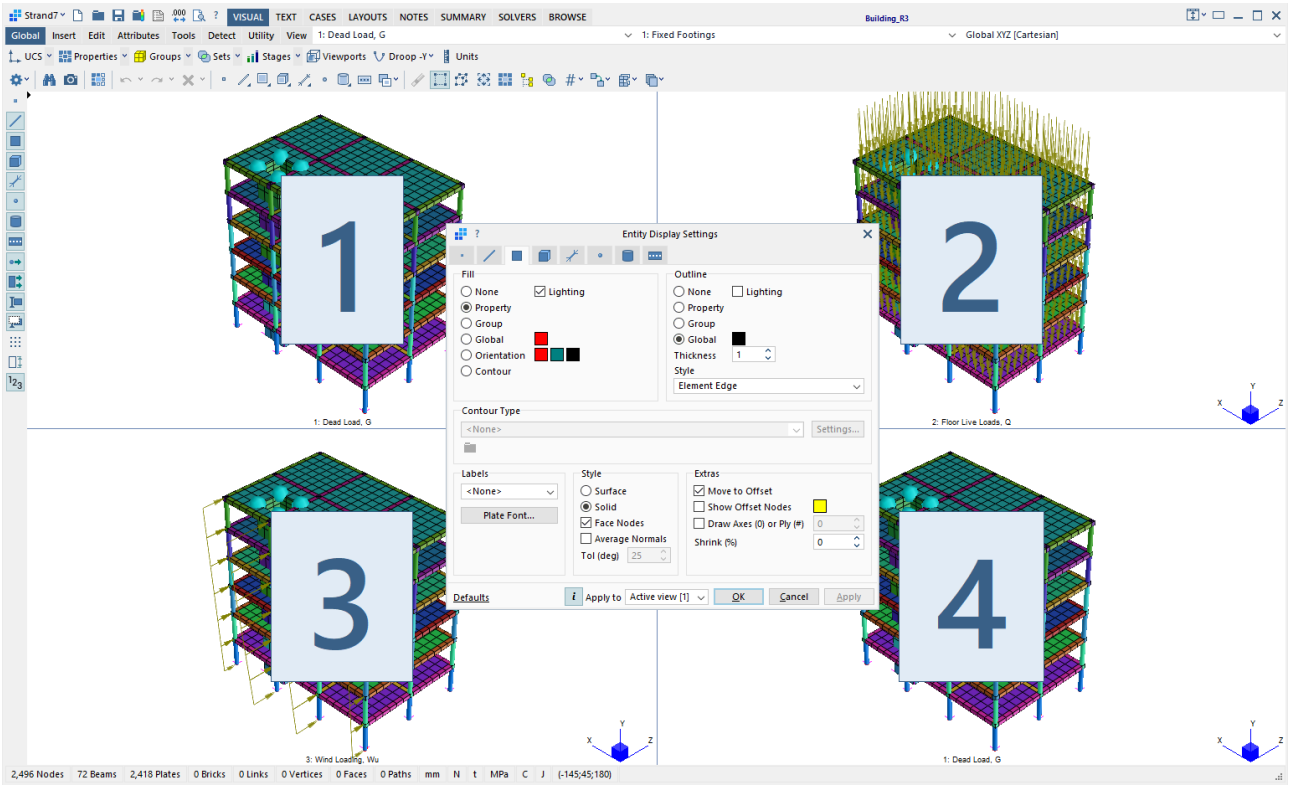
The following image shows a beam section name contour – there are more beam properties in the model than listed in the legend (i.e., there are duplicates), but sections with the same name use the same colour.



View-specific Settings

New options enable the setting of different parameters to different views in a multi-view display without the need to close the dialog and explicitly focus the view each time. The **Identify views** icon ("*i*") is used to identify the view number.





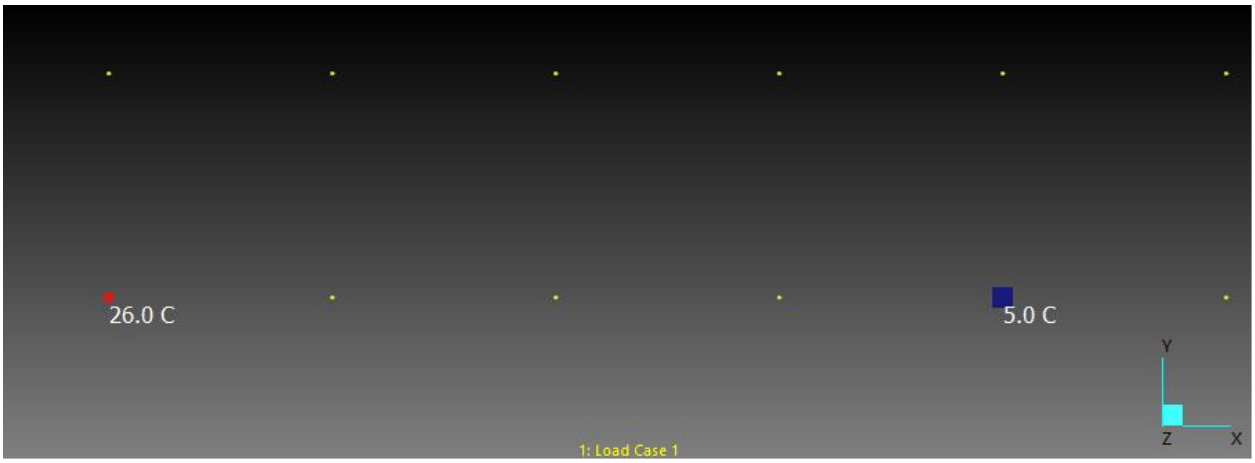
Attribute Display

Node Heat Source

This attribute now has two colours, one for positive values and another for negative values (similarly to node temperature attributes).

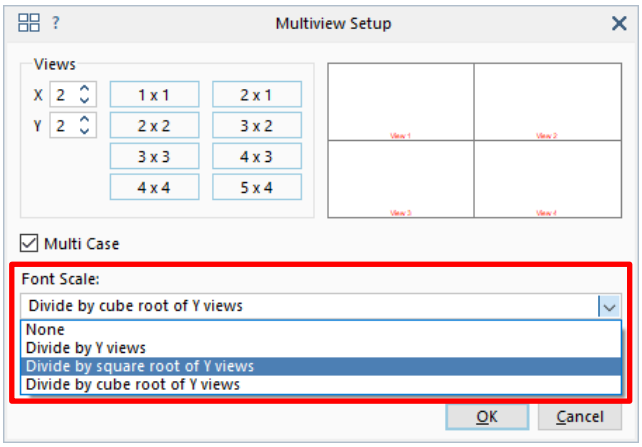
Node Temperature Symbol

The size of the node temperature cube shown in the model window is now proportional to the difference between the nodal temperature and the reference temperature (Tref). That is, it is now proportional to (T-Tref). In the image below, the load case reference temperature is 25°C, therefore the symbol for the 26°C temperature is smaller than that for the 5°C temperature. Temperature attributes above Tref are red; those below Tref are blue (by default).



View/Multiview

The dialog that defines the multiview layout now provides a new **Font Scale** dropdown list that controls the size of fonts when showing multiple views.

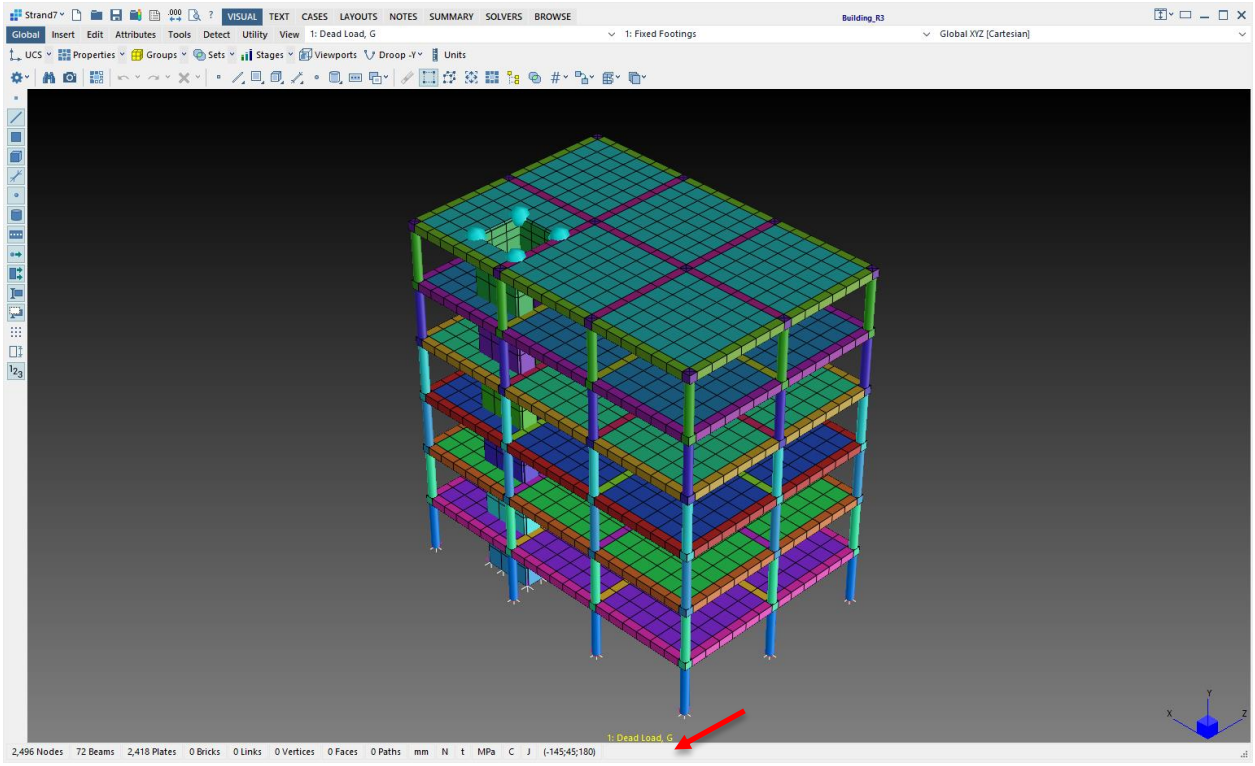


It offers the options:

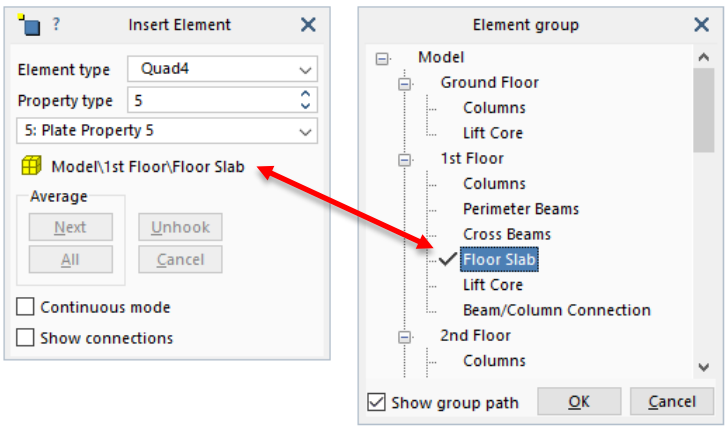
- None
- Divide by Y views
- Divide by square root of Y views
- Divide by cube root of Y views

Default Group

The default group that has until now been shown in the status bar has been removed. The concept of *default group* is now different to that of previous Strand7 releases in that, now, a group is selected as required for the particular function in question.



For example, when inserting a new element via **Insert/Element**, the dialog itself now provides the option to select the group into which the new element should be placed.

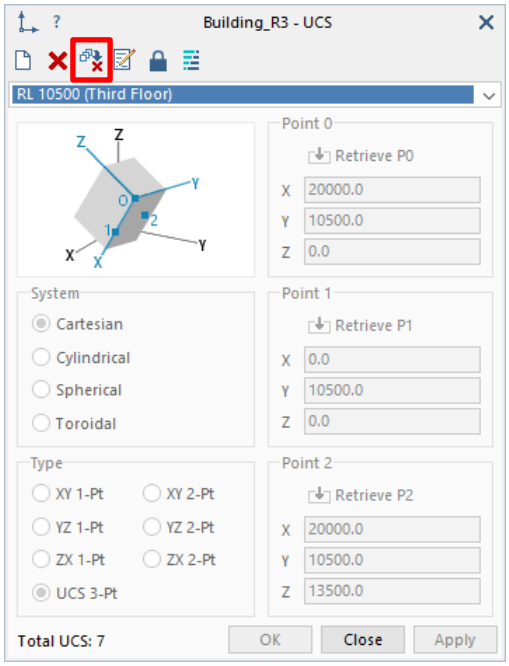


The explicit selection of groups is available on the following dialogs:

- **Insert/Element**
- **Insert/Link**
- **Insert/Path**
- **Insert/Multi-Element/Nodes and Beams by Line**
- **Insert/Multi-Element/Load Patch Plates on Beam Polygons**
- **Tools/Extrude/Targets**
- **Mesh/Solid Mesh from Plates**
- **Geometry/Face from Beam Polygon**
- **Snap Grid/Create**

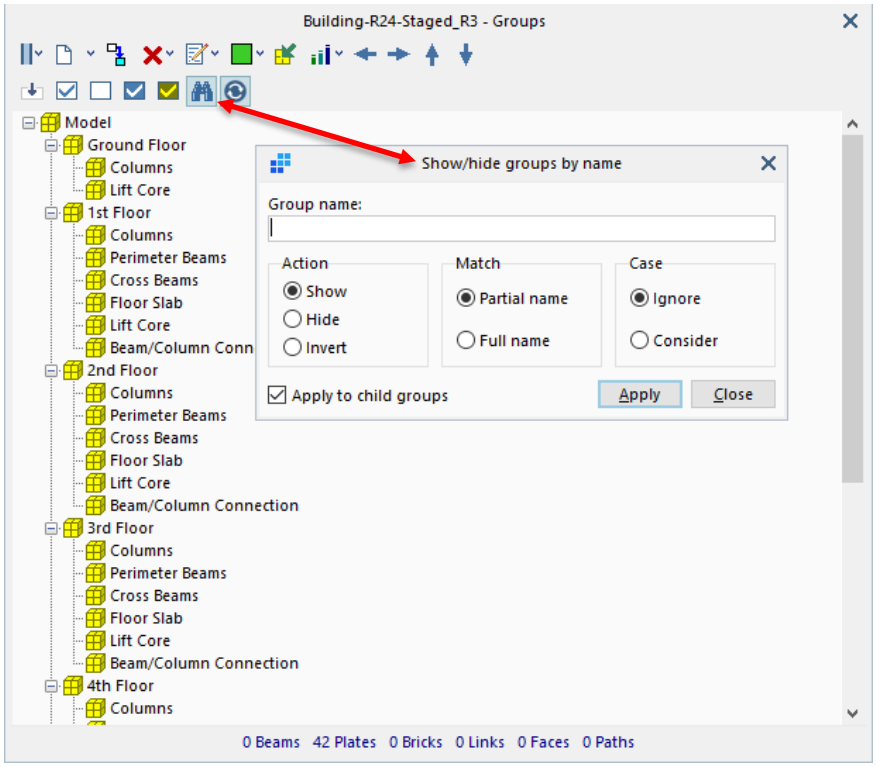
Global/UCS

An option to **Delete unused UCS** is now available on the UCS dialog.



Global/Groups

A new **Show/hide groups by name** option is available. This option allows for the showing, hiding and toggling of groups based on their name. The feature is also available on the **Select by group** dialog via the right-click popup.

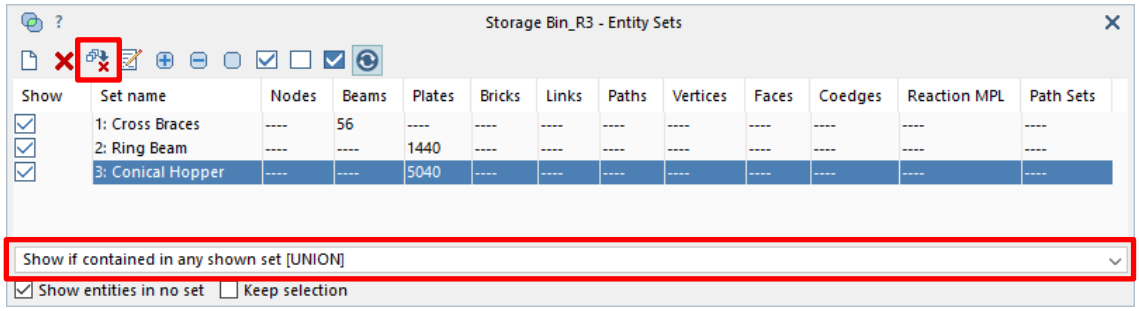


The previous **Set default** group icon is now removed from the groups dialog because, as mentioned above, the default group is now set on individual dialogs as required.

To toggle a range of groups in one operation, shift+click the yellow cube: all groups between the currently selected group and the newly clicked group will be toggled. The status of the clicked group will be applied to the range.

Global/Entity Sets

A **Delete unused entity sets** option is now available.



In addition, the dropdown list on the dialog offers a range of boolean operation options to define the visibility of entities based on their set membership.

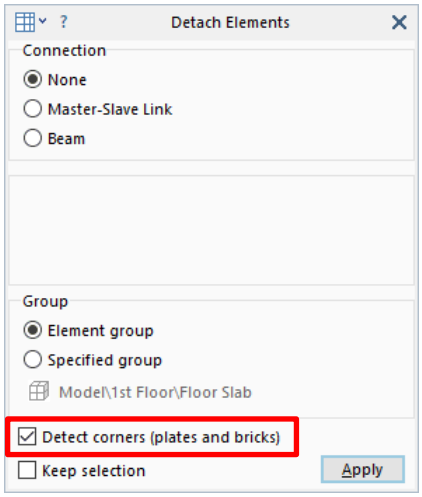
This set boolean feature is available wherever sets are relevant, including:

- TEXT Tab
- SUMMARY/Model Tab
- LISTINGS Tab

Tools

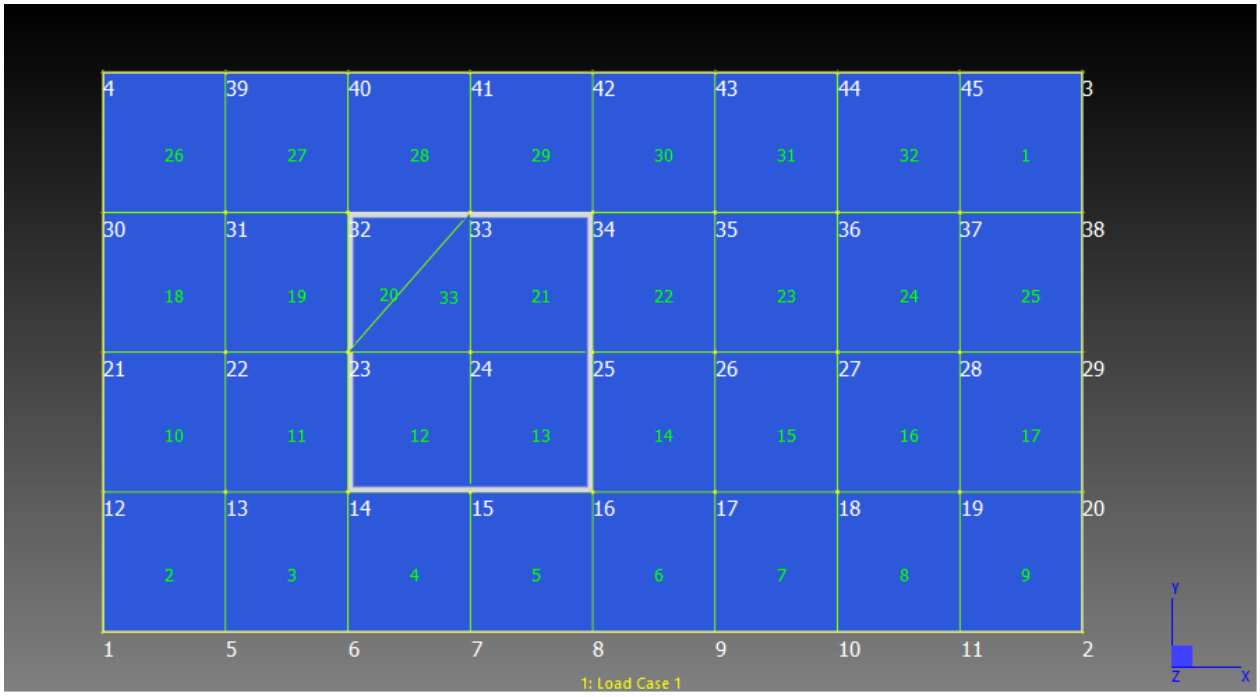
Mesh/Detach Elements

The dialog offers a new option: **Detect corners (plates and bricks)**. This option is useful when a plate edge (or brick face) to be detached has corners of triangular elements on the detach boundary.

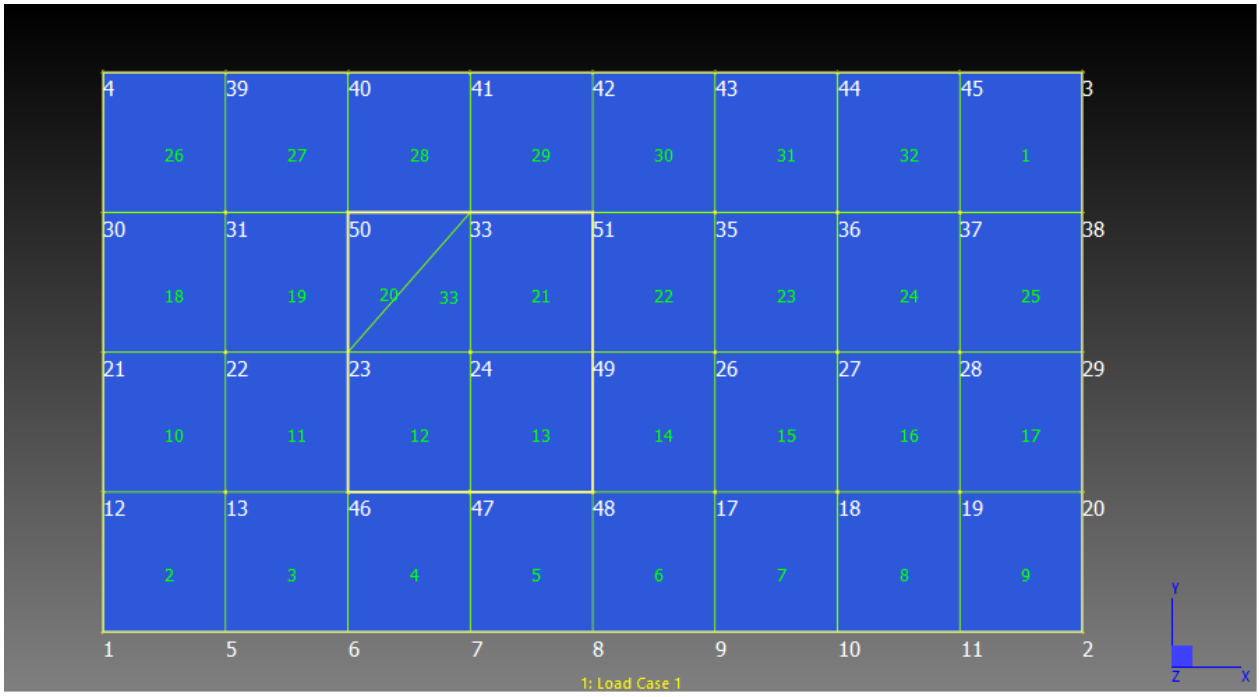


The use of this option is best illustrated by an example.

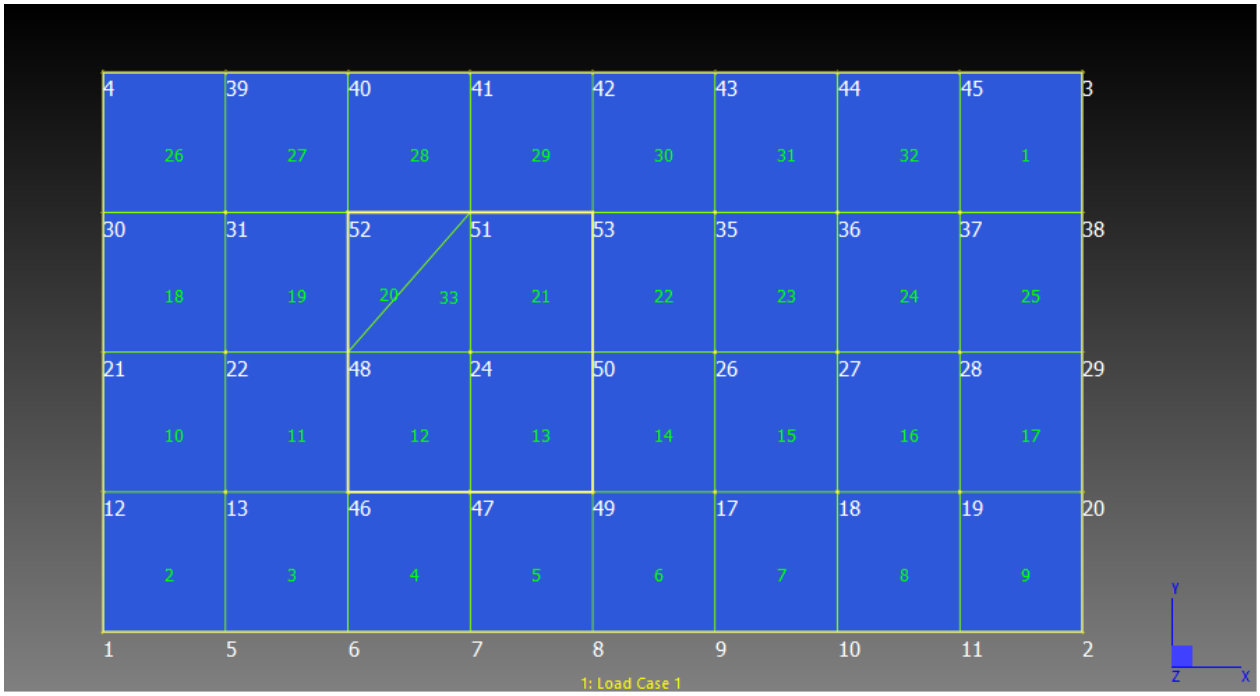
In the following plate mesh, the area highlighted by the selected edges is to be detached. Because the tool detaches edges, the corner of plate element 33 that lies on the detach boundary cannot be specified by edge selection.



If the new option is not set, the detach operation produces the following – all the edges are detached as requested, but plate element 33 is still connected to its original nodes (at the corners).



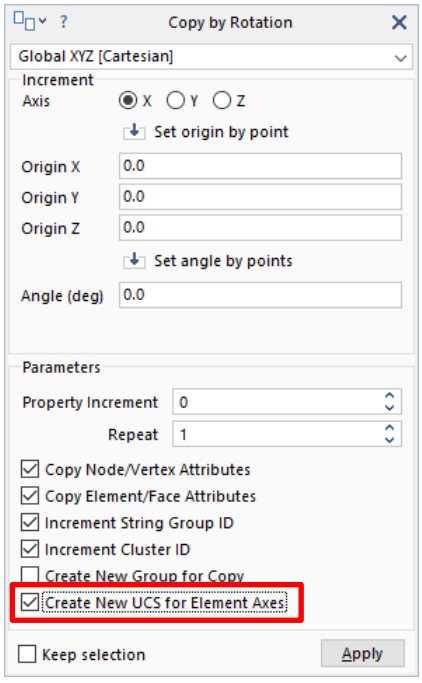
If the new option is set, these corners will be automatically detected and will be considered as detach candidates – as shown below, the triangle is now detached from the original nodes and is now part of the detached cluster.



Copy Tools

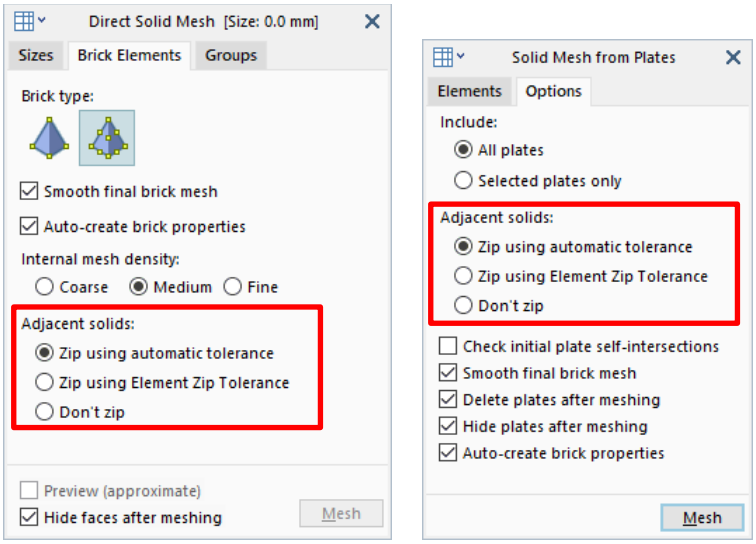
These tools offer the option to **Create New UCS for Element Axes**. The option is applicable to bricks and connection beam elements that have a UCS assigned as the local axes. In that case, the copies of the elements may be assigned new UCSs to ensure that the local axes of the copies follow the axes of the original according to the applied copy increment.

When the copy increments are defined in the same UCS as that of the currently assigned axes, no new UCS is assigned as it is not required. Similarly, when the copy increments are applied in any Cartesian coordinates and the original UCS is also a Cartesian one, no new UCS is assigned because the Cartesian copy will maintain parallel axes. But if the copy increments are applied in a cylindrical, spherical or toroidal UCS that does not match the UCS of the currently assigned axes, a new UCS will be created and assigned.



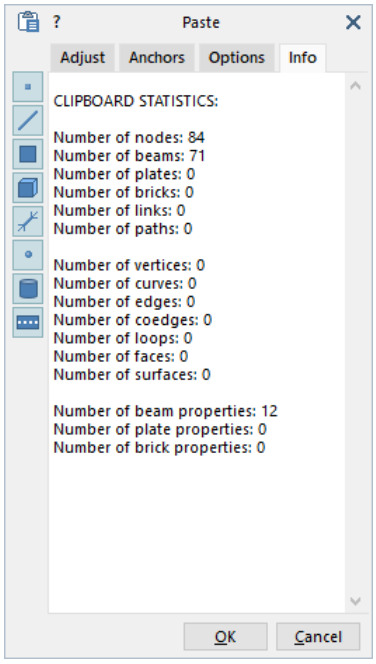
Mesh/Solid Automesh

The adjacent solids zipping option shown on the dialogs below provides control for the zipping of adjacent solids when tet-meshing multiple parts at once. The automatic option calculates a zip tolerance based on the element edge lengths on the surface of the solid. This option will be suitable in most cases. For more control, the other options can be used.



Copy/Paste

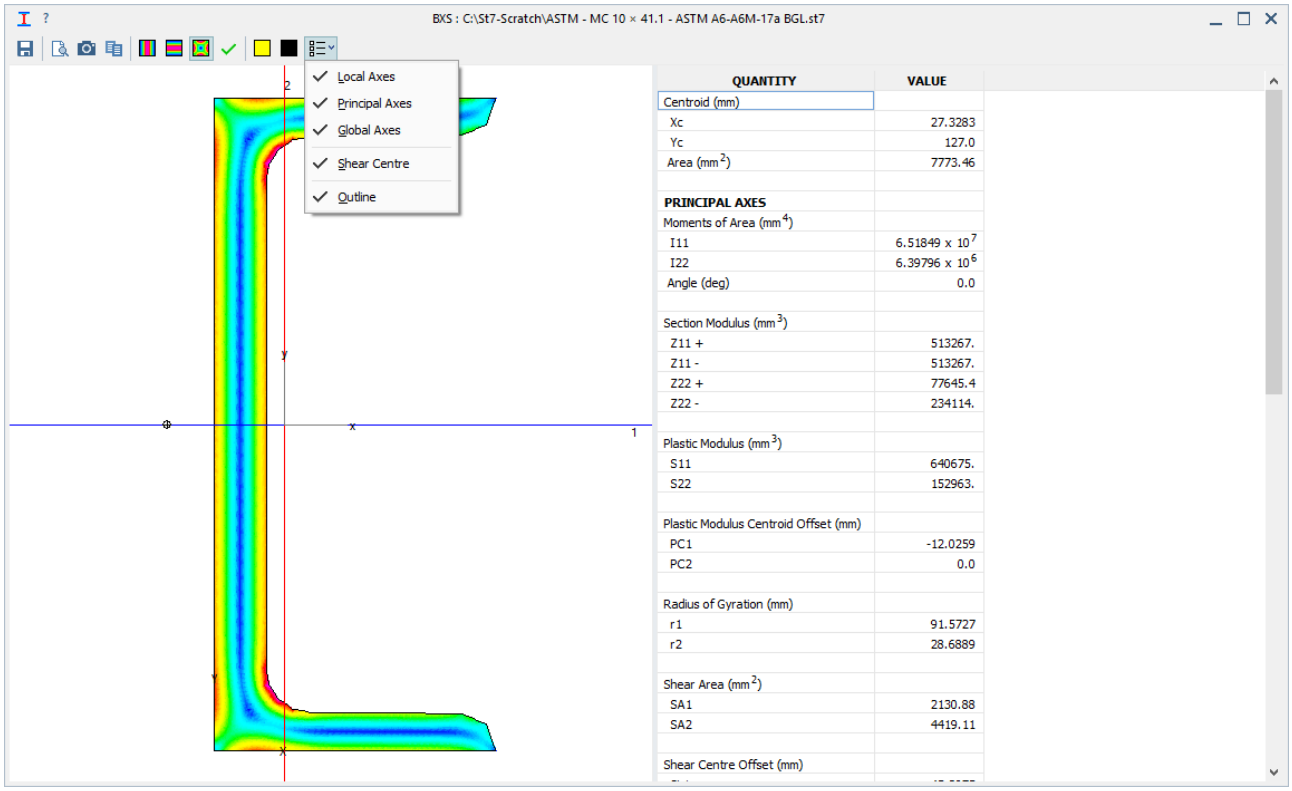
The paste dialog now offers an **Info** tab that provides statistics about what will be pasted, or what is currently contained in the Strand7 clipboard.



Utility/Make BXS

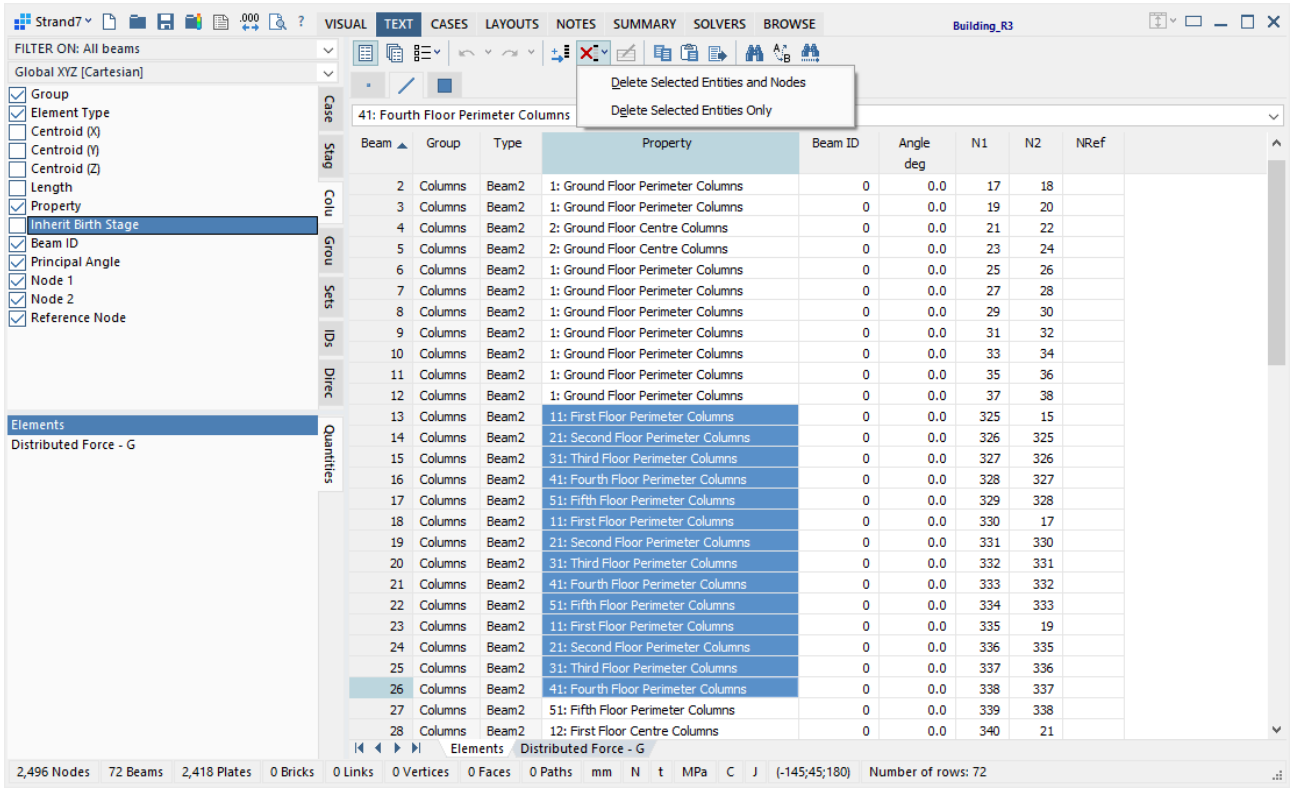
The BXS window has a slightly rearranged icon toolbar, in addition to the option to show/hide various components in the graphic, including:

- Local Axes
- Principal Axes
- Global Axes
- Shear Centre
- Outline



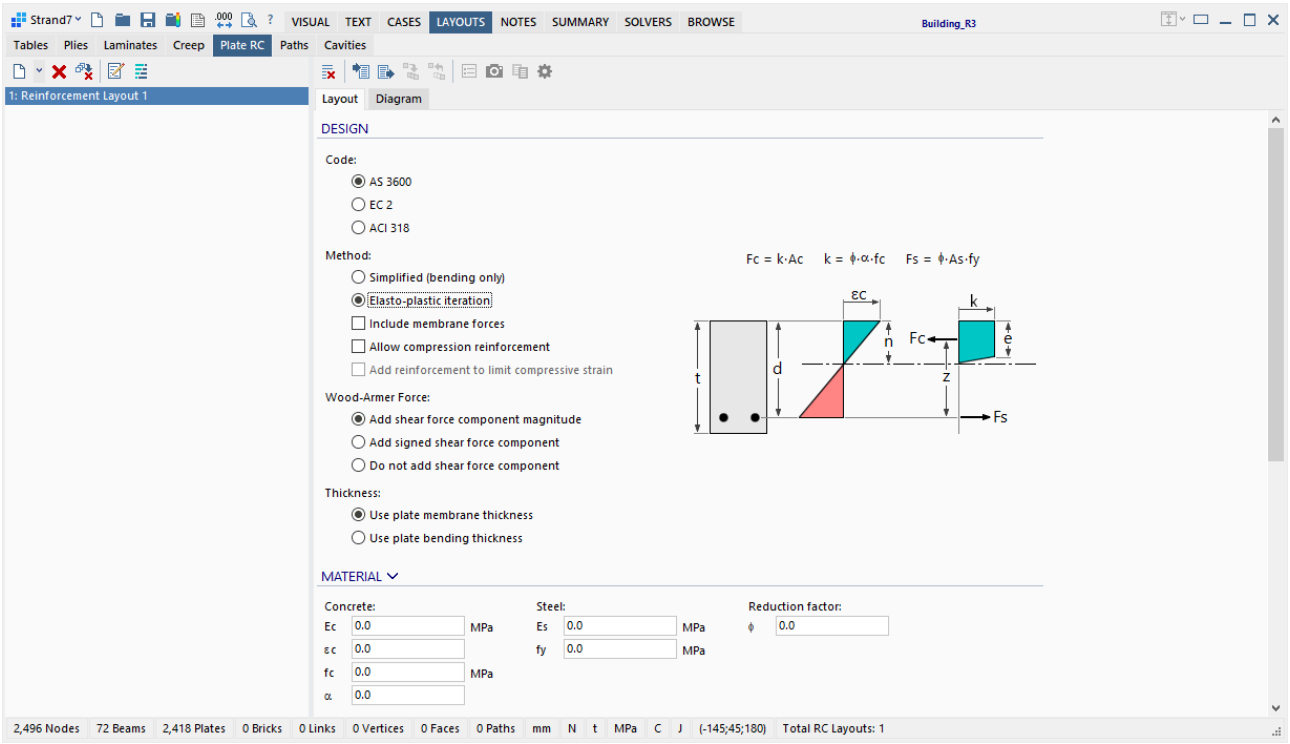
TEXT TAB

The Beam, Plate, Brick, Link and Path tabs have the option to delete entities directly from the grid. For the elements, the function may delete the elements together with their nodes (if they are not used by other elements) or delete just the elements.



LAYOUTS/PLATE RC

There is a new option in **Layouts/Plate RC** relating to how the Wood-Armer force is to be calculated; specifically, how to treat the in-plane shear force component.



The following options are offered:

- **Add shear force component magnitude** (the shear force will be added as a positive value).
- **Add signed shear force component** (the shear force will be added with the same sign as the membrane force in the direction of the reinforcement).
- **Do not add shear force component** (the shear force is not added).

Definition of terms (Wood-Armer notation):

$$\text{Shear} = \left|\frac{N_{yy}\cot\alpha - N_{xy}}{\sin\alpha}\right|$$

$$\text{Force}_{xx} = N_{xx} - 2N_{xy}\cot\alpha + N_{yy}\cot^2\alpha$$

$$\text{Force}_{\alpha} = \frac{N_{yy}}{\sin^2\alpha}$$

- **Add shear force component magnitude**

$$N_x^* = \text{Force}_{xx} + \text{Shear}$$

$$N_\alpha^* = \text{Force}_{\alpha} + \text{Shear}$$

- **Add signed shear force component**

$$N_x^* = \text{Force}_{xx} + \text{SIGN}(\text{Force}_{xx}) * \text{Shear}$$

$$N_\alpha^* = \text{Force}_{\alpha} + \text{SIGN}(\text{Force}_{\alpha}) * \text{Shear}$$

- **Do not add shear force component**

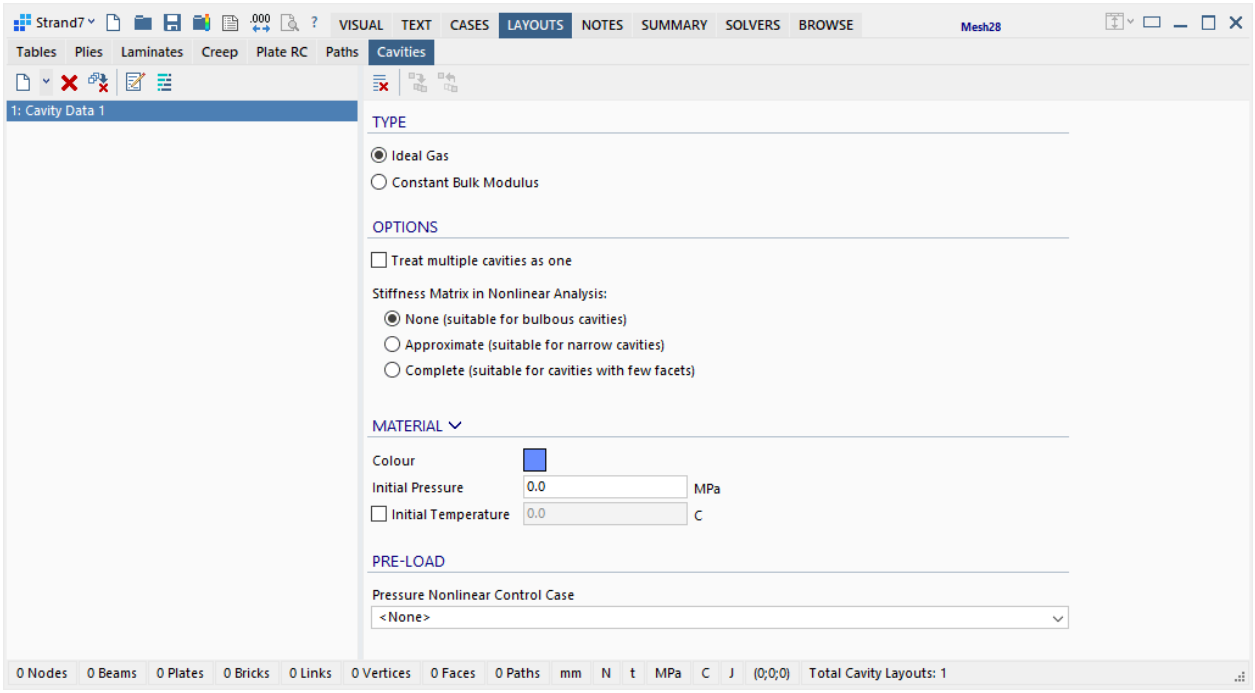
$$N_x^* = \text{Force}_{xx}$$

$$N_\alpha^* = \text{Force}_{\alpha}$$

where N_x^* and N_α^* are the Wood-Armer forces in the directions of the reinforcement.

LAYOUTS/CAVITIES

A new option, **Stiffness Matrix in Nonlinear Analysis**, provides additional capabilities to aid with the convergence of nonlinear cavity problems.

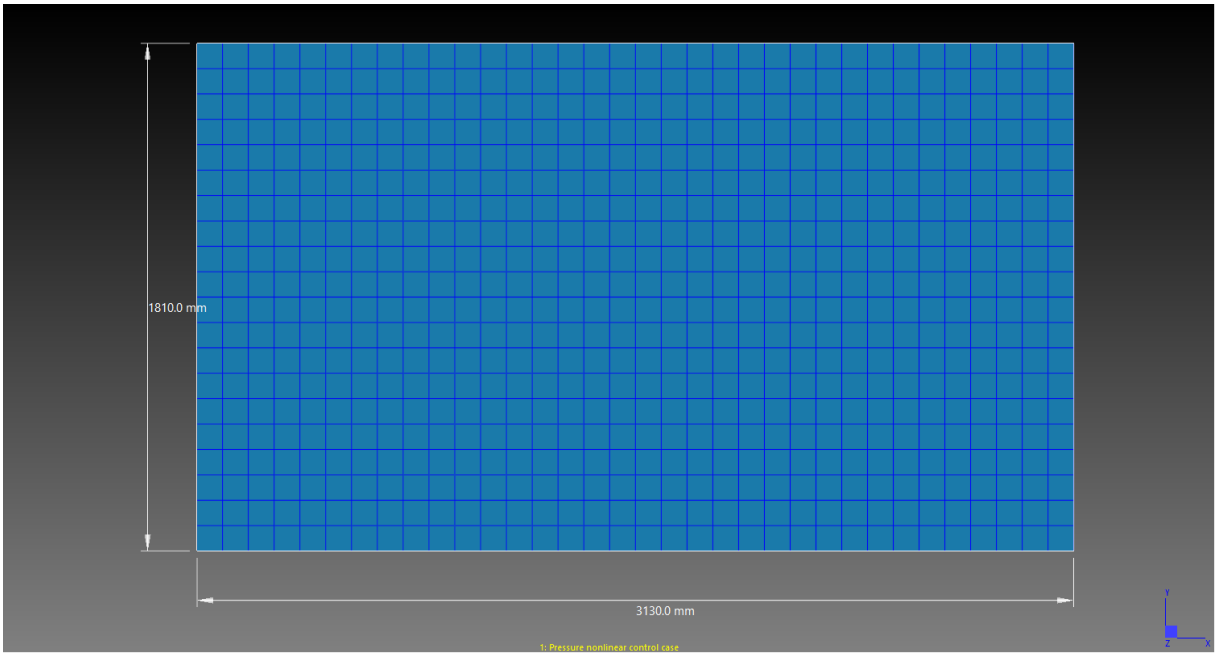


The option selects the numerical approach to be used for solving the nonlinear problem. Its purpose is only to help improve the convergence. For a converged analysis, all three options will produce the same results to within some numerical tolerance. The meaning of the three options is as follows:

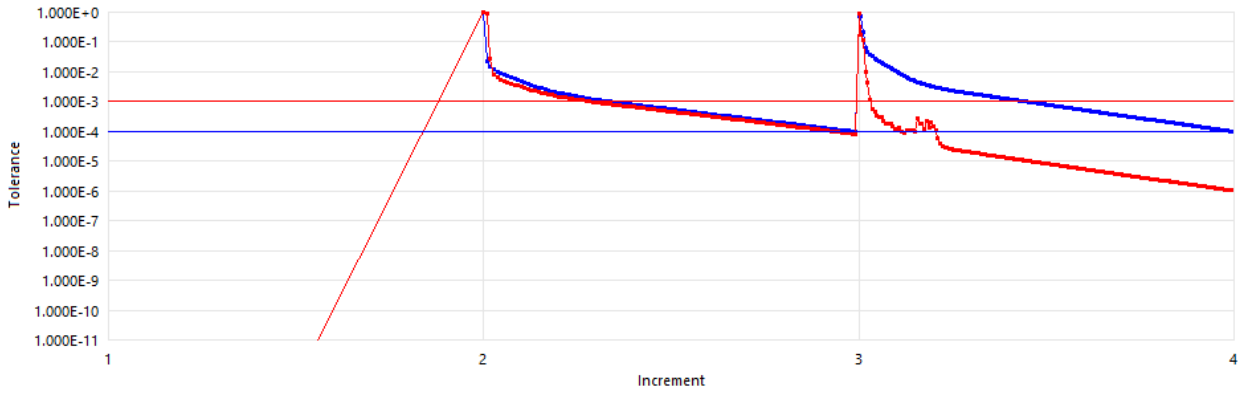
- **None**
  The cavity stiffness matrix is not added to the global stiffness matrix. The change in cavity pressure due to changing volume during the iterations is accounted for by monitoring the volume and applying the corresponding pressure to the relevant nodes. For large bulbous cavities, this option is usually the most efficient. It will typically require more iterations than the other options, but the per-iteration time is shorter. This is the option used in R3.1.1.
- **Approximate**
  The cavity stiffness matrix is approximated via the stiffness of a cluster of tetrahedral elements automatically generated at run time to fill the cavity. This stiffness matrix will not be the true tangent stiffness for the cavity, but its bandwidth will be significantly smaller than that of the **Complete** option. The purpose of the approximate stiffness matrix is to avoid numerical oscillations during the solution process that can occur when the **None** option is used. The **Approximate** method is usually the best one for the analysis of narrow cavities (e.g., IGUs) whose internal pressure may be sensitive to small displacement changes that can cause numerical oscillation and/or convergence difficulties.
- **Complete**
  The full (or "exact") tangent stiffness matrix for the cavity is added to the global stiffness matrix. This cavity stiffness matrix is assembled via the classical numerical integration procedure used for the assembly of displacement based finite elements. This method will require fewer iterations than the other methods. However, for large cavities (i.e., those that connect a large number of nodes) the method will produce a large bandwidth in the global stiffness matrix, which can significantly increase the per-iteration time. The **Complete** method is usually best for very small cavities (i.e., those comprising a small number of nodes).

Comparison of Stiffness Matrix Options

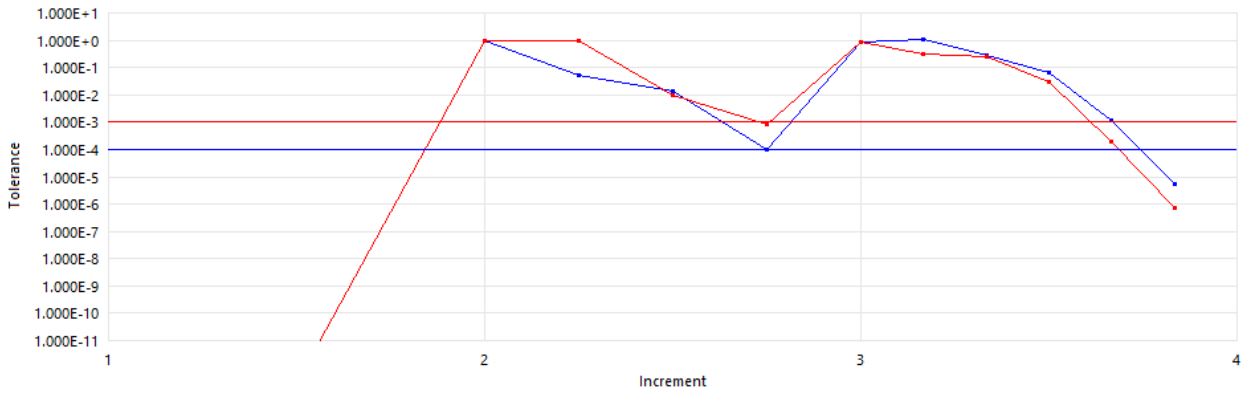
The convergence and results of a simple IGU model are compared for the three options.



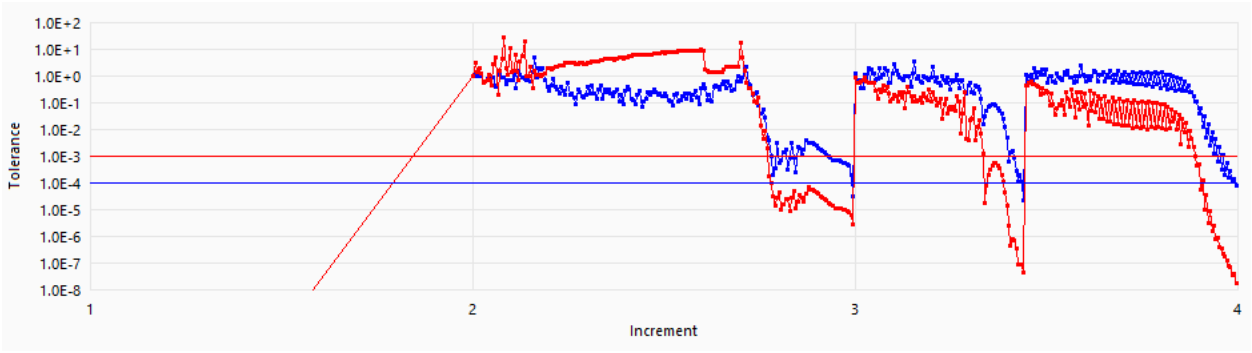
With the **Approximate Stiffness** option, the convergence history shown below is produced: a total of 278 iterations for a run time of 3 minutes and 30 seconds (that is, 1.32 iterations per second).



With the **Complete Stiffness** option, the convergence history shown below is produced (only 12 iterations). This is clearly more effective than the previous option. However, it is not as efficient since the total run time now becomes 9 minutes, representing a significant overall increase (that is, only 0.022 iterations per second).



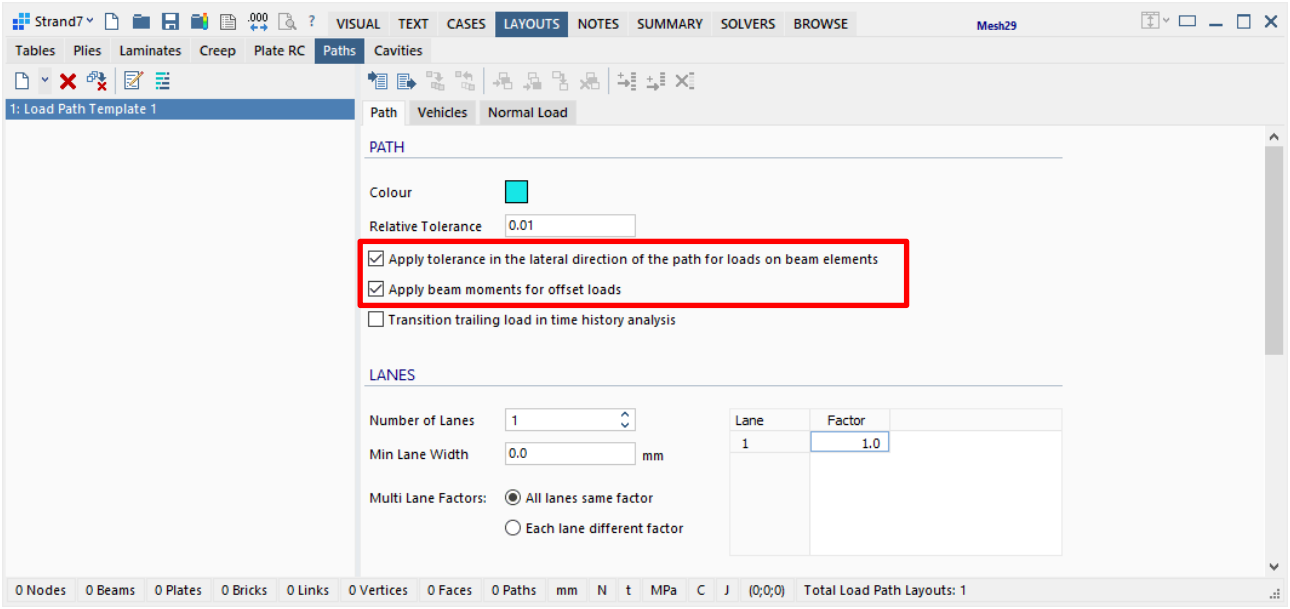
With the **No Stiffness** option the convergence history below is produced. A total of 396 iterations are required, including a sub-step for the third load step, giving a total run time of 4 minutes and 20 seconds (that is, 1.52 iterations per second). Although the per-iteration time is the fastest with this method, overall it is less efficient than the **Approximate Stiffness** method.



The results for the final load step are all within 0.02% of the average for this model, so all three methods will produce the same results when the analysis converges.

LAYOUTS/PATHS

Two new options are provided for the load paths.

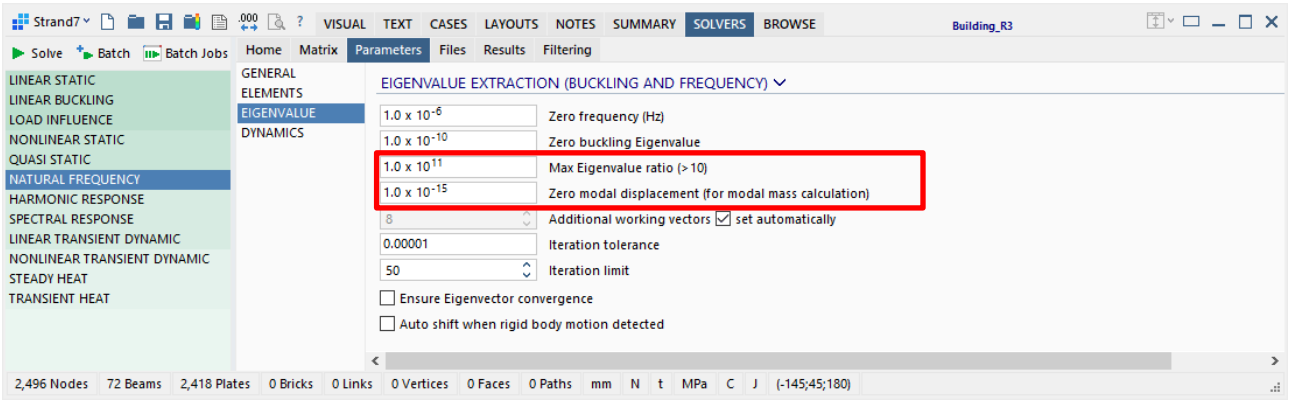


- When **Apply tolerance in the lateral direction of the path for loads on beam elements** is set, a load on a load path will only be applied to a beam element when the patch of the path intersects the beam element and the beam element is laterally within the specified **Relative Tolerance**. If the option is not set, the lateral distance is ignored, and therefore the load is applied to the beam element as long as the beam element intersects with the patch on the path.
- When **Apply beam moments for offset loads** is set, a path force applied to a beam that is offset from the point through which the load passes will also apply a moment to the beam. When the option is not set, the moment is ignored (only the force is applied).

SOLVERS

Parameters

EIGENVALUE

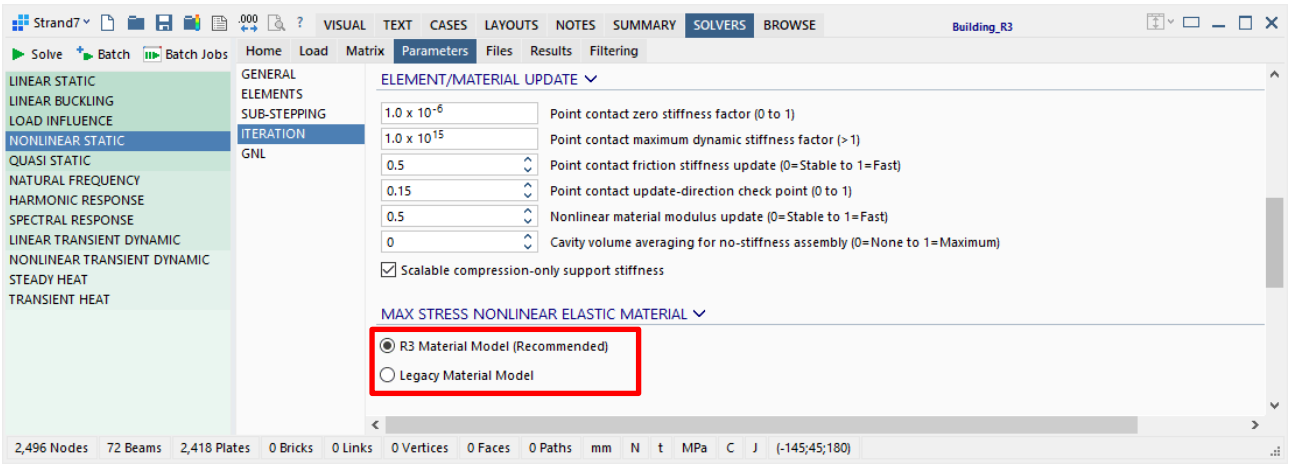


Two new options are provided:

- **Max Eigenvalue ratio** sets a limit between smallest and largest Eigenvalues within a sub-space iteration. An Eigenvalue will not be considered as zero unless the maximum Eigenvalue in the set is this ratio times bigger. This provides some control in situations where too many Eigenvectors get removed from the sub-space iterations because they are deemed to be zero, which could eventually produce far fewer modes than requested. The bigger this ratio the more vectors are retained within the sub-space.
- **Zero modal displacement** is used for the calculation of modal mass. The modal mass needs to normalise the Eigenvector relative to the maximum displacement component. If the maximum displacement component is smaller than this value, the Eigenvector is not normalised and it is assumed to be the zero vector unless a rotational degree of freedom is non-zero, in which case the Eigenvector is normalised using this rotational degree of freedom.

ITERATION/MAX STRESS NONLINEAR ELASTIC MATERIAL

A completely new treatment of the Max Stress material model is included. This new version will produce much better convergence compared with the previous version (which is still available by selecting the **Legacy Material Model** option).



RESULTS

Settings

New options enable the setting of different parameters to different views in a multi-view display without the need to close the dialog and explicitly focus the view each time. The **Identify views** icon ("*i*") is used to identify the view number. This is similar to the functionality offered in the Entity Display dialog, described above.

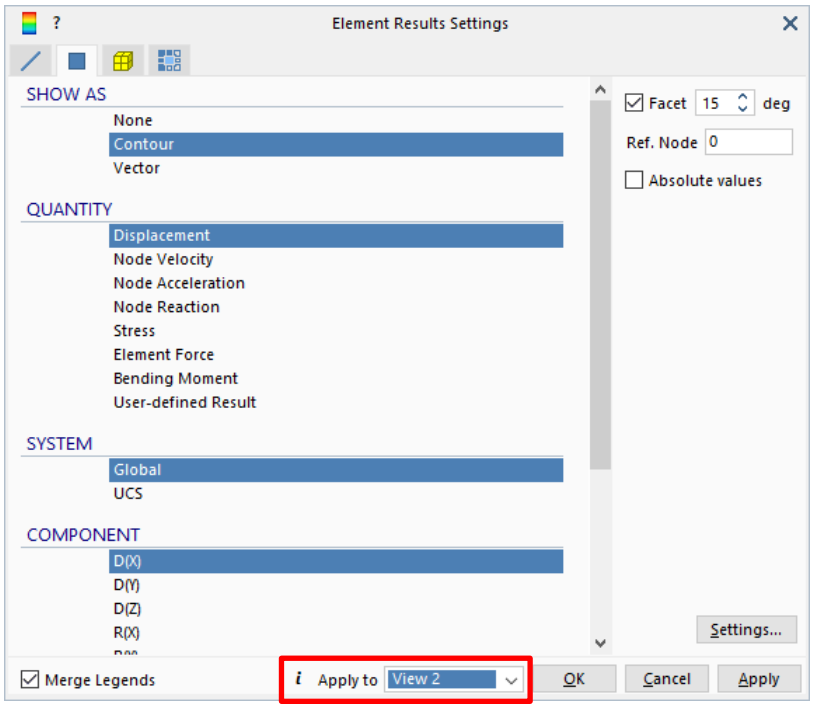
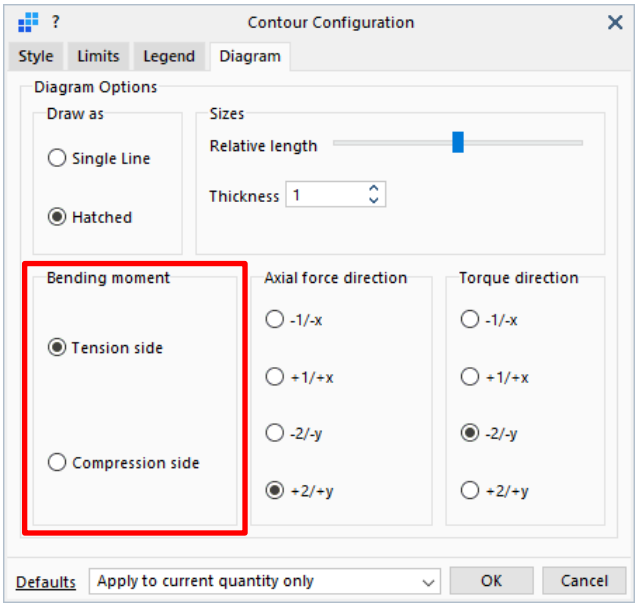


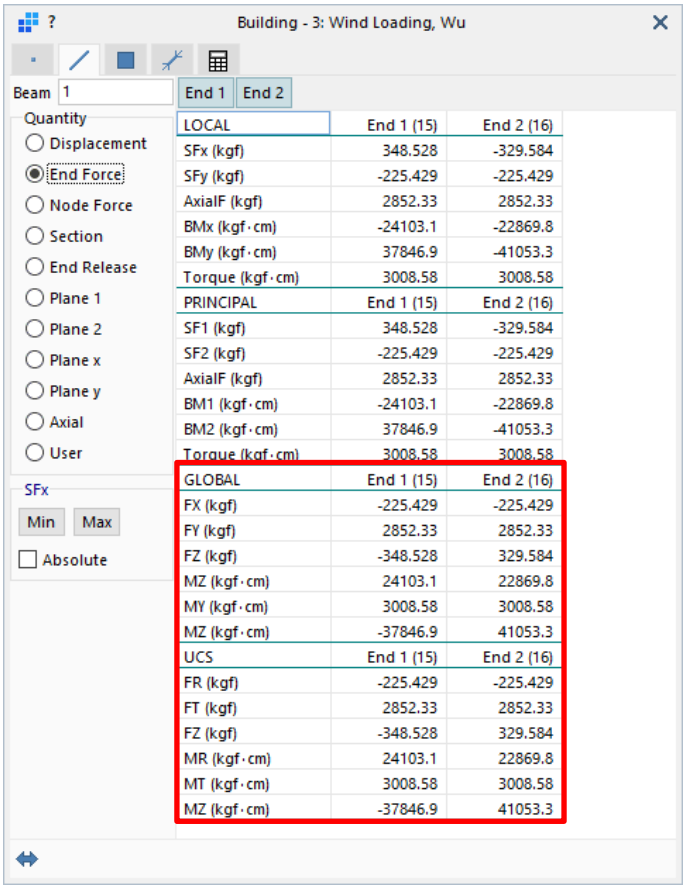
Diagram Options

A new option, **Bending moment side**, is available. This allows for the selection of the side of the beam on which to draw a positive bending moment diagram. The default convention in Strand7 is to draw the bending moment diagram on the tension side of the beam, but this can now be changed.

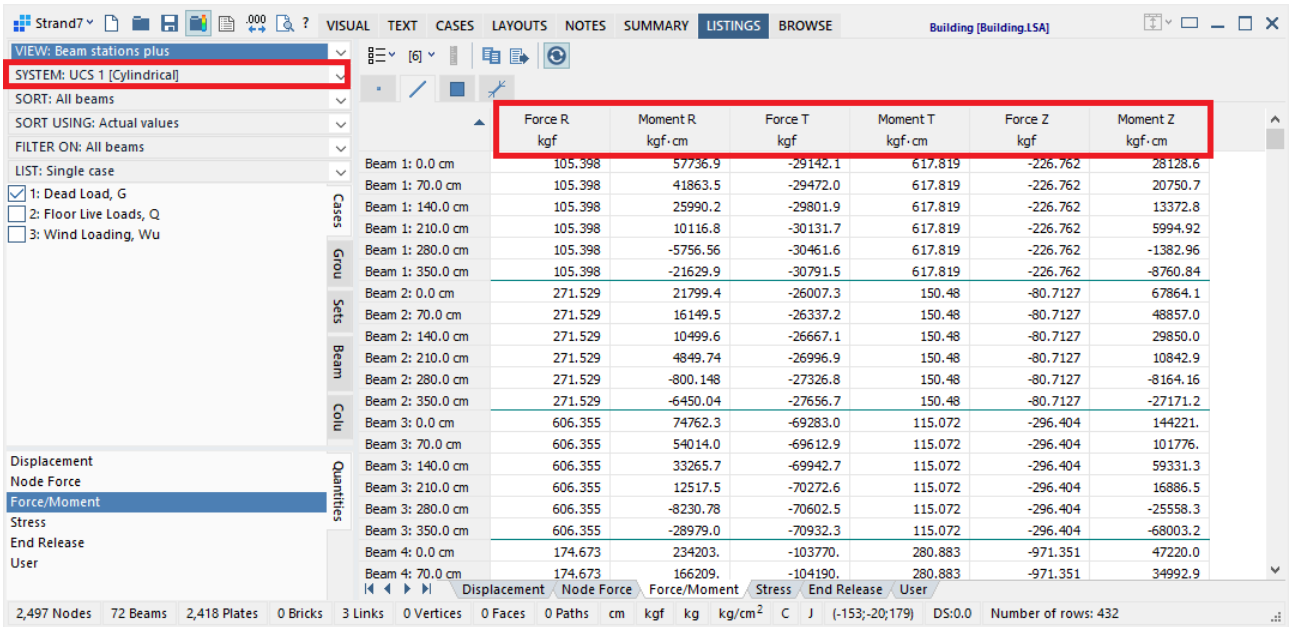


Beam End Forces in Peek and LISTINGS

Beam End Forces in **Peek** are now available in the **GLOBAL** and **UCS** reference systems in addition to the **LOCAL** and **PRINCIPAL** systems. See the Online Help for information about the sign convention.

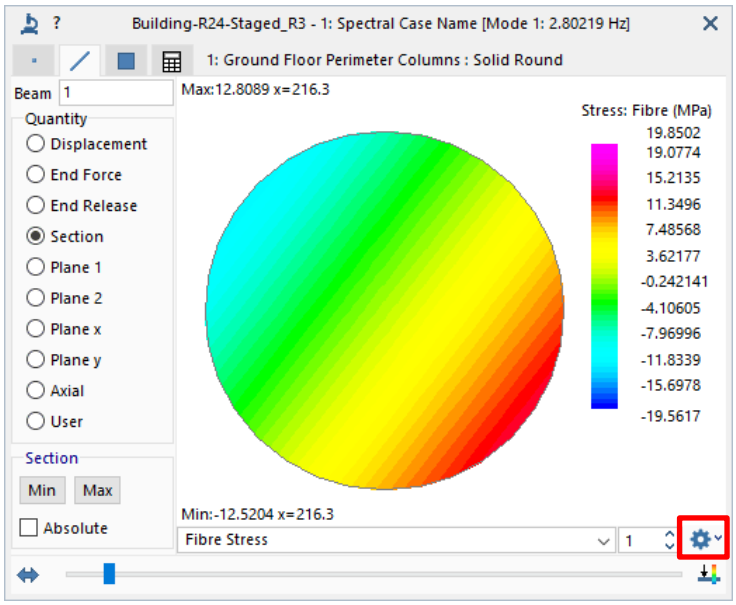


These quantities are also available in **LISTINGS** and in the API, where they may be extracted at any position along the beam, not just at the ends.



Beam Section Contouring in Peek

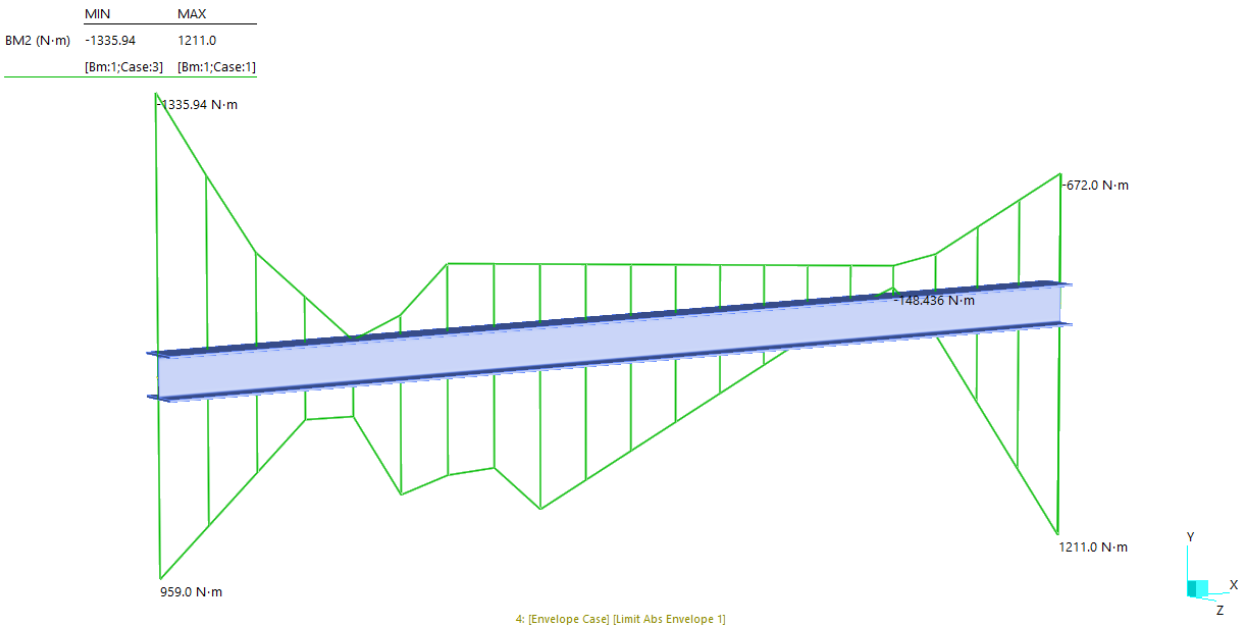
The **Settings** icon on the Peek window allows for customisation of the display, including showing/hiding outlines and setting their colour.



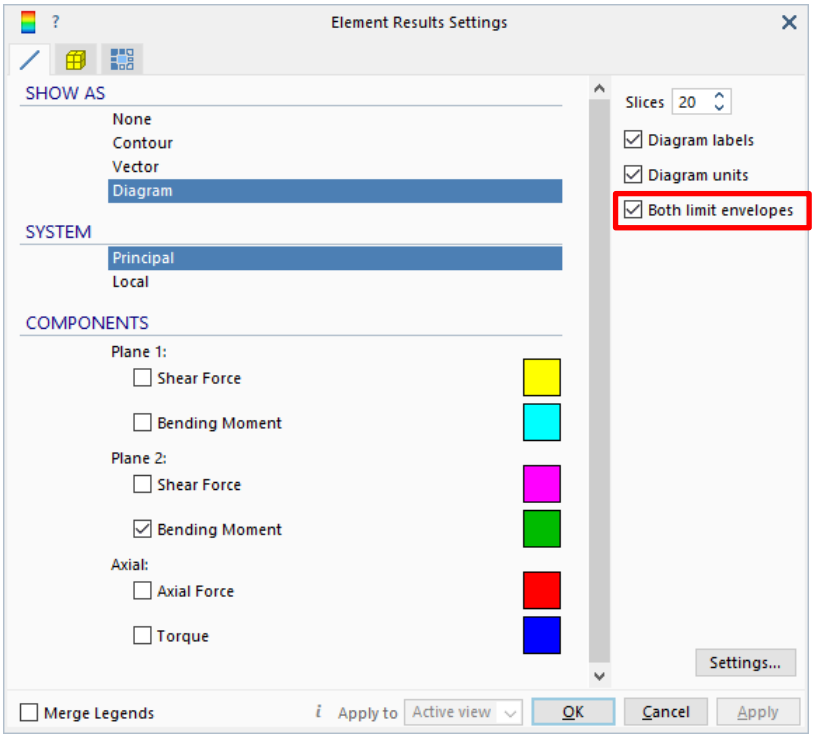
The slider offers a right-click popup to allow for a precise setting of the position, as well as for selecting from a list of pre-defined positions.

Beam Diagrams for Limit Envelopes

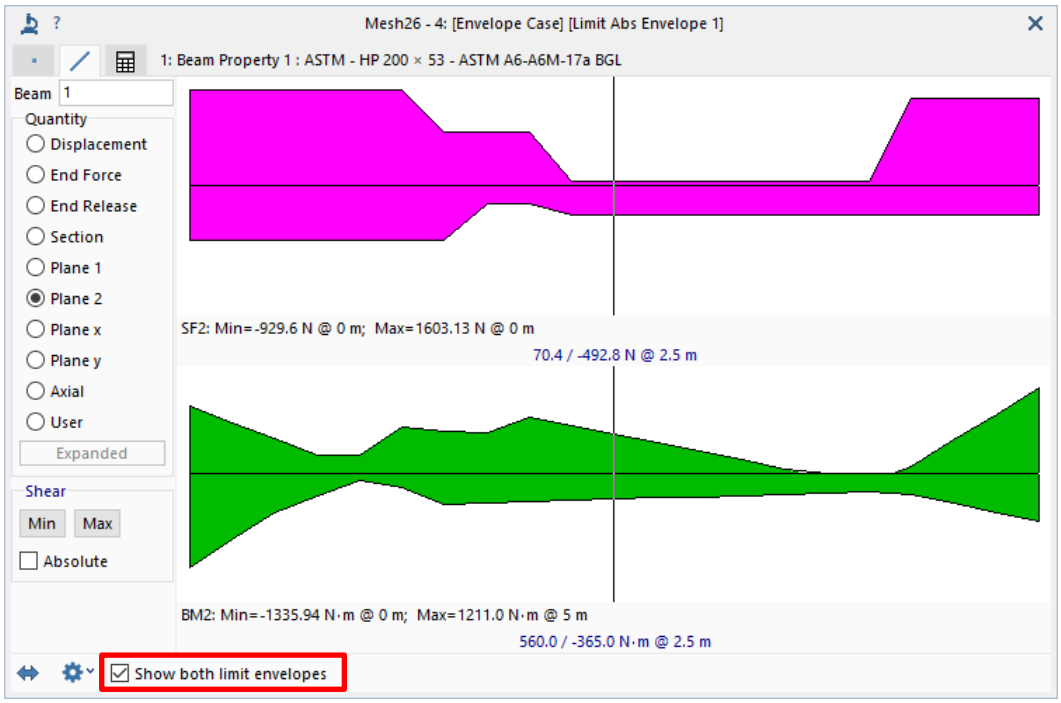
The feature that draws the enveloped beam diagrams for both the min and the max side at the same time is now available for the Limit Envelopes.



The feature is available as an option by setting the **Both limit envelopes** checkbox on the **Element Results Settings** dialog.



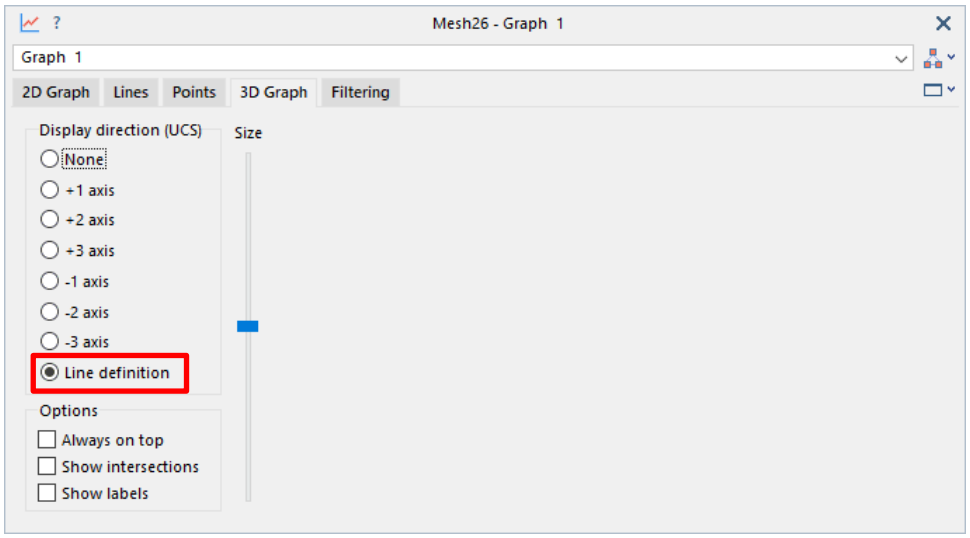
This feature is also available in the **Peek** window by setting the **Show both limit envelopes** checkbox.



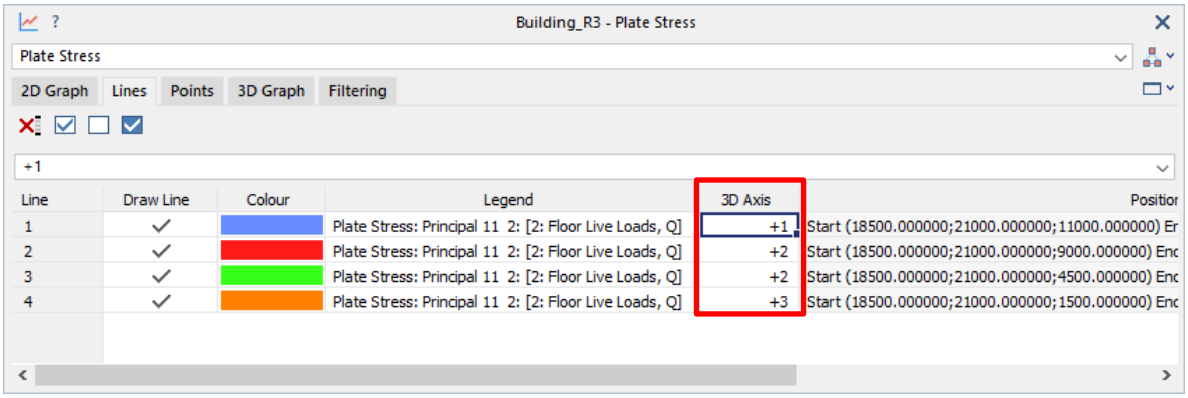
Similarly to the new options to configure outline colours, etc., for the section contour **Peek**, new options exist to configure the display of diagrams in **Peek**.

Graphs

The **3D Graph** tab of the graphs window adds the option **Line definition** to the **Display direction** group.



If this option is selected, the graph in the model window will be drawn based on the direction specified for each individual line segment of the graph (**3D Axis** column in the grid below).



New Applications

St7APIDiagnosticsTool

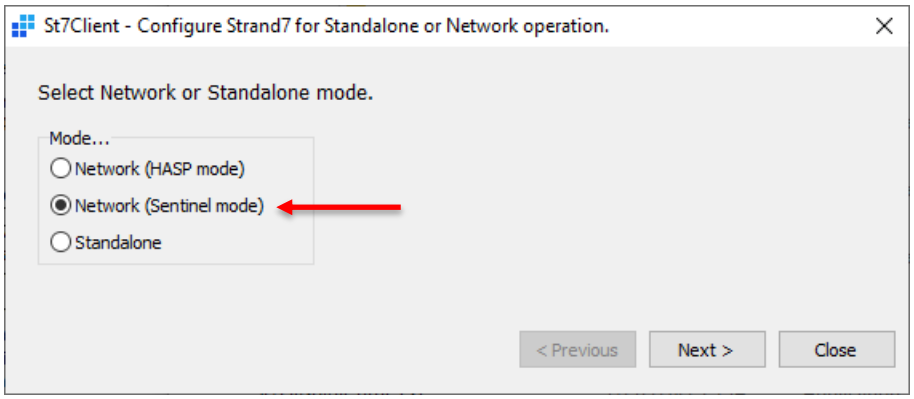
This application is used to diagnose API installation and licensing problems.

St7SetAPIPath

This application is used to set the Windows PATH environment variable so that any Strand7 API program on a particular PC can locate the API DLL (St7API.DLL). This application avoids all the manual steps required to set the Windows PATH environment variable via Windows/Settings.

New Network Licencing Mode

This new mode applies to users who use Strand7 in network mode via a NetHasp USB dongle.



The Hasp USB network dongle can now be run in two modes:

1. HASP mode (previous mode)
2. Sentinel mode (new mode)

Sentinel mode offers greater flexibility in the way the licences are deployed, managed and monitored. This is achieved using the Sentinel Admin Control Center through a browser interface.

To use your Strand7 network licence in Sentinel mode, the network dongle needs to be re-coded. To request this re-coding, please contact us (info@strand7.com).