Strand7

FINITE ELEMENT ANALYSIS

Strand7 R3.1 Feature Summary

JULY 2023

CONTENTS

Introduction

Strand7 Release 3 (R3) is a major new development of the Strand7 Finite Element Analysis System. It comprises the following features:

1. A new graphical user interface (GUI).
2. Hardware accelerated 3D graphics.
3. 64-bit architecture.
4. Increased functionality and performance compared with previous Strand7 releases.

This document summarises these features. Information on many other minor, but useful, features can be obtained via the context sensitive online help within R3. Information that helps users of Strand7 R2.4.6 (R24) to quickly get up to speed with R3 can be found in the **Strand7 R3 Quick Start Guide for R24 Users** (e.g., the section on keystroke shortcuts).

System Requirements

R3 can be executed on the following versions of 64-bit Windows®:

- Windows 7
- Windows 8
- Windows 10
- Windows 11

At least 4 GB of memory (RAM) is required, together with a graphics card that natively supports DirectX® 11.1 or above, with Feature Level (DDI) 11_1 or above. Most graphics cards manufactured in the past five years will have this functionality.

Architecture

64-bit

R3 is a 64-bit application throughout. This includes the GUI, the solver and the API, together with supporting applications such as the log file viewer, the animator and the Strand7 model viewer.

The most important benefit of the 64-bit architecture, from Strand7's point of view, is the accessibility of significantly more RAM compared with a 32-bit architecture. Whereas a 32-bit application is limited to 3 GB of RAM, irrespective of the hardware or operating system, a 64-bit application can access all of the RAM available on the system. For the analysis of large models, access to larger amounts of RAM can significantly reduce the solver run time.

An additional benefit of the 64-bit architecture is that the global stiffness matrix can be larger than 32 GB, which is the limit in previous Strand7 releases. This allows R3 to solve much bigger models.

Multi-Core Support

Multi-core functionality is currently supported in a few areas of R3, including:

- The surface automesher.
- The generation of facets for shaded rendering of geometry.
- Various functions in the pre-processing environment related to tools and rendering.
- The Eigenvalue routines used for natural frequency and linear buckling analysis.
- The back-substitution phase of the matrix reduction process in the solver.
- The processing of displacement vectors for solver runs that have multiple result cases (e.g., linear static analysis with multiple load cases).

Additional parallelised functionality is planned for future R3 updates.

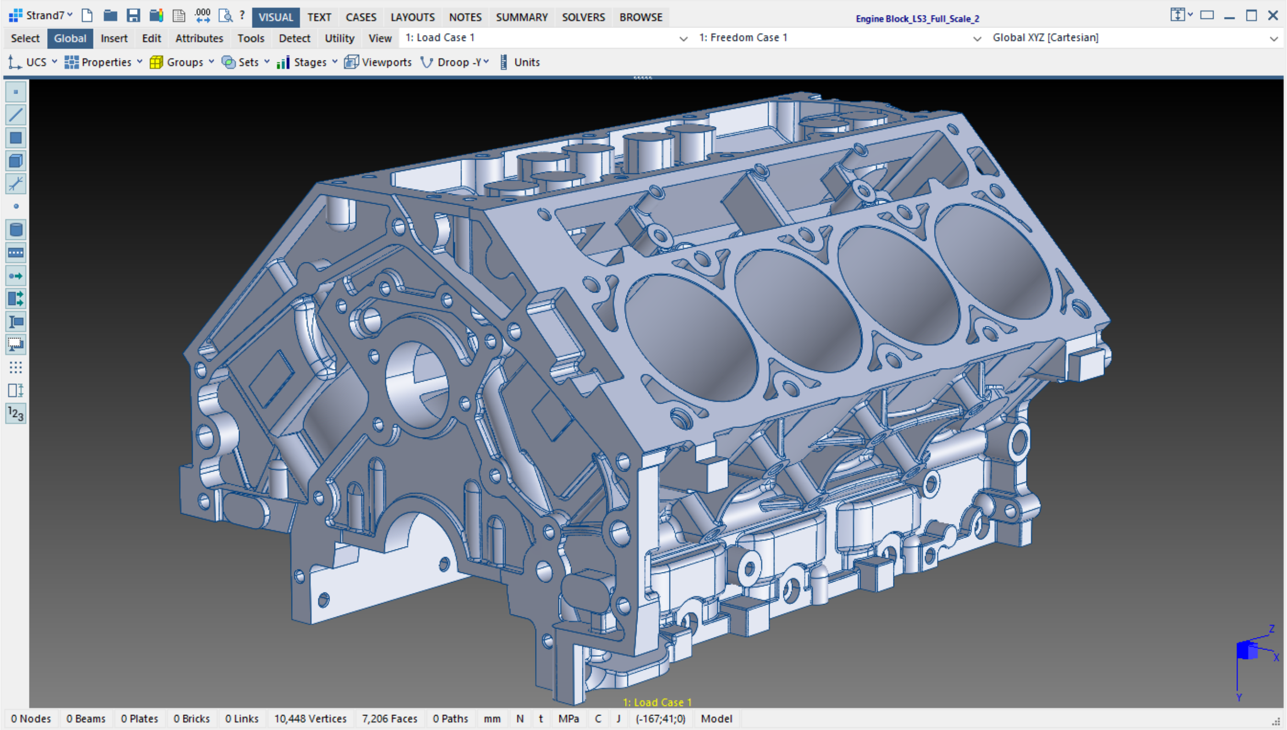
DirectX

R3 uses hardware accelerated 3D graphics through the DirectX 11 API for the rendering of the finite element model.

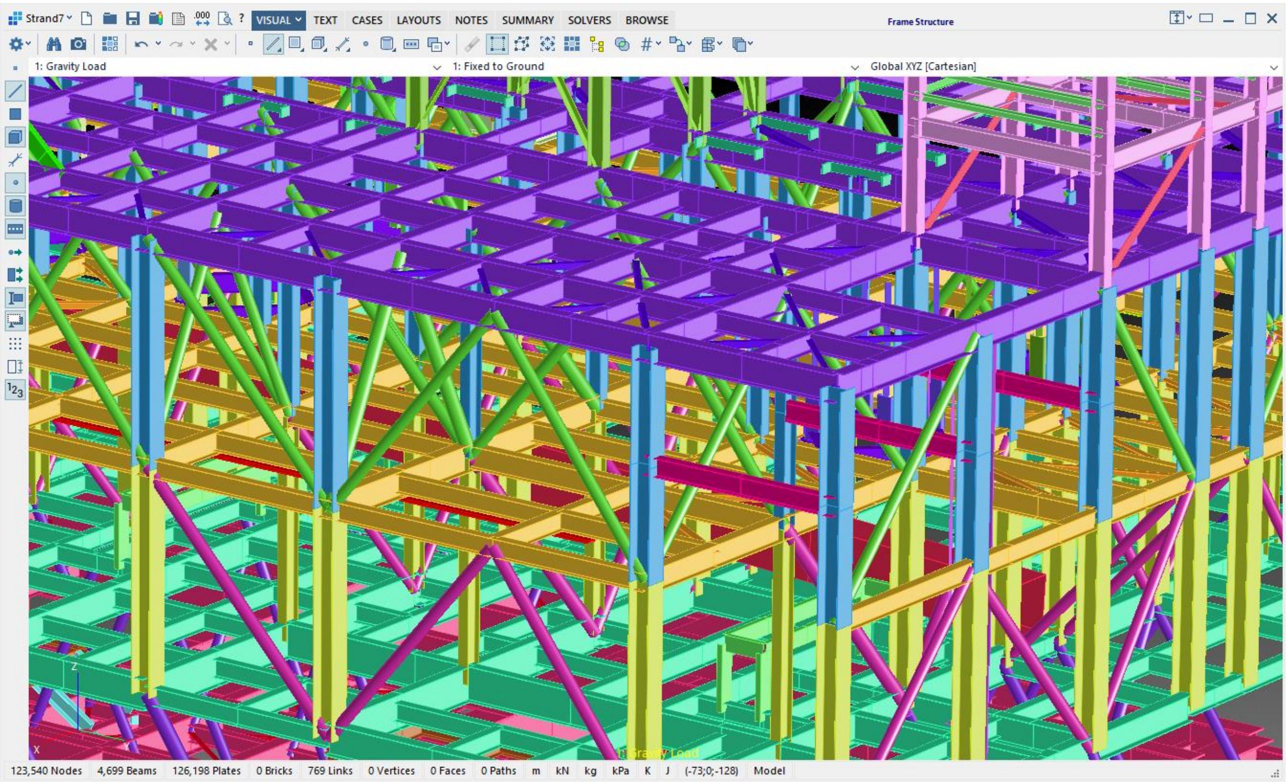
The principal benefit of this feature is that it offers significantly greater graphics performance compared with previous Strand7 releases, which use the CPU to produce the rendered image through the Windows Graphics Device Interface

(GDI). As the rendering in R3 is performed by the hardware (GPU), Strand7's graphics performance on a system can be improved by upgrading the graphics card.

An additional benefit of the hardware accelerated graphics for Strand7 is that CAD geometry can be rendered in shaded solid view, similarly to a CAD system (in addition to the previously available wireframe mode).



It also offers the possibility of including more detail in the representation of structures consisting primarily of beam elements.



FILES

Model ST7 Files

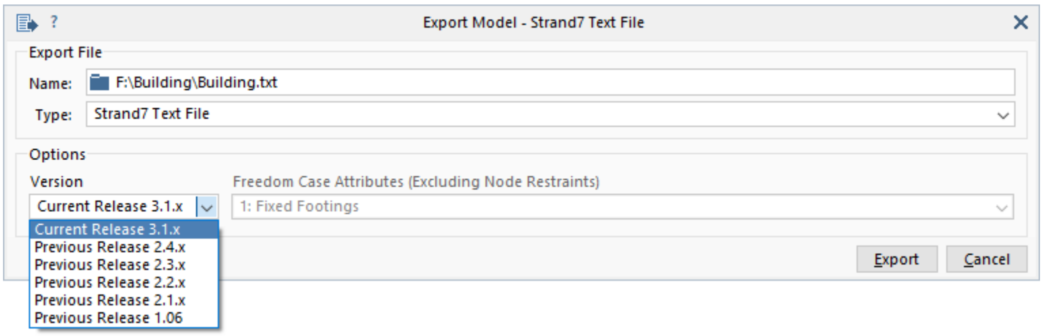
In R3, the Strand7 model file name retains the *ST7* extension, as in previous Strand7 releases.

R3 will open ST7 files from all previous Strand7 releases. Upon opening, older files need to be converted to the R3 format. This process is relatively fast except for models that contain a large number of beam cross sections. Beam cross sections need to be rebuilt for use in R3. More information on beam cross sections in R3 can be found further down in this document.

When saving an ST7 file from a previous Strand7 release as an R3 file, the suffix "*_R3*" is automatically appended to the file name (e.g., after opening the R24 file, *Building.ST7*, the first attempt to save that file in R3 will be suggested as *Building_R3.ST7*). Although the suffix can be removed and the file saved using the previous name, keeping the suffix avoids losing the old version file, allowing for both versions of the file to be kept.

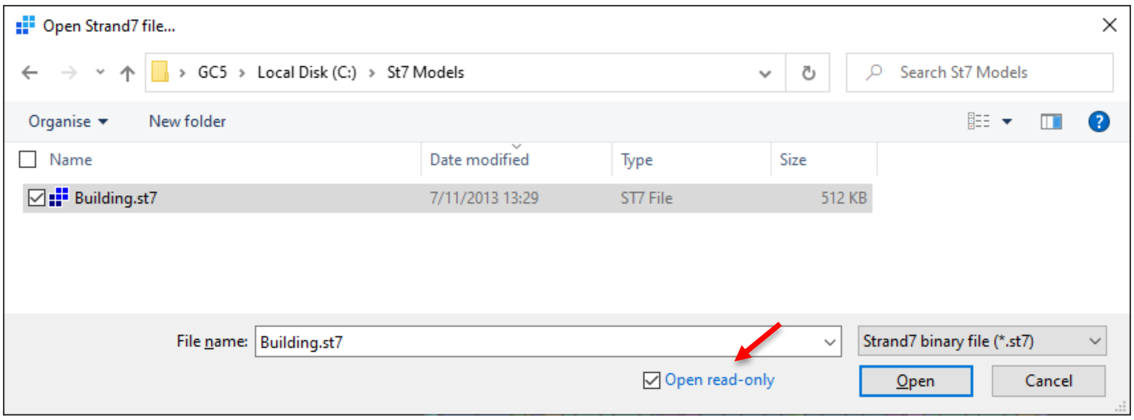
R3 stores much more information in the ST7 file compared with previous Strand7 releases. Consequently, the R3 version of an ST7 file will be larger than the R24 version. This is particularly the case for models that contain many beam cross sections; R3 stores an array of coefficients that relate shear forces and torque to shear stress on the cross section, and this will increase the ST7 file size considerably (more information about this new feature can be found in the **Beam Section Properties and Shear Stress** section below).

As the ST7 file format has changed, once saved in R3, the ST7 file cannot be opened in previous Strand7 releases. To convert an R3 ST7 file to a previous Strand7 release, R3 offers a text file export option that can export the model to any of the previous major Strand7 releases. New features in R3 that are not directly supported by a previous release are converted to equivalent features, where possible. If it is not possible to do that, some data may be lost, in which case a warning message is given on export.



Read-only Files

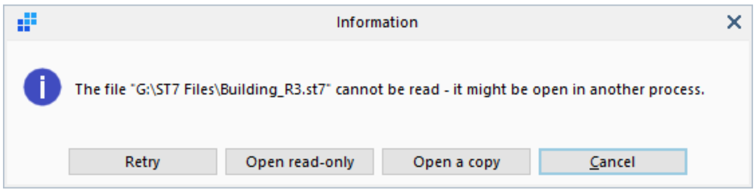
When opening a Strand7 file, R3 offers the option to open the file in read-only mode via a checkbox on the open-file dialog. Files opened in read-only mode cannot be modified or saved. However, result files can be opened for post-processing.



To open the file for editing, a file opened in read-only mode must be closed and re-opened without the read-only setting on the open-file dialog.

File Sharing

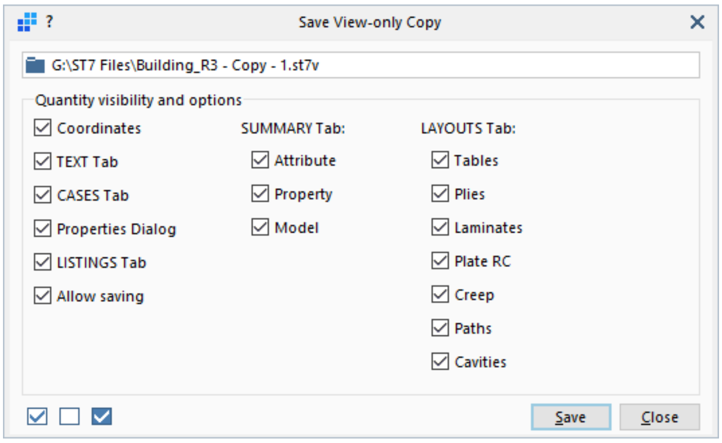
To help reduce the possibility of data loss or model corruption in a network environment with shared folder locations, ST7 files can only be opened by a single instance of R3 at a time. Any attempt to open an already open ST7 file via another instance of R3 (e.g., by another user on the network) will produce the following dialog, which allows for the file to be opened as read-only or as a copy (in addition to the Retry and Cancel options).



ST7V – View-only File

R3 introduces a new version of the Strand7 model file, the *ST7V* file. This is an encrypted, view-only file that allows only certain types of data to be visible, depending on the user-specified settings selected when the file is created. The main purpose of this file is to allow users to provide copies of Strand7 model files (and optionally result files) to a client while preventing the modification of the model data and keeping certain confidential components of the data hidden.

ST7V files are created by the **Saving/Save View-only Copy** function on the Strand7 menu.



ST7V files can only be viewed. They cannot be edited, exported or saved in the ST7 format, and hidden data cannot be copied from them. They cannot be executed by the solvers. Result files may be opened in an ST7V file, enabling post-processing functionality. An ST7V file may be re-saved as an ST7V file, if allowed when first created; the purpose of this feature is to enable the saving of changes to the view and display settings, including the addition of view related components such as sets and viewports. ST7V files can be opened by Strand7 and by the Strand7 model viewer.

Clone Function

In addition to the standard **Save**, **Save As** and **Save Copy** functions, R3 offers the **Clone** function. This function makes a copy of the current model, appends the 'Clone – x' suffix to the file name, and immediately opens the file. This is a more efficient workflow than **Save Copy** or **Save As** followed by **Open**, when both the original file and the copy are to be open at the same time. The feature is available in both pre-processing and post-processing modes.

Result Files

Previous Strand7 releases support a maximum logical result file size of 32 GB, consisting of up to 16 physical files, each up to 2 GB in size. For example, in R24, a 1 GB linear static result file will consist of the single file, *Filename.LSA*, whereas a 3 GB linear static result file will consist of *Filename.LSA* and *Filename.LSA~2*.

With the 64-bit architecture of R3, physical result files can be practically unlimited in size, so a result file will only ever be a single file in R3, no matter how large it is (i.e., R3 does not create multi-file logical result files).

R3 still reads and post-processes previous version Strand7 result files, including those that span multiple physical files. Although R3 can open and post-process result files created by previous Strand7 releases, to take advantage of new results and post-processing features in R3, the model needs to be re-solved in R3.

User Interface – General

The R3 interface layout has changed compared with R24, and consists of two distinct windows:

1. the Startup Screen, and
2. the Model Window.

Startup Screen

The startup screen is the first window displayed when R3 is first launched. It is a compact window that provides options for opening an existing ST7 file, creating a new ST7 file, configuring the settings that control the overall R3 environment (**Preferences**), and opening the online help.

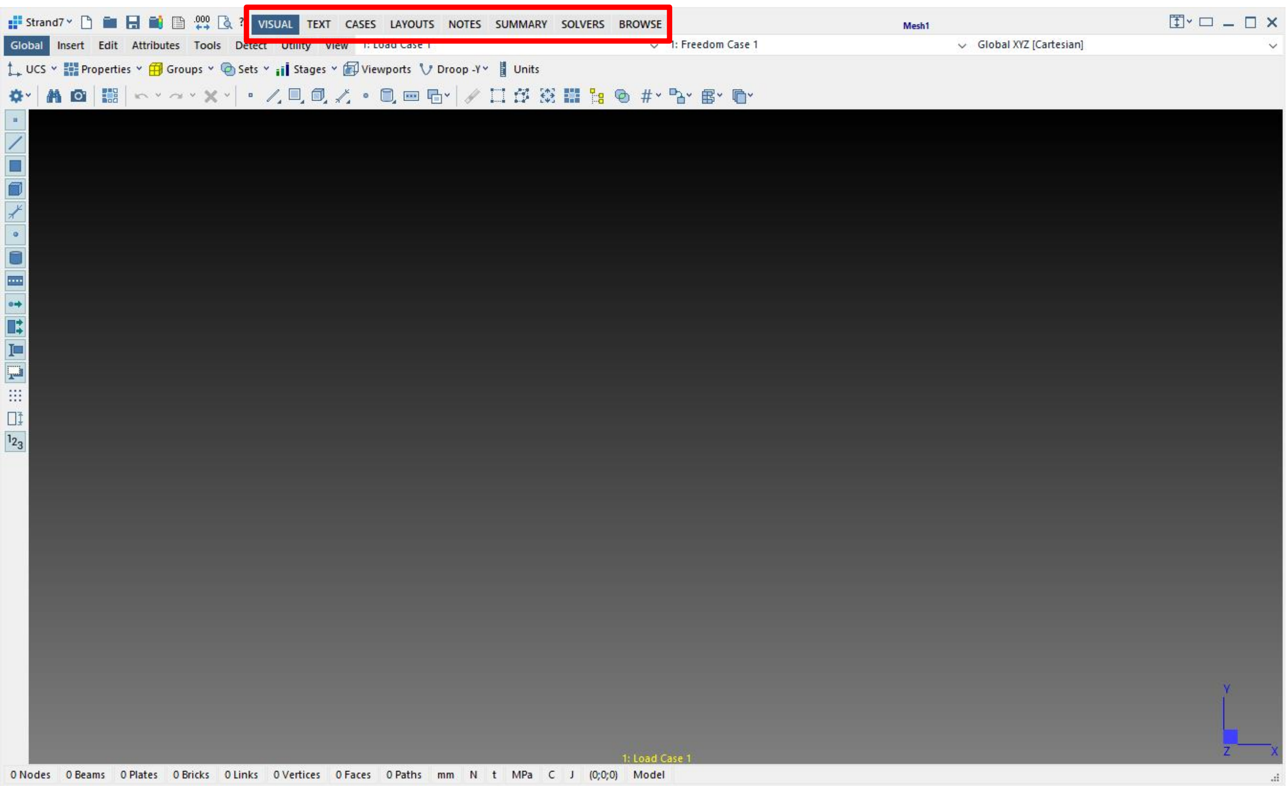
An existing ST7 file can be opened via a file-open dialog by clicking the open folder icon, or by using drag-and-drop from the Windows desktop or File Explorer. Drag-and-drop enables not only the opening of an ST7 file, but also the import of files such as CAD geometry files into a new model, and the opening of Strand7 solver log files.

The startup screen is hidden whenever a model window is open. It is shown again when all model windows are closed. This provides a unique solution to the issue of the application terminating when the last file is closed: by showing the Startup screen after closing all model windows, another Strand7 file can be opened without having the launch the Strand7 application again. This behaviour can be changed via **Preferences** if automatic closing of the application is preferred.

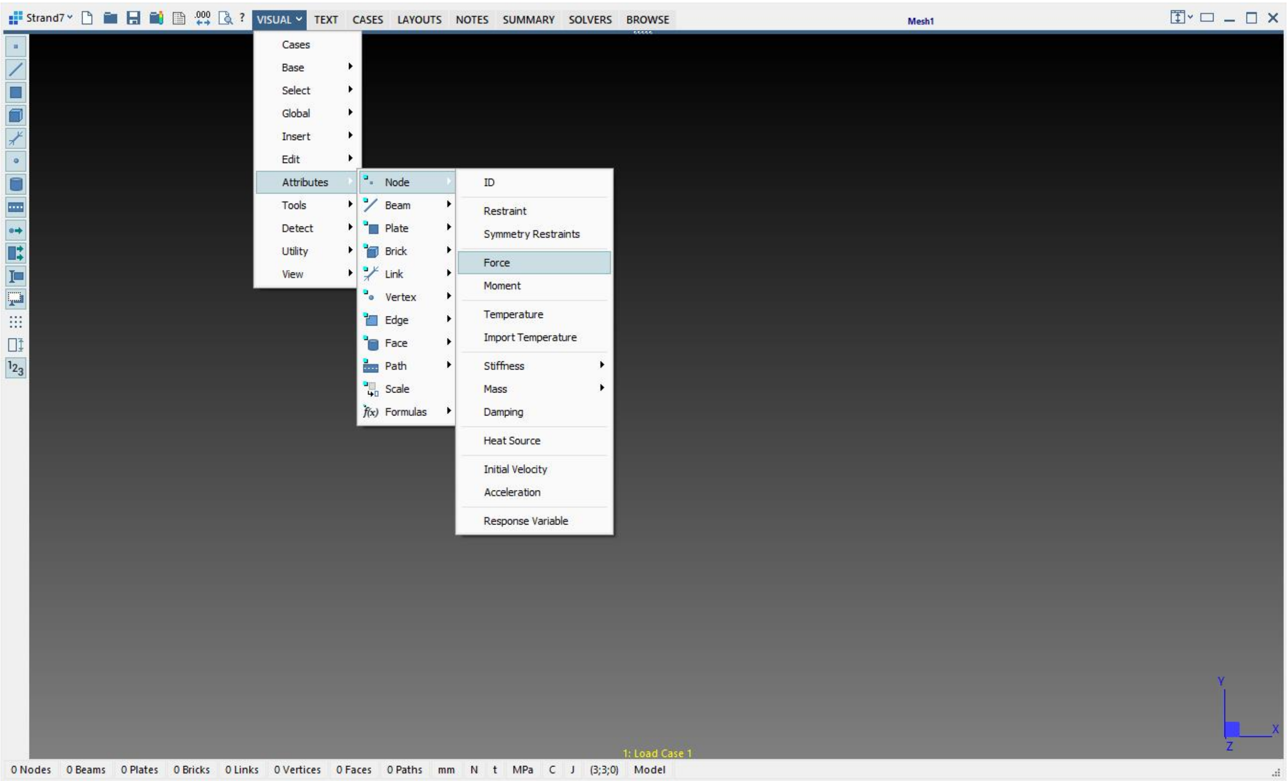
The Startup screen can be minimised when not in use. In this minimised state it uses no CPU resources and very little RAM.

Model Window

R3 retains some elements of the R24 interface, but organises the model data and data views into a set of **Main Tabs**, (**VISUAL/TEXT/CASES/LAYOUTS/NOTES/SUMMARY/SOLVERS/LISTINGS/BROWSE**) some of which contain one level of sub-tabs. Unlike R24, there is no common menu toolbar for multiple model windows. Instead, each model window consists of its own set of tabs, controls and menus.



The tabbed layout offers a reasonable amount of customisation; from the fully expanded view shown above, to the fully compacted view shown below, and various levels in between. In compact view, all functionality is still available, but it is provided via dropdown menus instead of through persistent icons and sub-tabs. Compact view is particularly useful for maximising model viewing area when multiple models are open and tiled side by side on a single screen.



At the top-left of the model window, various commonly used icons are permanently displayed alongside the Strand7 menu icon. These are the **New**, **Open**, **Open/Close Results**, **View Log File**, **Format Numbers**, **Print Preview** and **Help** icons.

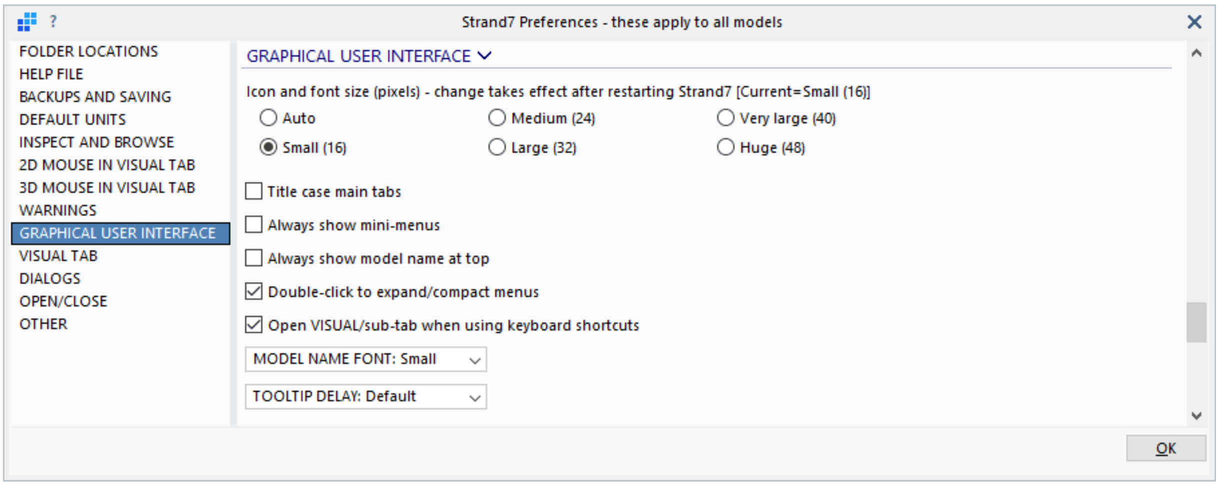
At the top-middle of the model window the main tabs are displayed. These tabs provide fast access to different components of the model data, or different views of the same data. For example, the **VISUAL** tab provides a graphical (i.e., visual) representation of the model data whereas the **TEXT** tab provides a numeric (i.e., text based) view of the same data. Both views allow for data to be inspected, modified, inserted and deleted; one does it graphically, the other does it numerically.

Multiple ST7 files can be open at the same time. Another ST7 file can be opened by clicking the **Open** icon on the model window or by using drag-and-drop from the Windows desktop or File Explorer directly onto any Strand7 model window. CAD and other files can be imported into the current model by using drag-and-drop onto the model window.

Icon Size and High DPI

The R3 GUI offers five icon and text sizes for the display of text and icons. Icon pixel sizes can be selected from the set of 16x16, 24x24, 32x32, 40x40 and 48x48 pixels, with corresponding font sizes automatically applied. This enables the selection of an appropriate icon and font size for comfortable viewing on virtually any screen resolution, and avoids the blurry icons that result when an application is automatically scaled by Windows. In addition, it does not compromise the graphics resolution inside the model window, which also occurs when using automatic scaling by Windows (for example, a screen with 1920x1080 pixel resolution and 150% scaling by Windows will appear to have 1280x720 pixels to a scaled application). The R3 model window resolution is always one to one with the true screen resolution.

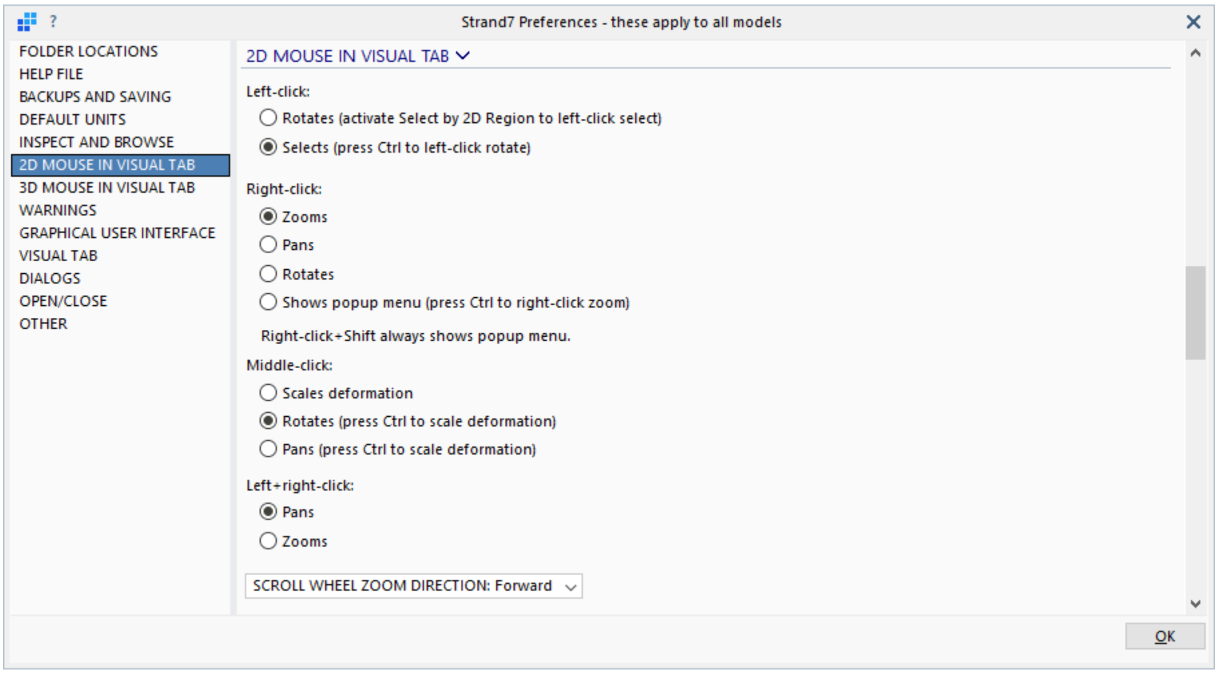
The **GRAPHICAL USER INTERFACE** section of **Preferences** is shown below. In addition to the explicit icon and font size selections, the **Auto** option is available; this attempts to automatically select a good size based on the screen resolution and screen DPI settings. Changing the icon and font size for the GUI requires a Strand7 restart.



Dynamic Rotation

In R3, the view manipulation functions are always available without the need to enter a special dynamic rotation mode (in R24, F4 puts the model window into dynamic rotation mode and it stays in that mode until the dynamic rotation toolbar is closed). In R3, dynamic rotation is always available whenever R3 is not in the middle of some function (e.g., when not part way through a selection operation), and therefore the view can be easily manipulated at any time by using the mouse.

Similarly to R24, the default actions are left-click to rotate, right-click to zoom and left+right-click to pan. Additionally, R3 offers some customisation of the functions to be performed by the various mouse buttons via the **2D MOUSE IN VISUAL TAB** section of **Preferences**, as shown below.



For rotation, two modes are provided: screen axis rotation and model axis rotation. By default, a left-click-drag action will rotate the model about the screen axes (x, y and/or z, depending on the location of the mouse pointer). To rotate about the model's global X axis, hold the '1' key on the keyboard while dragging; similarly, hold the '2' key to rotate about global Y and the '3' key to rotate about global Z. To rotate only about the screen x axis, hold the '4' key; similarly, hold the '5' key to rotate only about the screen y axis and the '6' key to rotate only about the screen z axis.

When R3 is part way through some function (e.g., during a selection operation), dynamic rotation is available by pressing the Ctrl key on the keyboard (depending on the preferences set above). This keystroke temporarily pauses the active function, enabling the mouse to be used for view manipulation. As soon as the Ctrl key is released, R3 immediately resumes the currently paused function. All functions can be paused in this way, and therefore it is not necessary to terminate a function in order to change the view.

R3 also responds to the scroll wheel on the mouse for zooming (in addition to the right-click zoom). The zoom in/zoom out behaviour can be configured for either scroll forwards or scroll backwards, depending on the user's preference.

The rotation origin in R3 can be easily changed by a double-click action on any point on the model. For example, double-click an element to set the rotation origin to the spatial location of that point on the element. The default rotation origin can be reset by pressing F5.

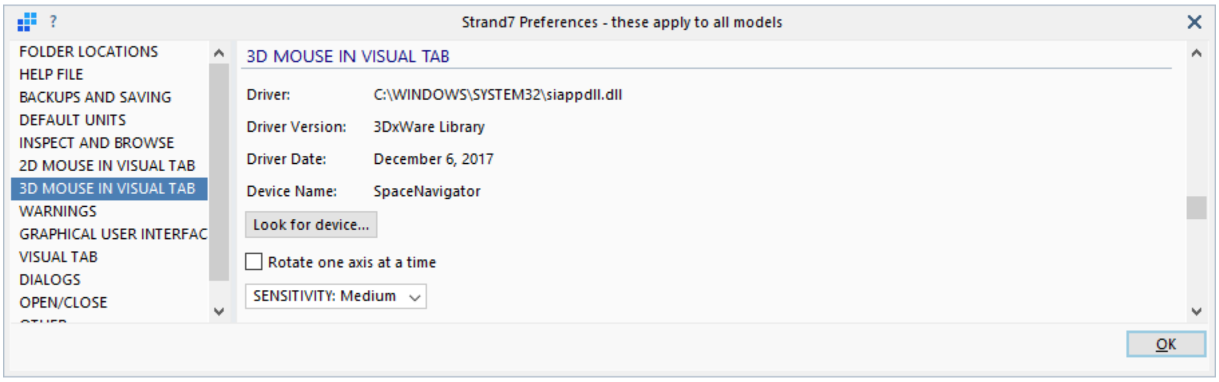
As the dynamic rotation functions are always available in R3, the right-click popup menu is no longer provided on the model window by default (this can be changed via **Preferences** if desired). To show the popup menu, use shift+right-click in the model window. Additionally, the popup menu is always available by a right-click on the main toolbar of the model window.

3D Mouse

3D mice such as SpaceMouse are supported by R3. These devices can provide an intuitive way of interacting with a Strand7 model for view rotation, zooming and panning.


Images from https://3dconnexion.com/au/spacemouse/.

R3 offers some options for these devices, including the option to manipulate a single axis at the time. The single axis mode can be easier to use when using these devices for the first time. The options can be set via **Preferences**.



Touch

The default 2D mouse functionality used by R3 can be duplicated on a touchscreen such that virtually all of the view manipulation and modelling actions can be performed by touch. The complete set of gestures supported is tabulated below.

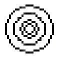
Gesture	Results
One-finger drag 	Rotates the model as with the left mouse button.
Two-finger drag 	Pans the model horizontally and vertically as with the left+right mouse buttons.
Two-finger rotate 	Rotates the model about the axis out of the screen by relative rotation of fingers.
Two-finger pinch 	Zooms the model in and out as with the right mouse button.
One-finger long tap 	Tap-and-hold for one second before releasing to open right-click popup menus.
Two-finger tap 	Shows the Entity Inspector; equivalent to Shift+Hover mouse.
Press and tap 	Tap-and-hold one finger while tapping another finger to retrieve attribute values from the entity at the held-down finger; equivalent to Shift+Ctrl+Click.

Scroll Wheel Button Displacement Scaling

Besides the zoom functionality, when in post-processing mode (i.e., when a result file is open) the scroll wheel button can be used to directly change the displacement scale of the deformed model: click and hold the wheel as a button and drag the mouse upward to increase the scale; click and hold the wheel as a button and drag downward to decrease the scale.

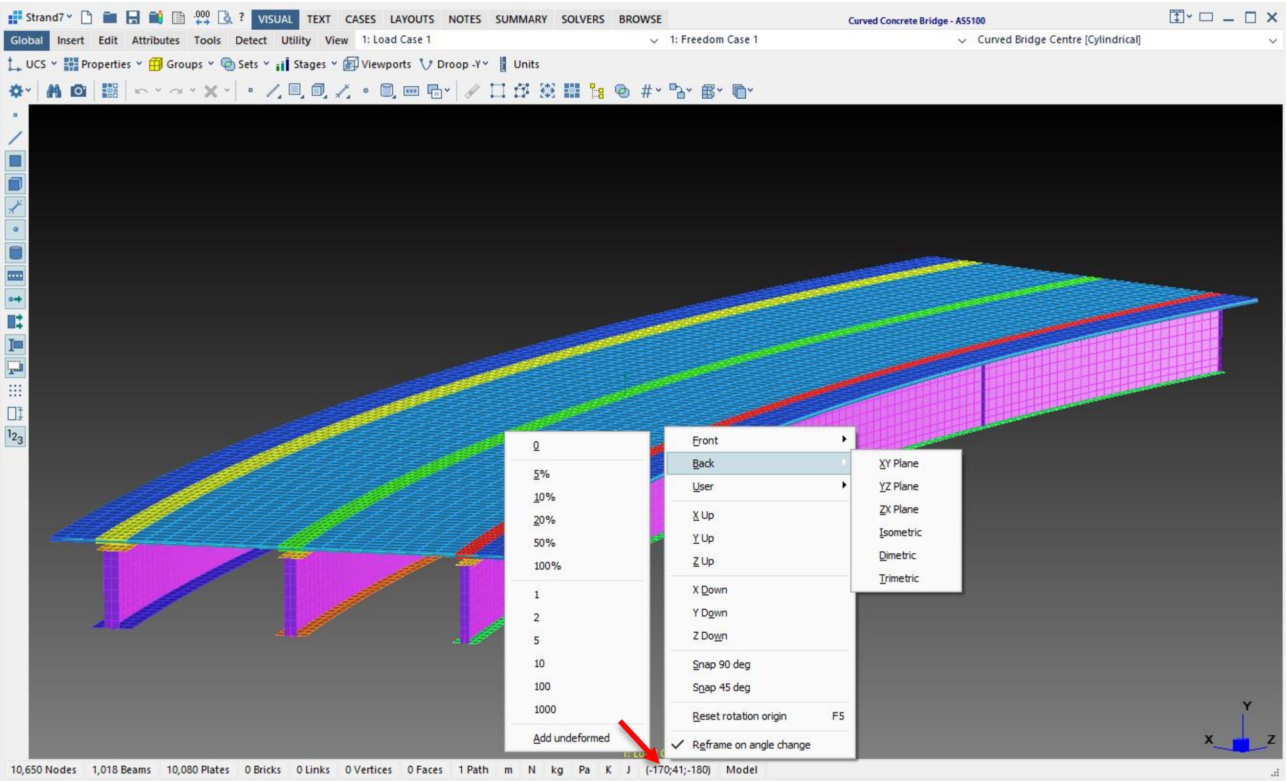
Context Specific Mouse Cursor

In the VISUAL tab, R3 uses a range of mouse cursors to signify the active function. This provides instant feedback about the current mode.

Cursor	Function
	Default Windows cursor – no specific function.
	Dynamic rotation with 2D mouse.
	Dynamic zoom with 2D mouse.
	Dynamic pan with 2D mouse.
	Double-click point for setting rotation origin.
	Dynamic displacement scaling with 2D mouse via scroll wheel button.
	Explicit **VISUAL/View/Pan** function with 2D mouse.
	Explicit **VISUAL/View/Zoom in** function (zoom window) with 2D mouse.
	Explicit **VISUAL/View/Zoom out** function (zoom window) with 2D mouse.
	Insert element by clicking nodes, vertices or grid points (**VISUAL/Insert** – **Element/Link/Path**).
	Select by 2D region.
	Select by 3D region.
	Zoom in / zoom out in print preview mode.
	Equation editor.
	Retrieve by hot pointer to dialogs such as the Attribute dialogs.
	Peeking pointer to extract a node or element result to the Peek window.
	Drag-and-drop to reorder lists.
	Drag-and-drop files onto the startup screen or model window.
	Help topics.

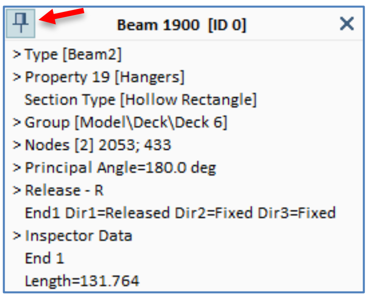
Quick Views and Quick Displacement Scale

A right-click popup menu on the view angles cell of the status bar offers a list of quick pre-set views. This includes useful new functions to align one of the axes up or down, or to snap the view to the nearest 90 deg or 45 deg viewing angles. A similar popup is available in post-processing mode to quickly set the displacement scale.



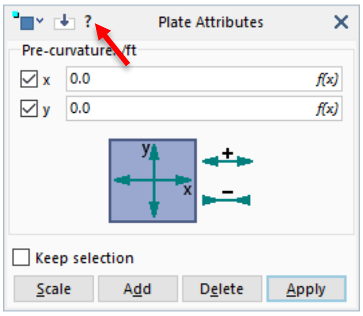
Pin Entity Inspector

The entity inspector can be pinned to keep it in one place instead of it following the mouse around. The font can be configured for the entity inspector via **Preferences** or via a right-click popup menu on the entity inspector itself or via the popup menu in the model window.



Help Icon

The help button, with the **?** symbol, is available on most of the R3 dialogs and on the model window to open a context-specific help page. Alternatively, the F1 key can be pressed at any time to open the currently relevant help topic.

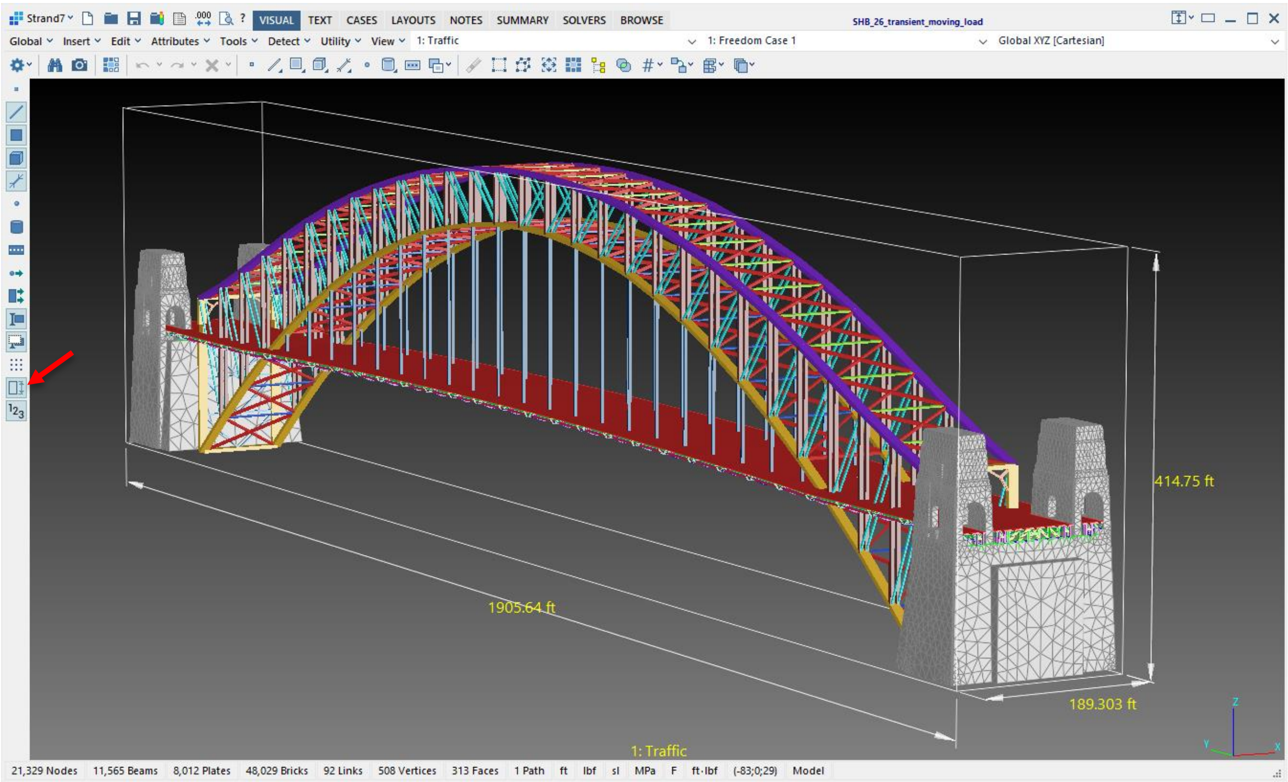


User Interface – Productivity Features

The features listed in this section provide useful functionality within the R3 working environment.

Model Bounding Box

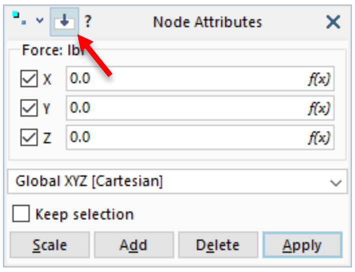
A quick dimensioned bounding box is available by clicking the bounding box icon on the entity toggles toolbar. The bounding box can be configured for colours, fonts and other options by right-clicking the icon.



Retrieve Icon

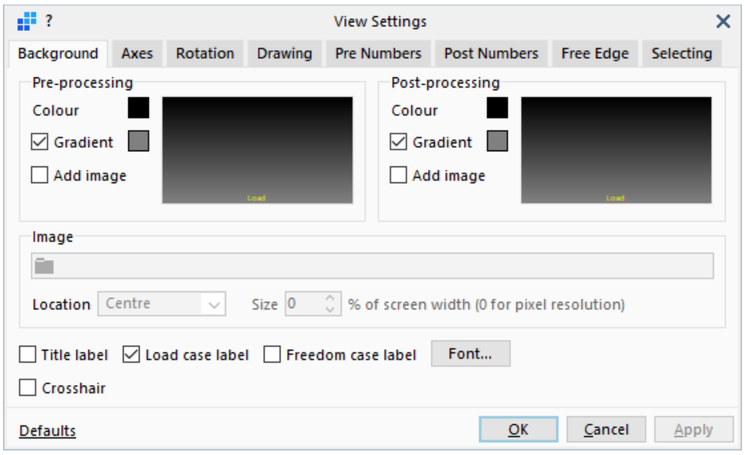
Throughout the R3 interface there are situations where data can be extracted graphically from an entity in the **VISUAL** tab of the model window for retrieval into a dialog. This includes the retrieval of entity attributes from the model to the attribute dialog, the selection of an origin to define a User Coordinate System (UCS), the selection of the end points for a graph, and so on.

In all of these situations the retrieve icon appears on the respective dialog. Clicking this icon activates the hot pointer so that the next click on the model window retrieves the required value(s). Note that the keyboard sequence Ctrl+Shift can also be used on the attribute dialogs to immediately enable the hot pointer without pressing the retrieve icon.



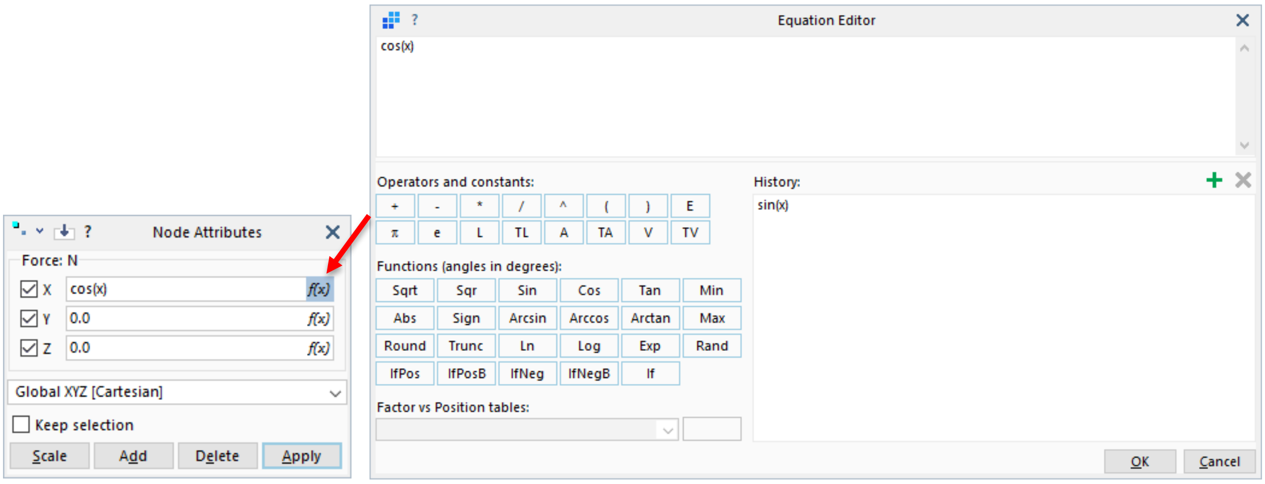
Window Background

The background settings can be configured differently for pre-processing and post-processing. The background automatically switches from one to the other as result files are opened and closed.



Edit Box Equations

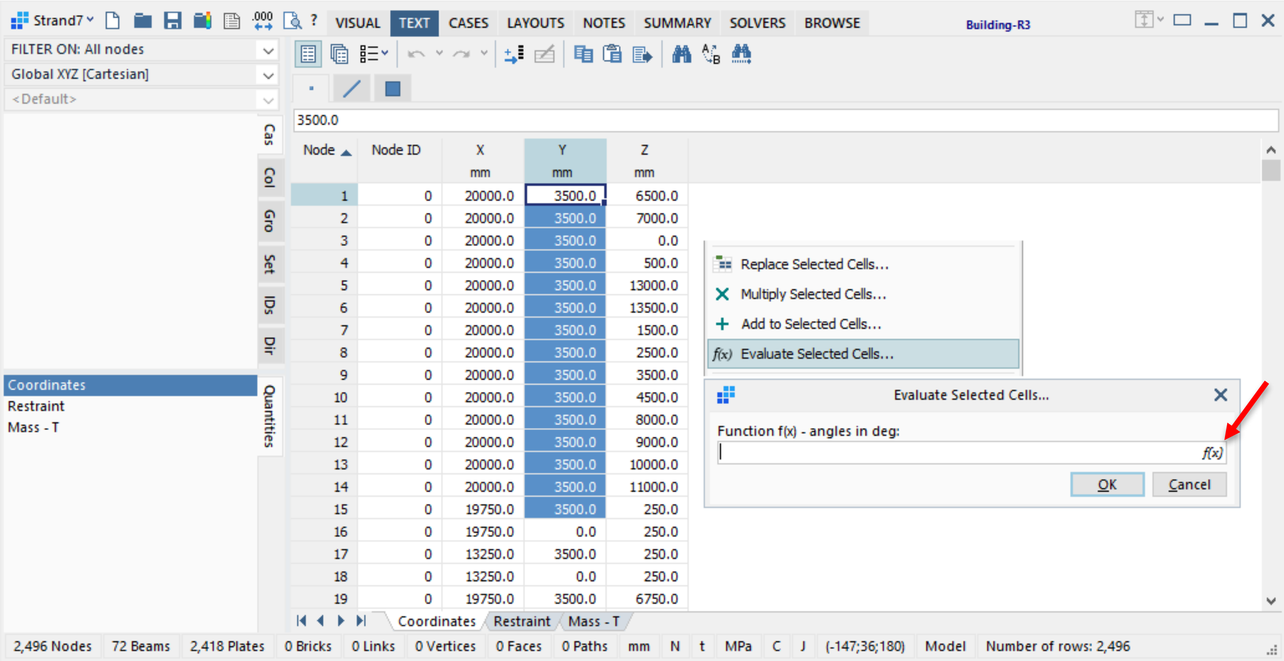
Some edit boxes used for the input of real numbers offer the option to open the equation editor for the insertion, editing and retrieval of equations. Multiple equations can be defined and added to the history list. These equations are stored with the model and can be reused by selecting them from the list.



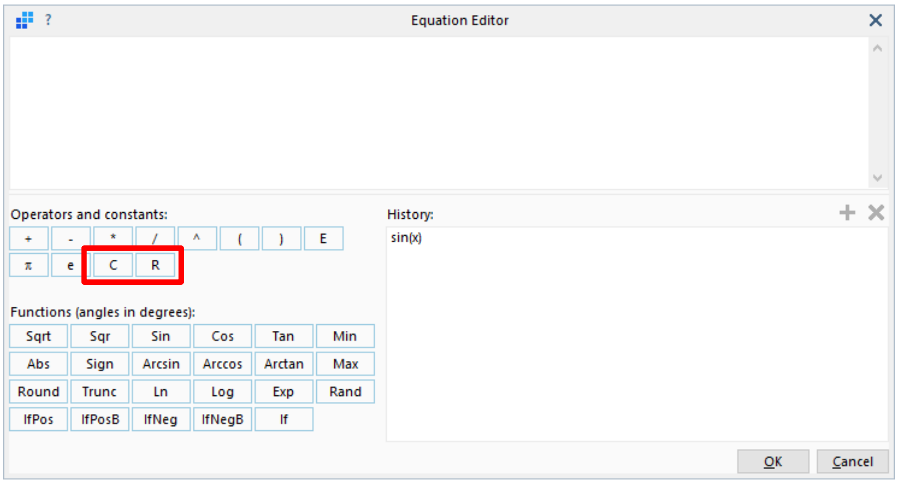
New functions are available in R3, including multi-parameter functions such as MIN(,,,...) and MAX(,,,...), and a three-parameter IF(,,) function.

Grids – Evaluate Selected Cells

In addition to the previously available **Replace Selected Cells…**, **Multiply Selected Cells…** and **Add to Selected Cells…**, the new function **Evaluate Selected Cells…** is available in R3 editable grids. With this function, the value of the highlighted cells can be referenced in an equation via the symbol ***x***. This enables more complex modification of grid data by using the existing values as variables in an equation that produces new values.



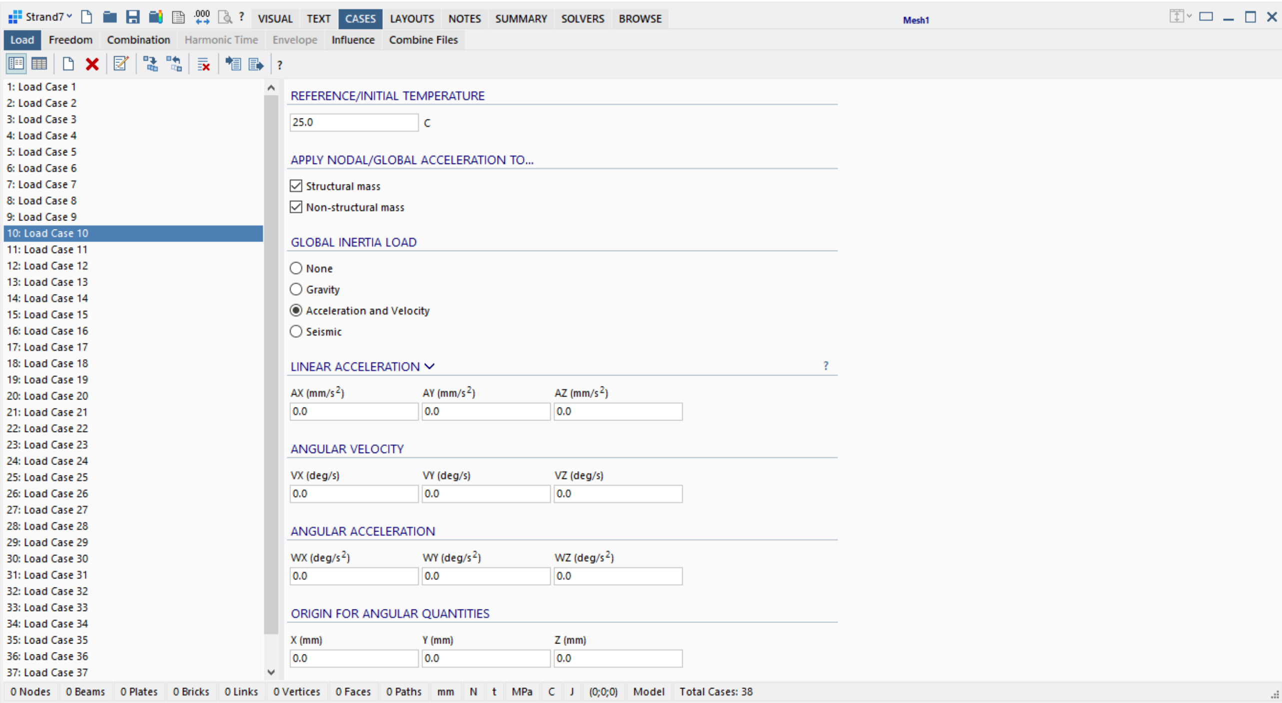
Note that when the equation editor is used in grids, a different set of operators and constants is used. This includes the operators **C** and **R**, which refer to the grid columns and rows, respectively. This is useful for automatically generating column/row specific data in a grid.



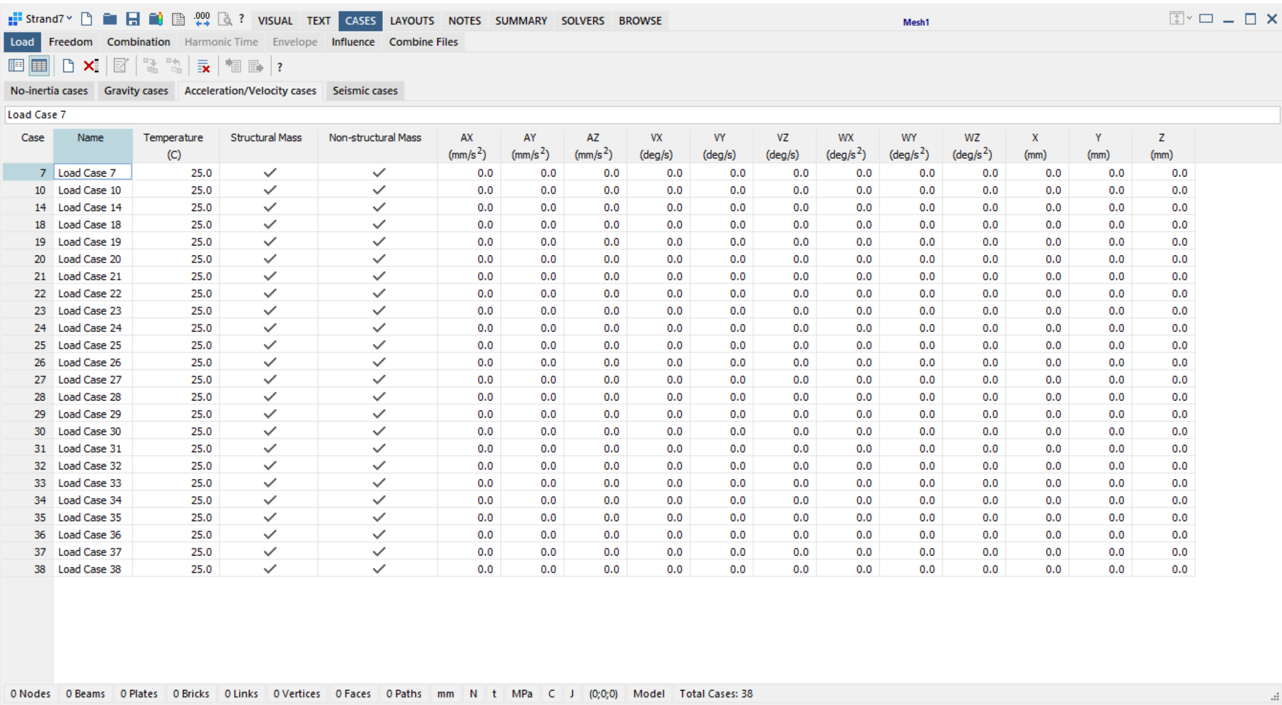
Load and Freedom Cases – Case View vs Grid View

The load case and freedom case sub-tabs under the **CASES** tab, offer two different views: the *list view* and the *grid view*.

List view provides the list of all available cases together with a data pane on which data can be inspected and edited. This is similar to R24.

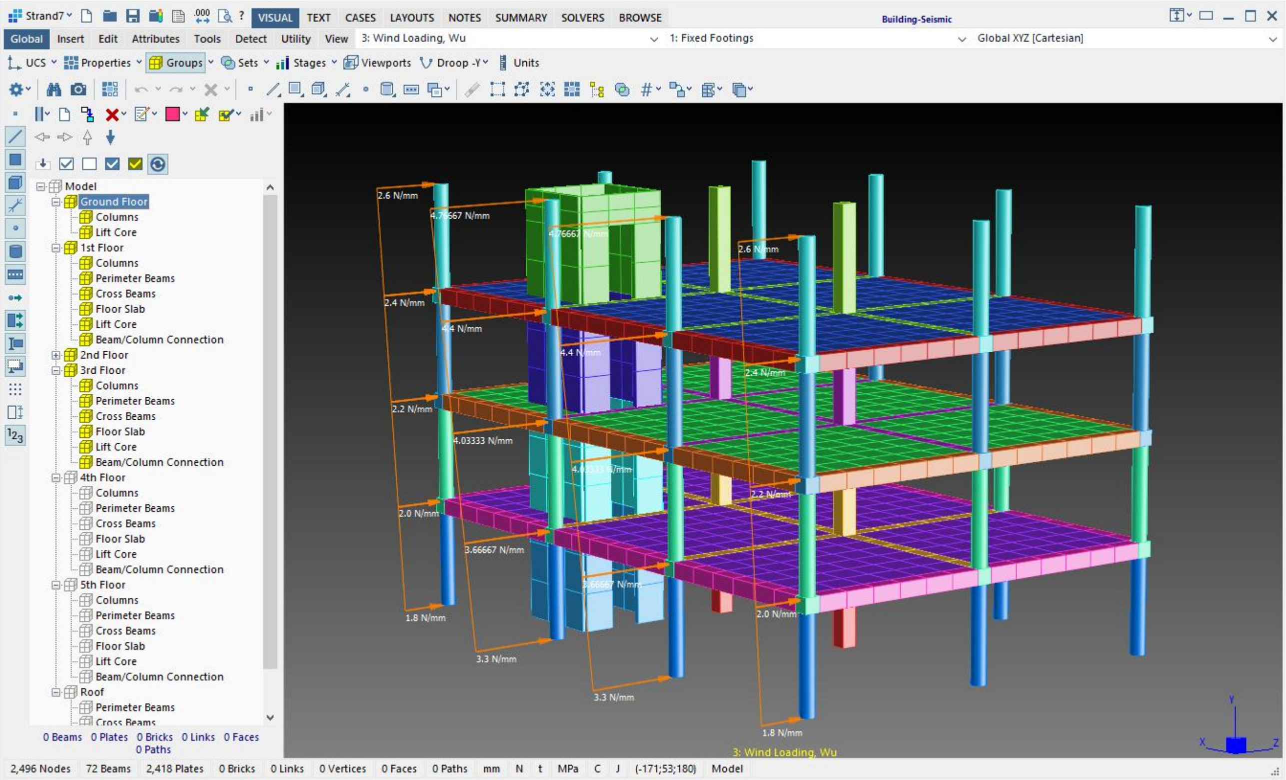


Grid view presents the data for all cases of the same type in a grid for easier bulk editing of data in multiple cases at once.

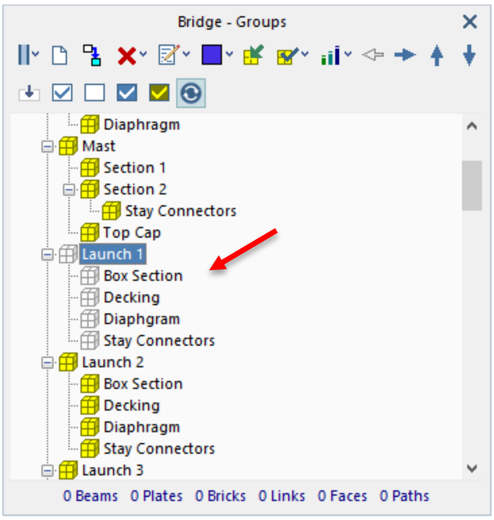


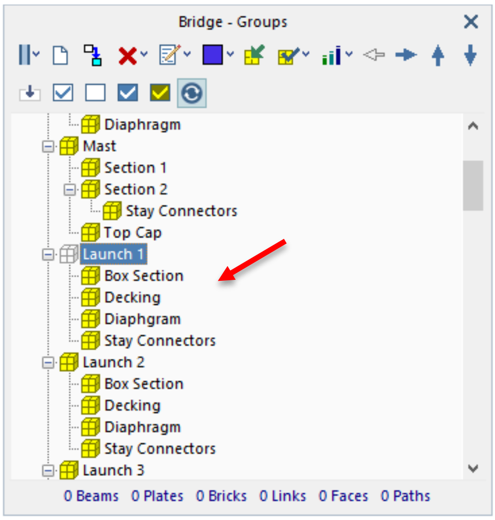
Group Tree Functions

The group tree is a floating window by default, but in R3 it can also be docked (left or right) inside the **VISUAL** tab of the model window.

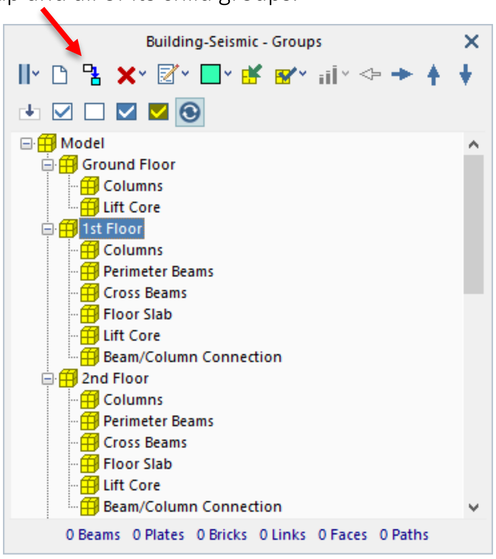


- Click the yellow cube to toggle the clicked group and all of its child groups, as in previous Strand7 releases.
- Ctrl+click the yellow cube in R3 to toggle only the clicked group.
- Shift+click a group to toggle the range of groups between the currently selected group and the newly clicked group. The range can cross group levels.

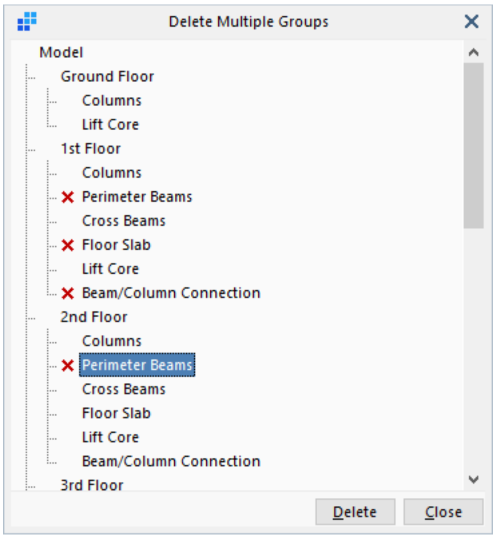




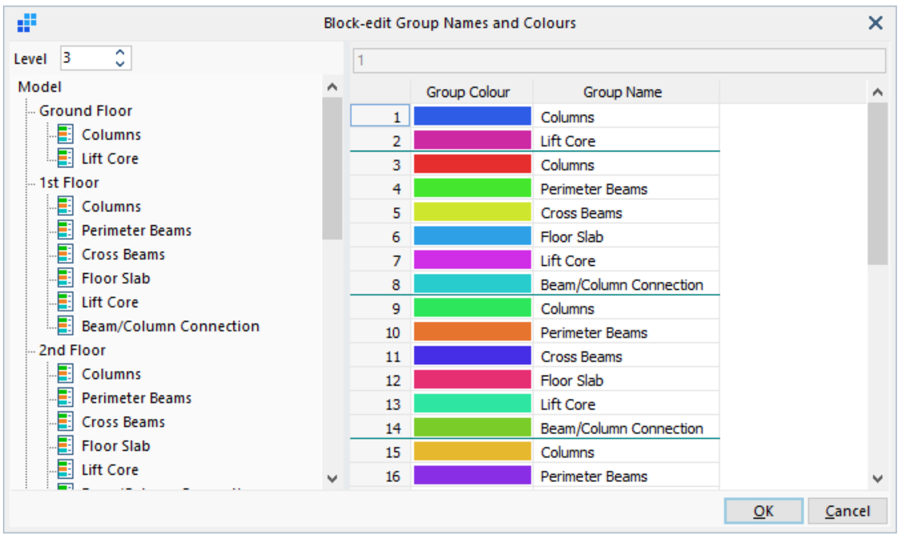
Click the clone button to clone a group and all of its child groups.



The R3 multi-delete feature facilitates the deletion of multiple groups in one delete operation after selecting them on a group tree. Groups can be selected for deletion no matter where they are in the hierarchy; all selected groups will be deleted. Groups not selected for deletion, but which have parent groups that are selected for deletion, will be re-parented to the nearest parent in the hierarchy so that the group tree remains connected.



The block edit function facilitates the bulk editing of group names and colours in a grid view. A level is selected and all groups at that level are listed together for editing.

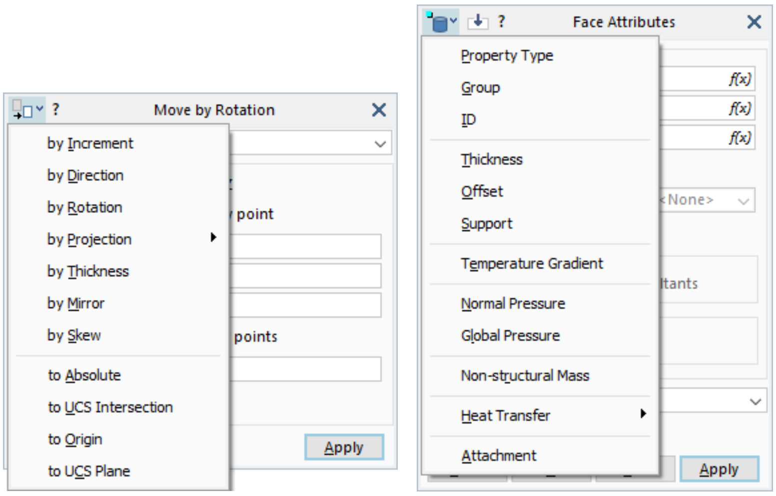


Other productivity tools on the Group tree, available from the right-click popup menu, include:

- **Expand/collapse to level**
- **Set parent colour level**
- **Set groups according to stage definition**

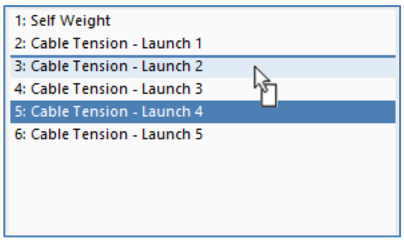
Dialog Box Menus

A number of dialogs, including attribute and tool dialogs, provide a menu of related tools/attributes for faster access.

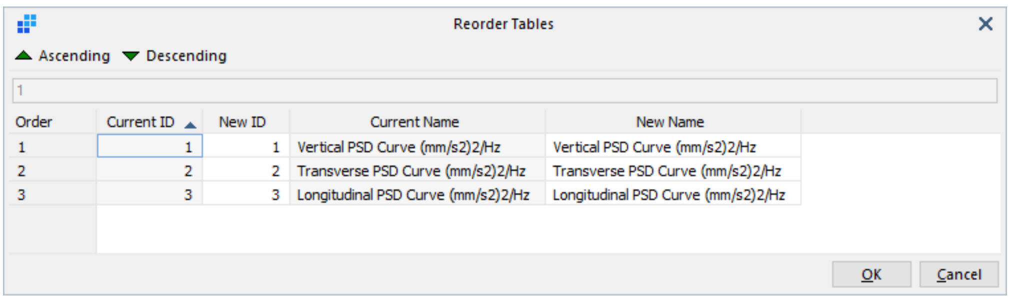


Drag-reorder Lists

The majority of lists for data sets such as load cases, freedom cases, tables, element properties and so on, can be reordered by using standard drag-and-drop actions directly on the lists. Item numbers are automatically updated since they must always be in increasing order in the list. References to the items are automatically updated so that a reorder does not modify model data.



Bulk reordering, renaming and renumbering is also available via a grid view of the data.

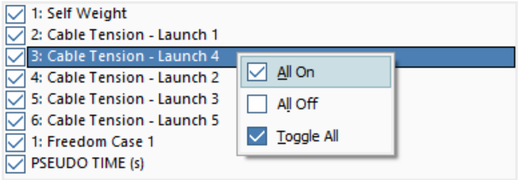


Right-click Functions

Many of the R3 controls have control-specific right-click functions via popup menus.

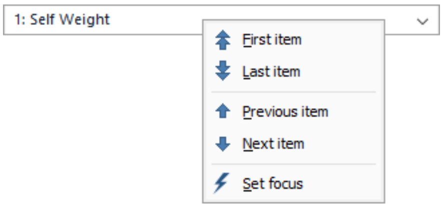
Checkbox lists

Select, unselect or toggle options.



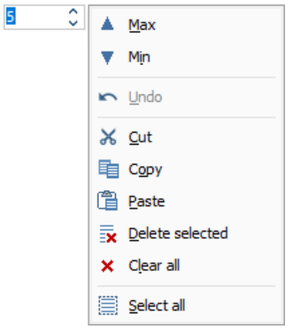
Combo boxes

Quickly navigate to the first item, last item, previous item or next item. Focus the control to then enable changing the item by using the arrow keys.



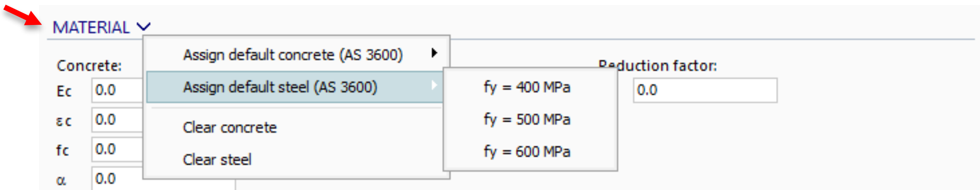
Up-down controls

Quickly set the Min or Max value.

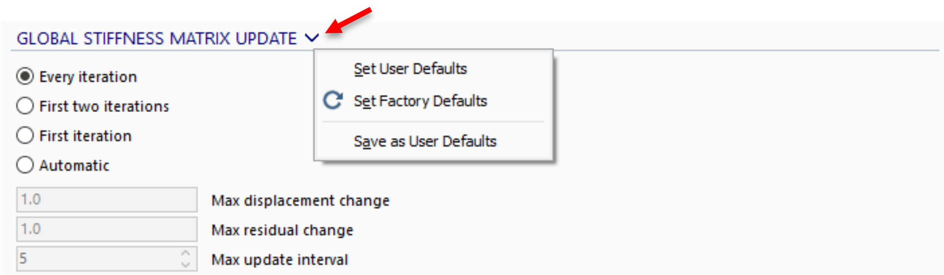


Caption menus

Throughout the R3 GUI, sets of controls can be found grouped together under an all-caps caption menu. In most cases these caption menus provide a popup menu of relevant functions by a left or right click on the caption. For example, in the RC Layouts tab, the MATERIAL caption provides a list of commonly used material parameters for quick selection.

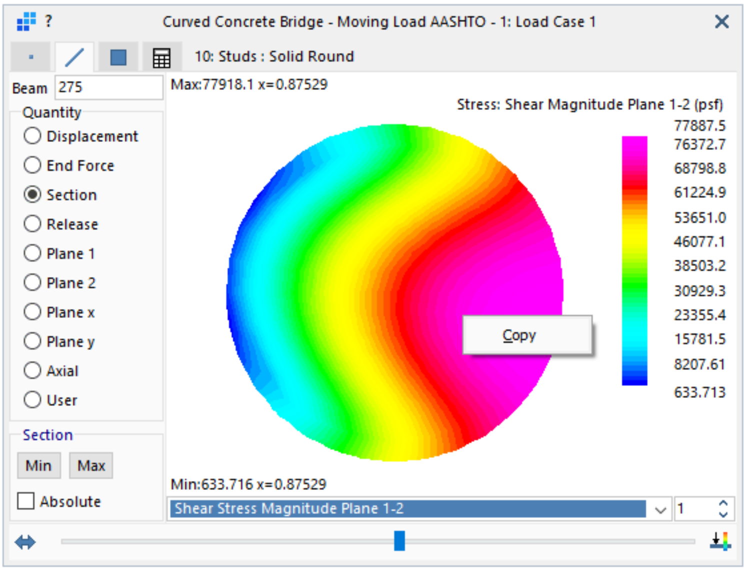


In the SOLVERS/Parameters tab, the various captions provide options to set user or factory results, or to save the current settings as user defaults.



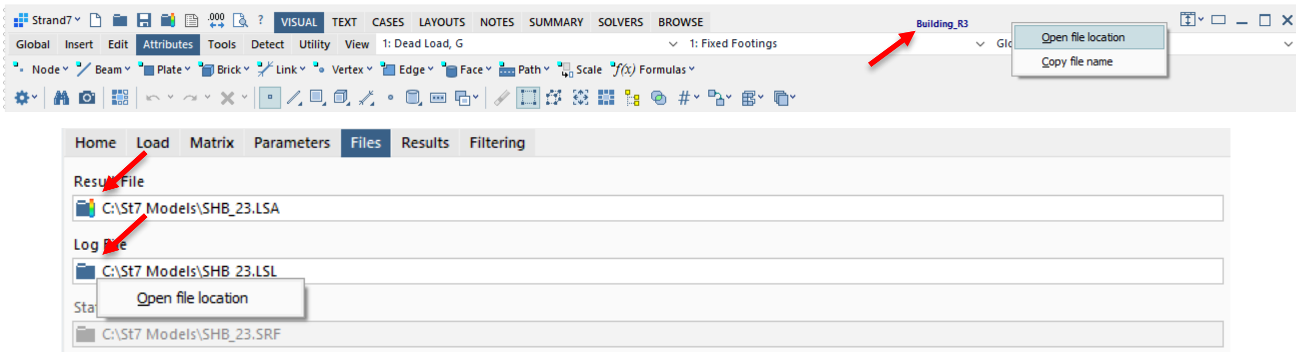
Copy image

Dialogs that display images offer a right-click **Copy** function to copy the image to the Windows clipboard without copying the entire dialog. The image can then be pasted into another application. Note that this does not apply to the model window, where the image can be copied to the clipboard by clicking the camera icon on the main toolbar, or by the keyboard sequence Ctrl+Alt+F.



Open file location

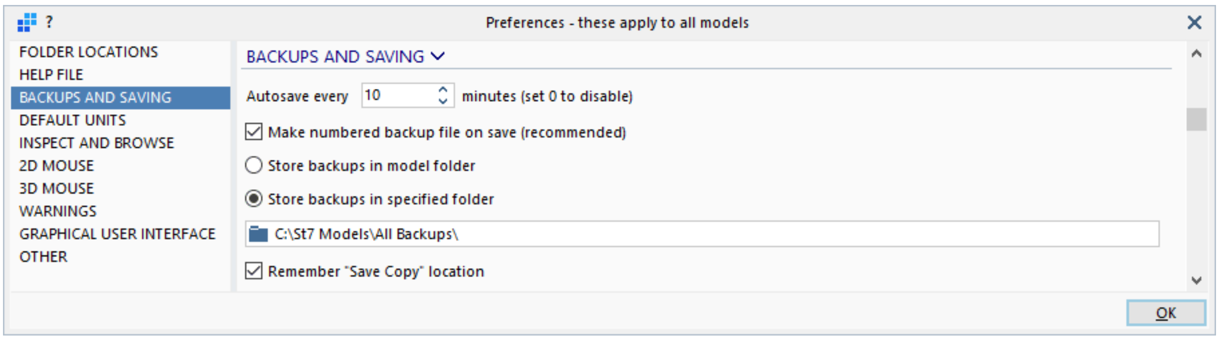
This is available by right-clicking the model name at the top of the model window, and by right-clicking any of the open folder icons associated with file names. It brings up a popup menu to launch File Explorer to open the folder.



Preferences

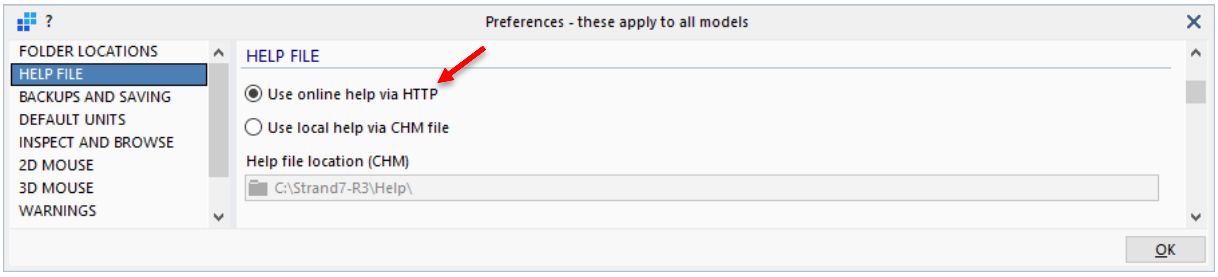
Backup Location

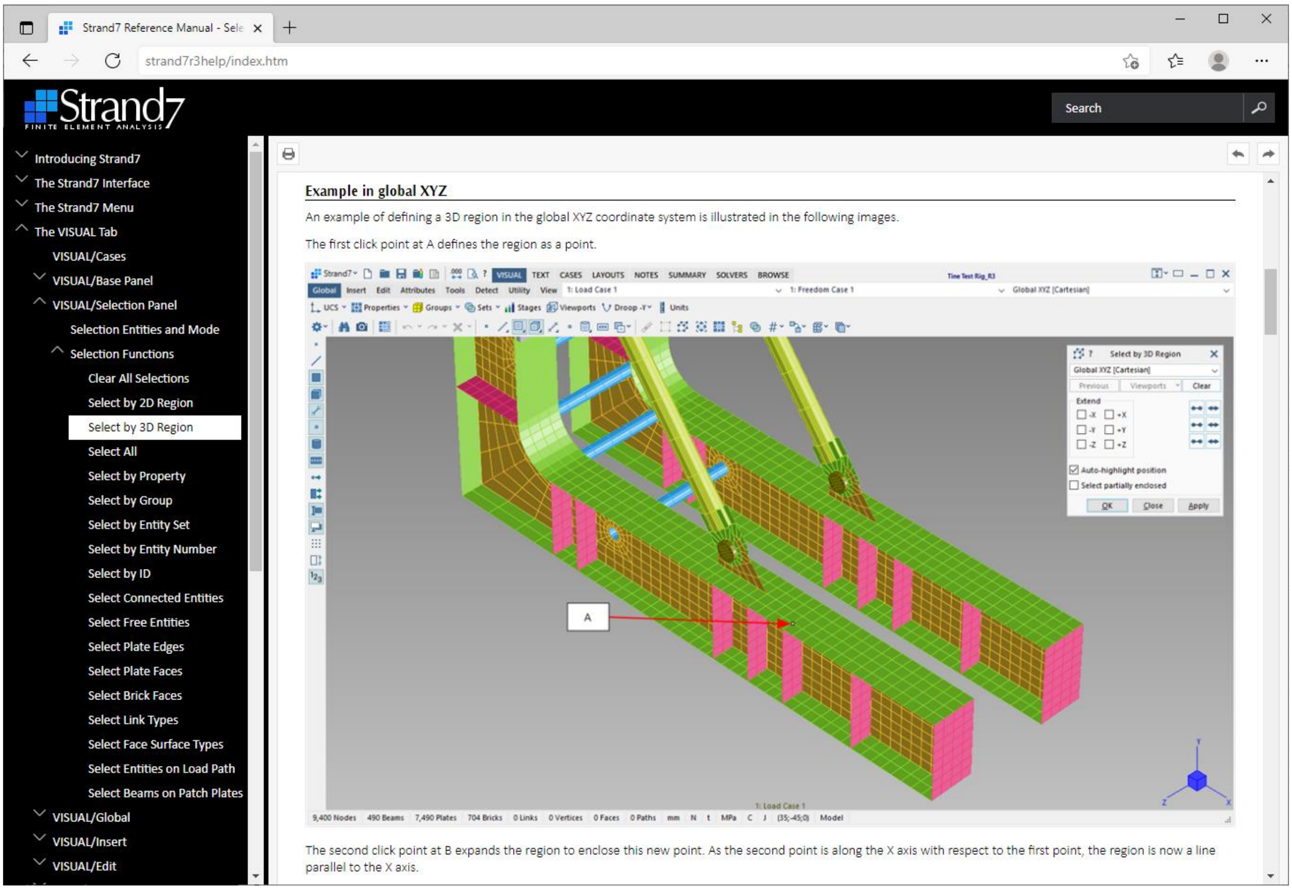
In addition to the previously available **Autosave** and **Make numbered backup file on save** options, the location of the folder in which to store the backups can be user-specified in R3. When the backups are stored in the model folder, a subfolder with the name _*St7Backup* is created and backups are stored there.



Help File

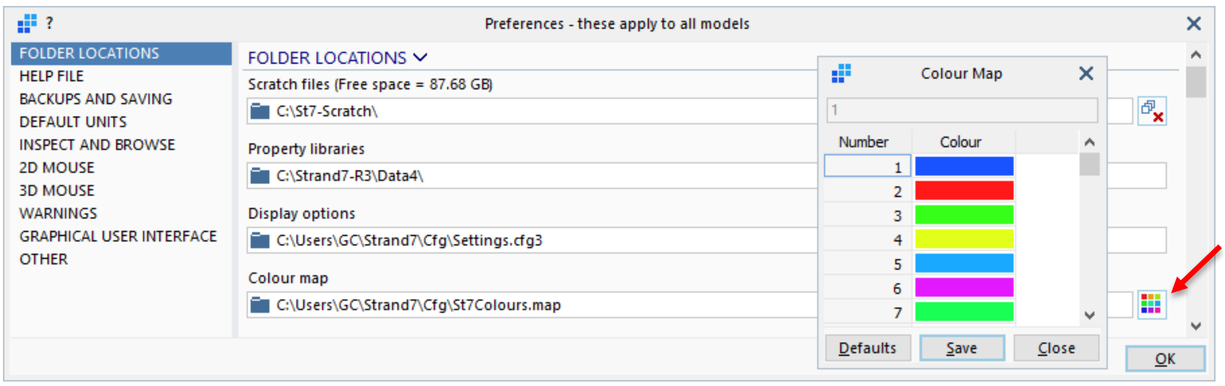
R3 offers two options for the provision of help pages: either online via http (requiring connection to the Internet) or via a CHM file located on the local drive or a network drive.





Colour Map

When a new data set is created and this data set uses colours (e.g., an element property set) a default colour is assigned based on the ID of the data set being created. The default colours associated with each ID can be user-defined by editing the colour map. Colour maps can be stored to a file for later retrieval.



Selection Tools

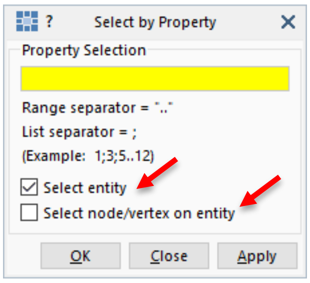
All of the selection tools in R24 are available in R3. R3 offers new selection tools and additional options on the previous tools. Selecting a region with the mouse on the model window in screen space is now referred to as **Select by 2D Region** (2D referring to screen coordinates). What was previously referred to as *Select by Region* is now **Select by 3D Region**, as this selects based on the 3D model coordinates.

Entity Select Toggles Multi-select

Unlike R24, which uses an additional toggle icon to distinguish between single and multi select toggles, the toggles in R3 are in single select mode by default. Multiple toggles can be activated at the same time in R3 by using Ctrl+Click. To turn a multi-select set of toggles into a single-select toggle, double-click the single toggle or press the U key to deactivate them all.

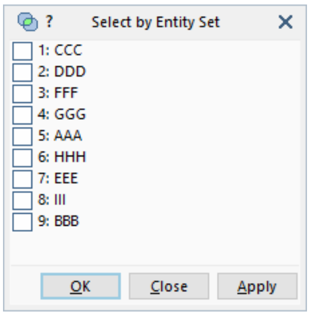
Entity Select

All of the selection tools offer the option to select the entity and/or the node or vertex on the entity.



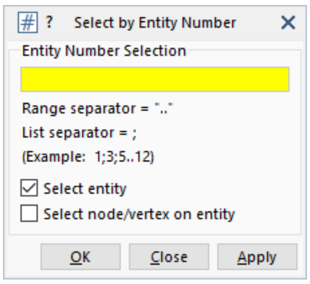
Select by Entity Set

Entity sets are described below. Once sets are defined, entities within sets may be selected by this new selection tool.



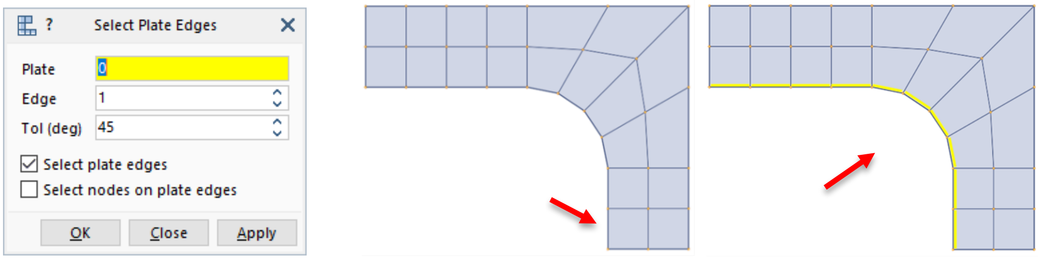
Select by Entity ID or Number

This selection tool selects entities by their ID (if assigned), or by their number (which is always assigned). The edit box supports multiple values as well as ranges (these are defined by a ".." two dot separator – for example, "2..12" will select property types 2 through 12).



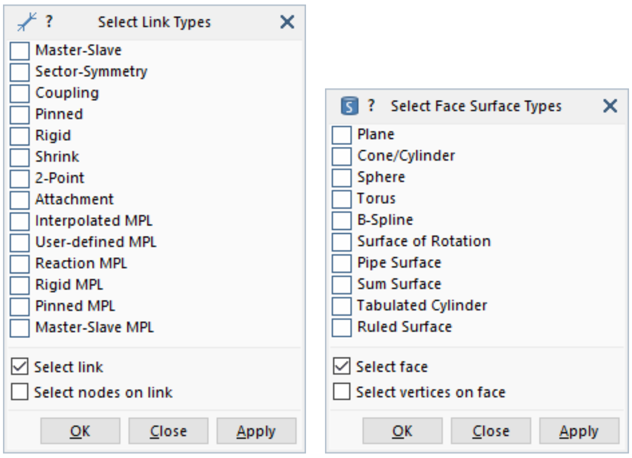
Select Plate Edges

This selection tool complements the previously available **Select Plate Faces** tool. It enables a continuous edge of plates to be selected by choosing a starting edge and an angular tolerance that determines where the continuous edge stops. In the following image, the entire inside edge of the mesh is selected using a single selection action.



Select Link Types / Select Face Types

These selection tools select links and faces, respectively, based on their underlying types.



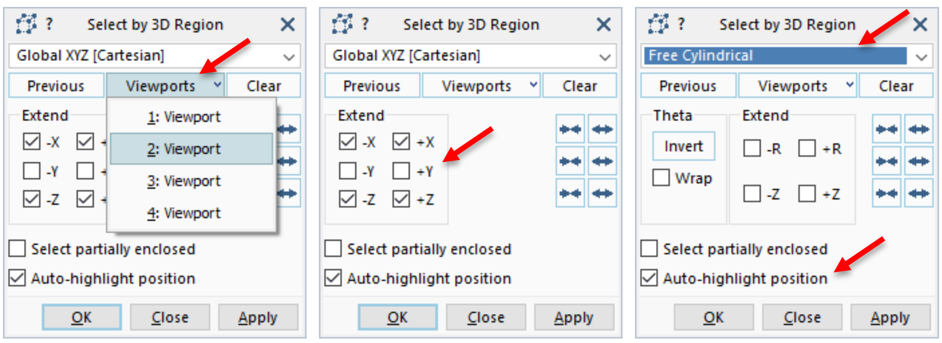
Select by 3D Region

This selection tool can use a previously defined viewport as the selection region in R3. This effectively provides a way to store a selection region for reuse.

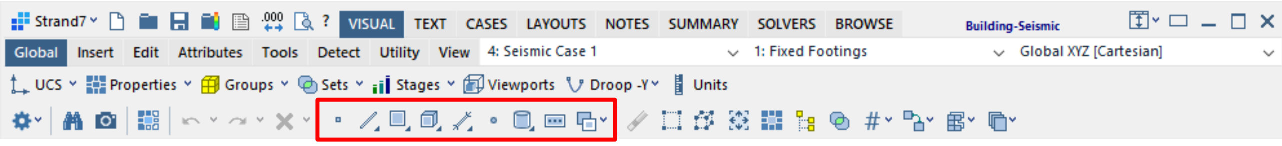
The **Extend** options allow for the selection region to be infinitely extended in specified directions without the need to ensure that a limiting point has been included in the defined region.

The new **Free Cylindrical UCS** complements the previously available *Free Cartesian UCS*. A free cylindrical UCS is defined by clicking three points that lie on the radius of a circle on the cylinder. Additional points clicked will then extend the region in the radial, tangential and/or axial directions of the cylinder.

The **Auto-highlight position** option on the dialog provides a way to see the coordinates of points under the mouse cursor by using the entity inspector without having to press the Shift key. This is independent of the state of the entity selection toggles, and makes it easier to accurately select points to define the region.

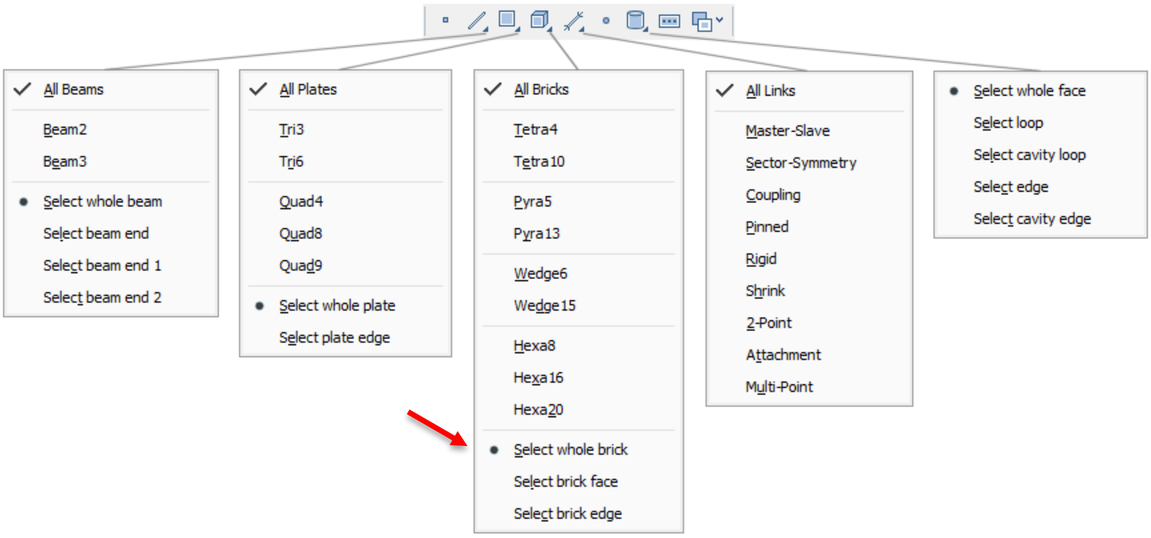


Right-click Entity Select Toggles



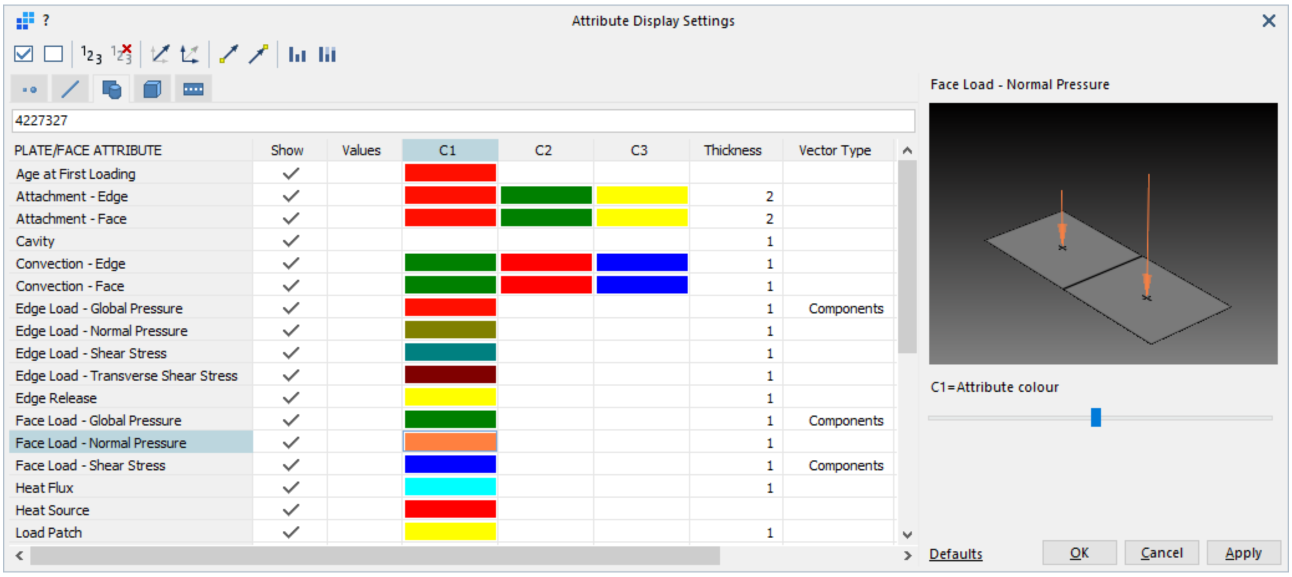
The entity selection toggles on the main toolbar respond to a right-click action similarly to R24. This brings up a popup menu with various options. In addition to the entity type selection functions previously available, it is now also possible to specify the part of an entity to be selected (for example, always select beam end 1, no matter which end of the beam is clicked, or select a plate edge instead of the whole plate, and so on).

For selection operations, Strand7 has always provided context-specific selection functionality. For example, when applying a brick face pressure attribute, the selection mode is automatically set to select brick faces, not the whole brick, as soon as the brick face pressure dialog is displayed. The new options available in R3 provide the possibility of setting the selection mode before the context is automatically established.



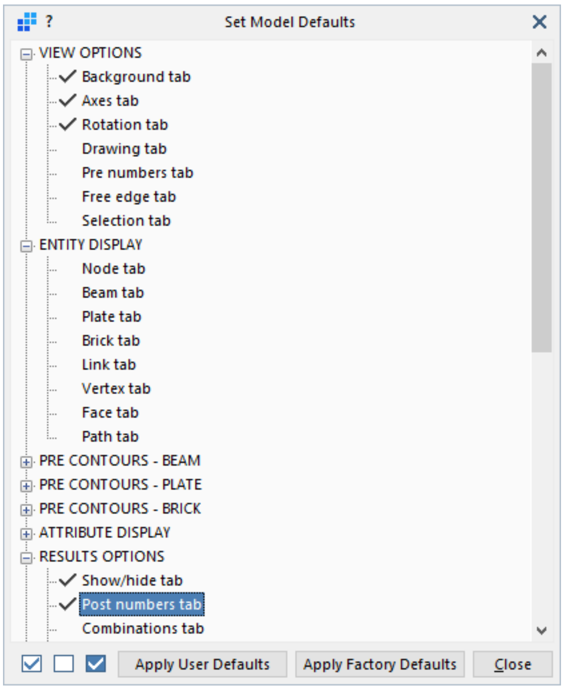
Rendering

R3 offers numerous enhancements to model rendering and customisation compared with previous Strand7 releases. These can be selected via the respective entity and attribute display dialogs.



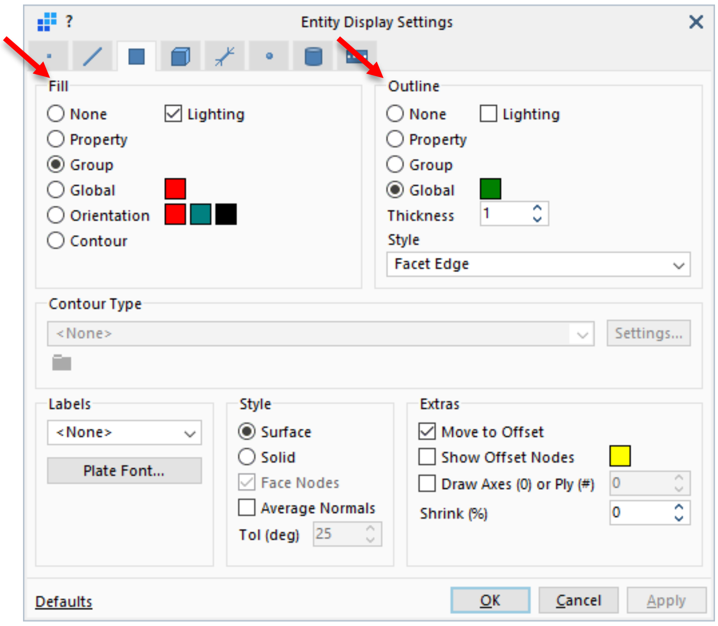
Display Options/Defaults

R3 offers the possibility of configuring most aspects of the model, the display, the entities and the attributes. Each of these is configured in their respective dialogs. The new **Set Model Defaults** option facilitates the setting of default or user-preferences for the entire model; all of these can be set from the one place. Within each main category, it is also possible to selectively set the defaults for sub-categories.



Fill vs Outline

For the rendering of entities (beams, plates, brick and faces) the colours and options used for the fill are independent of the colours and options used for the outlines.



Geometry Faces as Solid

Geometry faces, typically imported from CAD files, can be rendered in shaded solid mode in R3. This greatly improves the visualisation of geometry and the efficiency with which geometry may be edited and prepared for meshing.

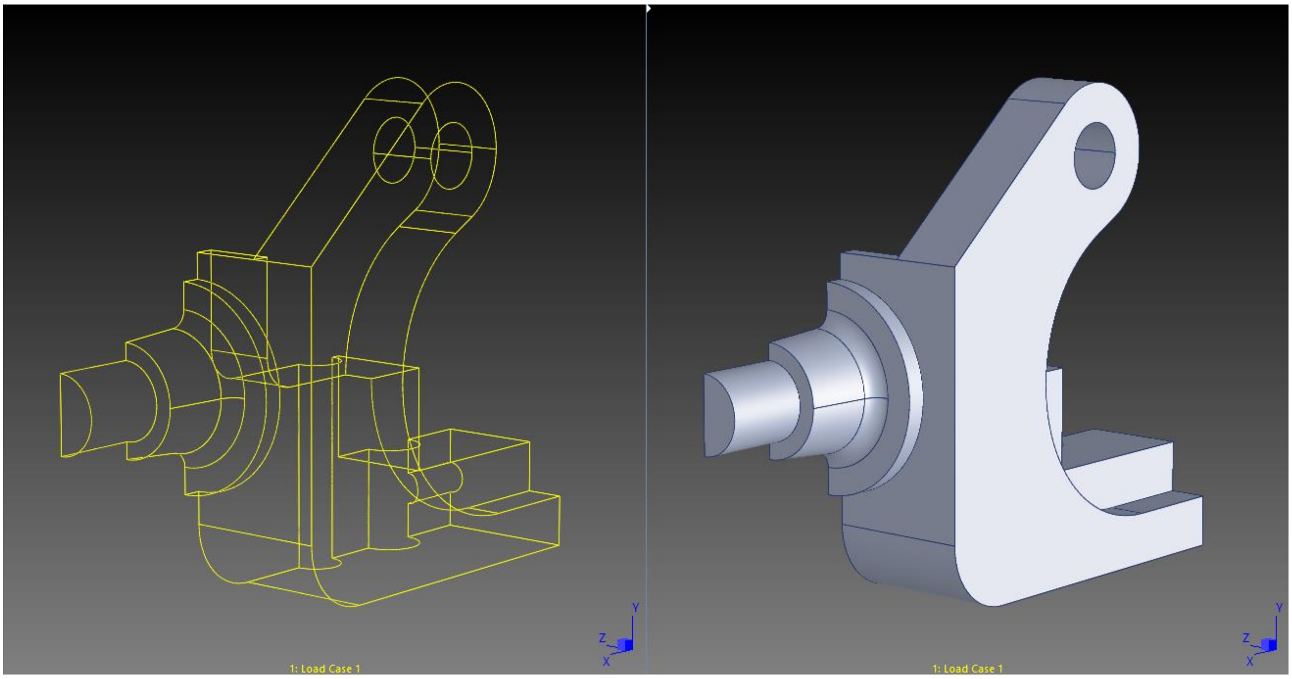
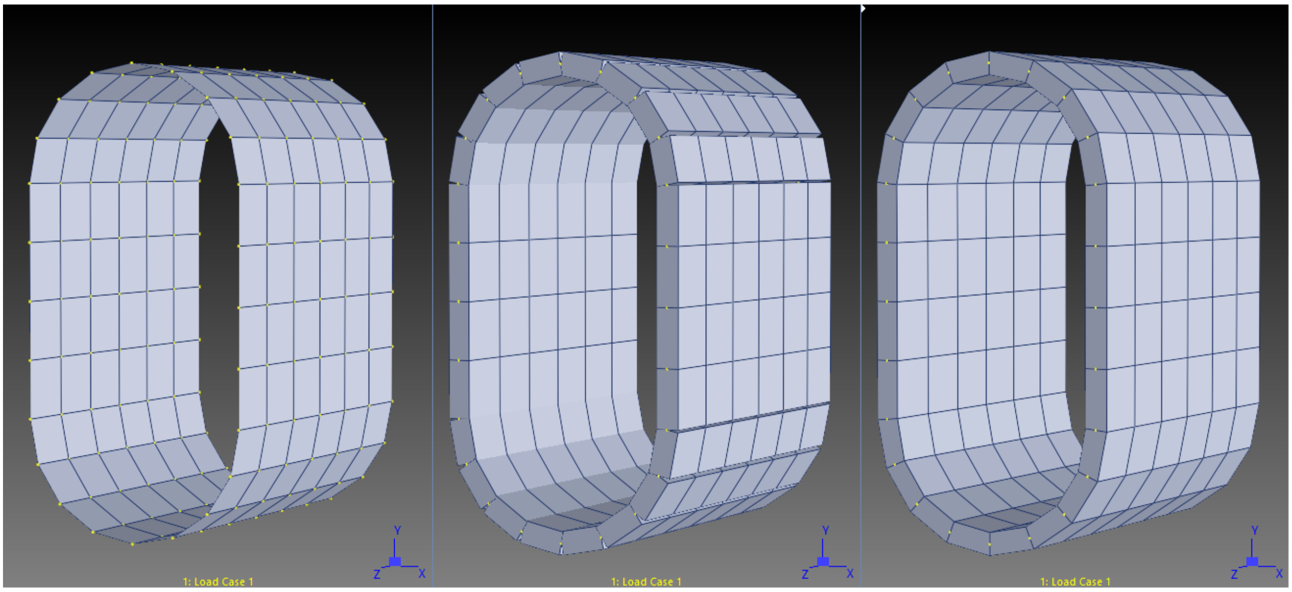


Plate – Average Normals

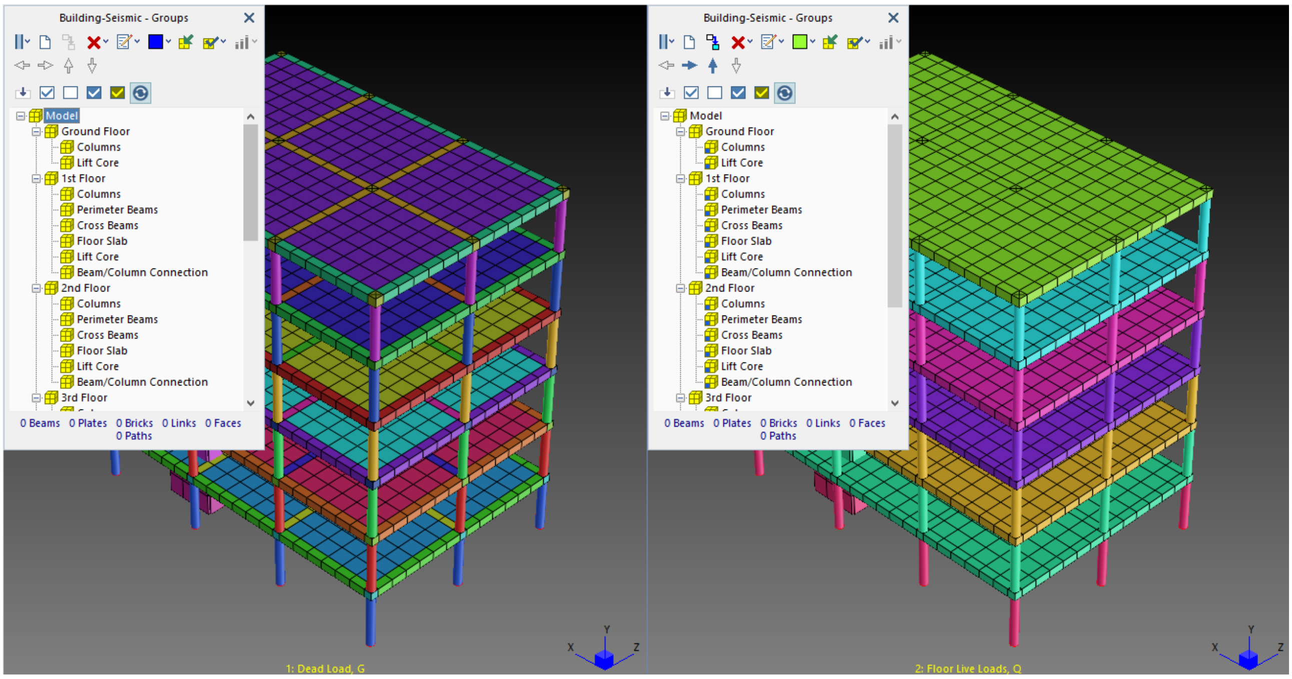
For the rendering of deformed plate elements, an option that averages the normals from adjacent plates is available. This is useful for producing better quality images in the case where the mesh represents a continuous geometry rather than a folded one, especially when plate elements are rendered as solid, and particularly for post-processing of deformed meshes.



Mesh. Mesh with thickness. Mesh with thickness and average normals.

Groups as Parent Colour

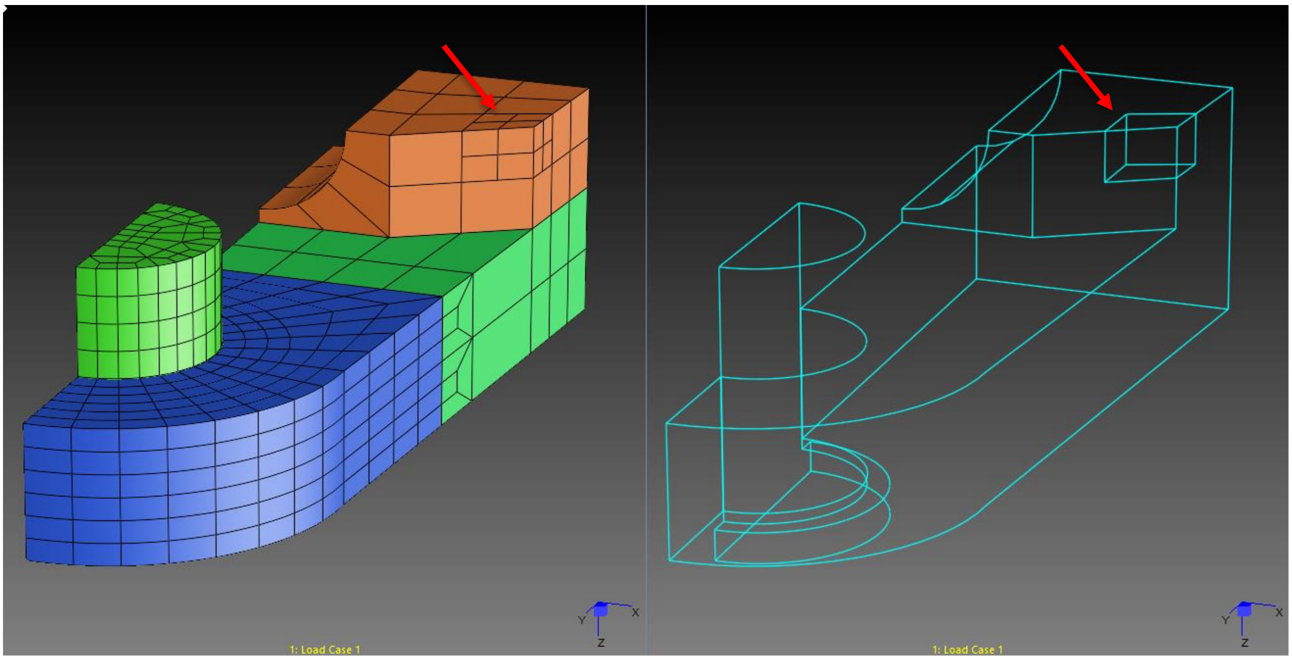
This feature enables a child group to be rendered using the colour of its parent group. Multiple levels can be traversed in setting the parent group colour. The right-click option, **Set parent colour level**, makes this easy to set on the whole group tree in a single operation.



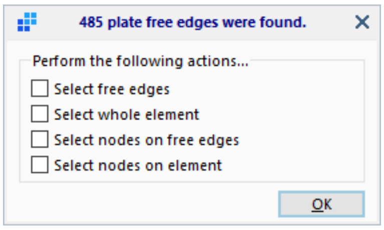
Detect

The previously available detection options, such as show beam and link free ends and show plate free edges, are retained in R3. These are augmented by an option to show beam junctions and another to show brick faces. These are now accessed via the **VISUAL/Detect** tab rather than via the entity toggles toolbar.

The following image shows the brick free face display for a brick model. Unintended internal free faces are easily detected. The number of lines shown can be controlled by a facet angle parameter. This detection feature is equally useful for tetrahedral brick models as it is for hexahedral models.



In addition, every **Detect** request produces a dialog with information about the detection result together with options to make selections based on those results. For example, in response to a plate free edge detection request, the nodes on those free edges could be selected.

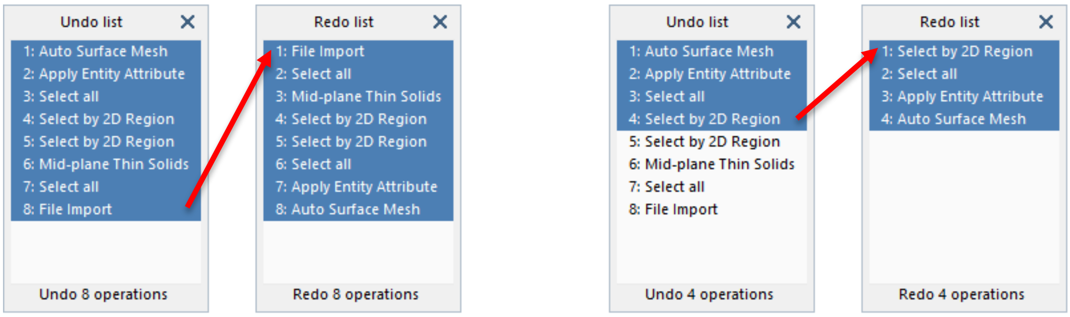


Modelling

Undo/Redo

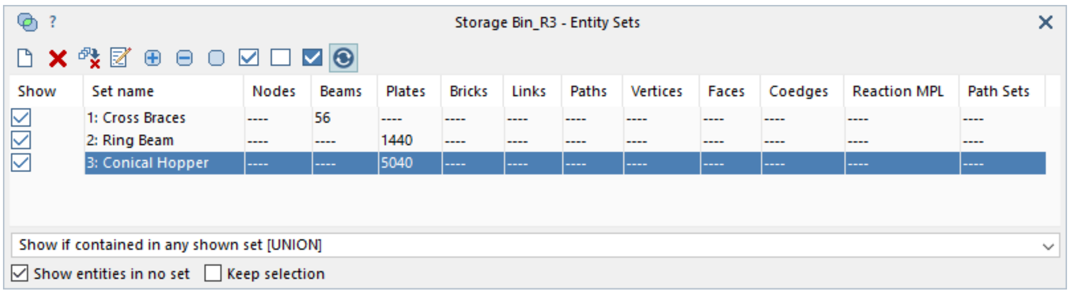
R3 offers greater undo/redo functionality compared with R24. Firstly, many more modelling actions now appear in the undo/redo lists. This includes changes to the group tree and the creation and deletion of load and freedom cases.

The undo and redo lists work in unison in that an undo operation is removed from the undo list and is immediately pushed to the top of the redo list as the potential next redo operation. If this operation is redone, it is removed from the redo list and pushed back onto the top of the undo list. This cycling of undo/redo can be repeated until the model is modified in some other way (i.e., outside of the undo/redo functions). At that point the redo list is cleared and remains empty until the next undo operation is performed. But until then, it is possible to keep undoing and redoing single or multiple operations, moving them from one list to the other and back.



Entity Sets

Entity sets define collections, enabling entities to be grouped together for different purposes including for showing and hiding parts of a model, for filtering results, for selecting, and so on.



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

Entity sets may also be used for selecting the elements that contribute to a Reaction MPL, for the selection of elements to be considered by a load path, and for other purposes.

Entity sets can be given meaningful names, and their position in the list can be changed by moving them using drag-and-drop actions.

All entity types can be contained in an entity set. Additionally, an entity can be contained in multiple sets, or in no set at all.

Entity sets are conceptually similar to groups as both represent a collection of entities. The main differences between entity sets and groups are:

1. entities can be members of any number of entity sets, including a member of no entity set, whereas an entity must be a member of one and only one group;
2. nodes and vertices can be members of entity sets, but they cannot be explicit members of a group;
3. entity sets do not have a hierarchy, whereas groups are hierarchical;
4. entity sets store the selection states of entities as they were when they were first added to the set, whereas groups do not store the selection state of entities in the group.

Load Cases

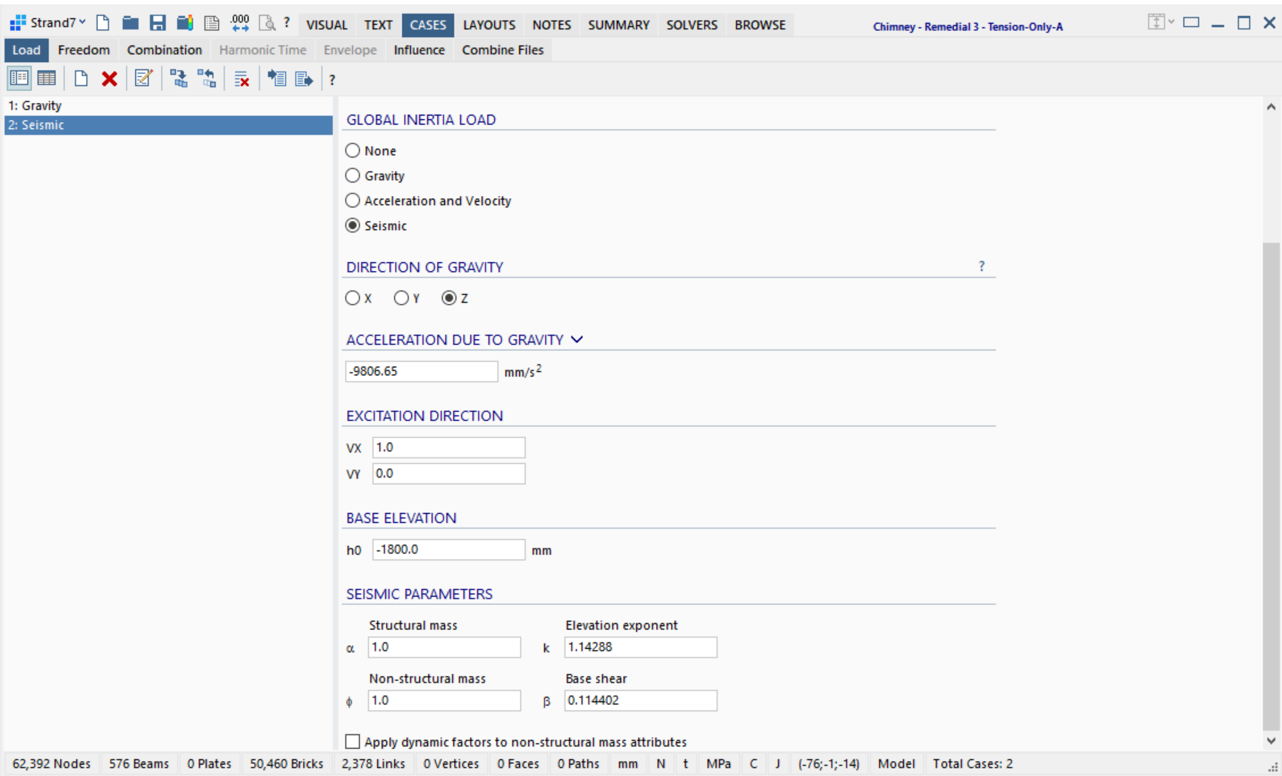
Equivalent Seismic as a General Load Case

Equivalent seismic load cases are integrated together with the other load case types in R3. When opening an R24 file in R3, seismic load cases are automatically converted to the R3 load case type. Non-structural mass referenced from other load cases in the R24 model are inserted into the new load case in R3 to ensure that the same loads are applied.

The distinction between load case types is made based on the **GLOBAL INERTIA LOAD** setting for the load case. This can be one of:

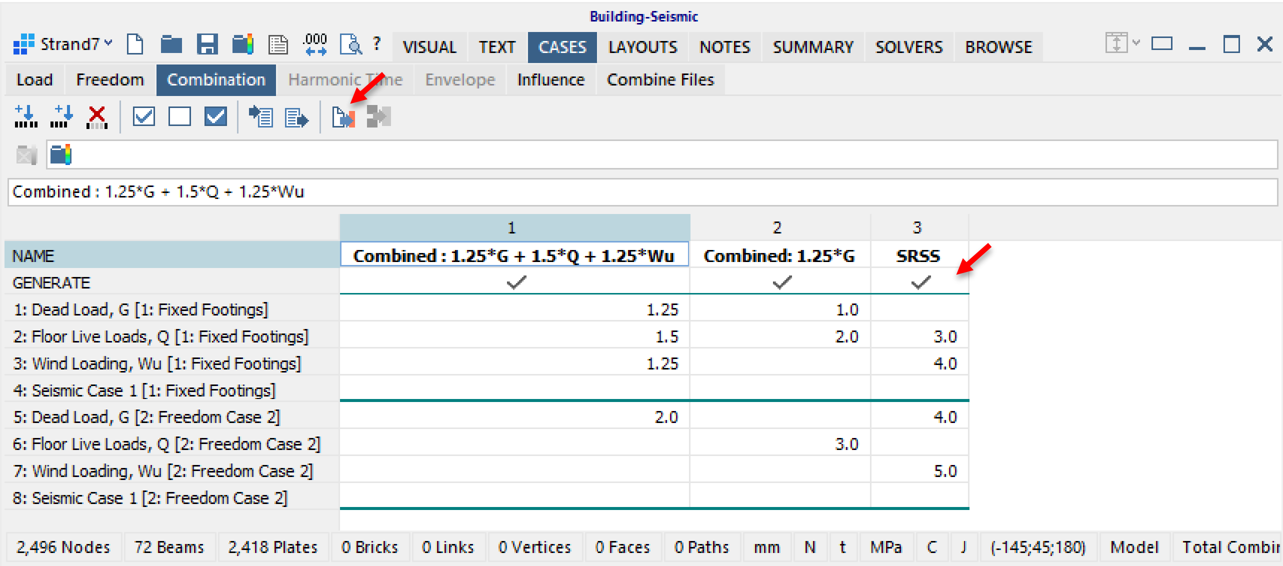
- None (no inertia load is applied).
- Gravity (inertia load is applied in a single direction in one of the global X, Y or Z axes).
- Acceleration and velocity (inertia load is applied by linear and/or angular accelerations in and about the global X, Y and Z axes, and/or by angular velocity about the global X, Y and Z axes).
- Seismic (lateral inertia load is applied based on the assumed vertical distribution of seismic loads).

With this change, seismic load cases are treated as any other load case. Seismic load cases in R3 may therefore include node and element load attributes and non-structural mass (this avoids having to combine the result cases later). Significantly, these load cases can be used in any way that other load cases are used, including, for example, in nonlinear analysis.



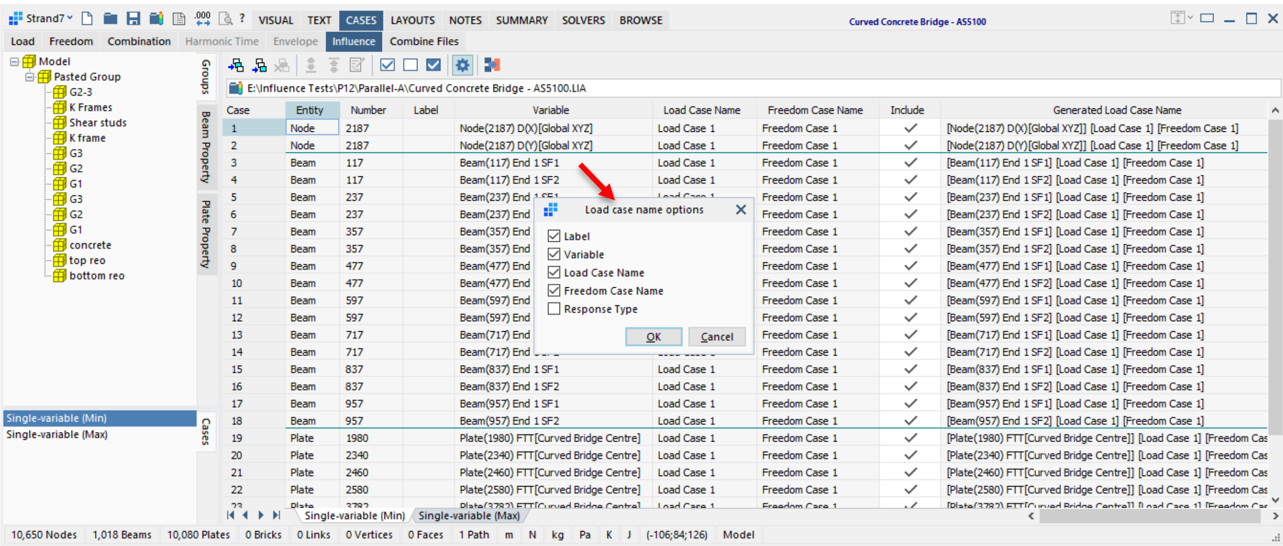
Create Load Cases from Combinations

A new feature in the load case combinations tab, CASES/Combination, offers a function to create a new primary load case from the factors defined in one or more combinations. It is also possible to select the combination cases to be generated so that not all combination cases have to be generated for post-processing.



Influence Case Names

R3 offers greater flexibility for the generation of load case names produced by load influence analysis. In addition to information about the influence variable, a user-specified label may also be included.



TABLES

New Table Types

R3 introduces the following new table types:

- Factor vs Velocity
- Displacement vs Time
- Velocity vs Time
- Temperature vs Time

For the import of tables from a text file, the new bulk import option makes the import of multiple tables more efficient.

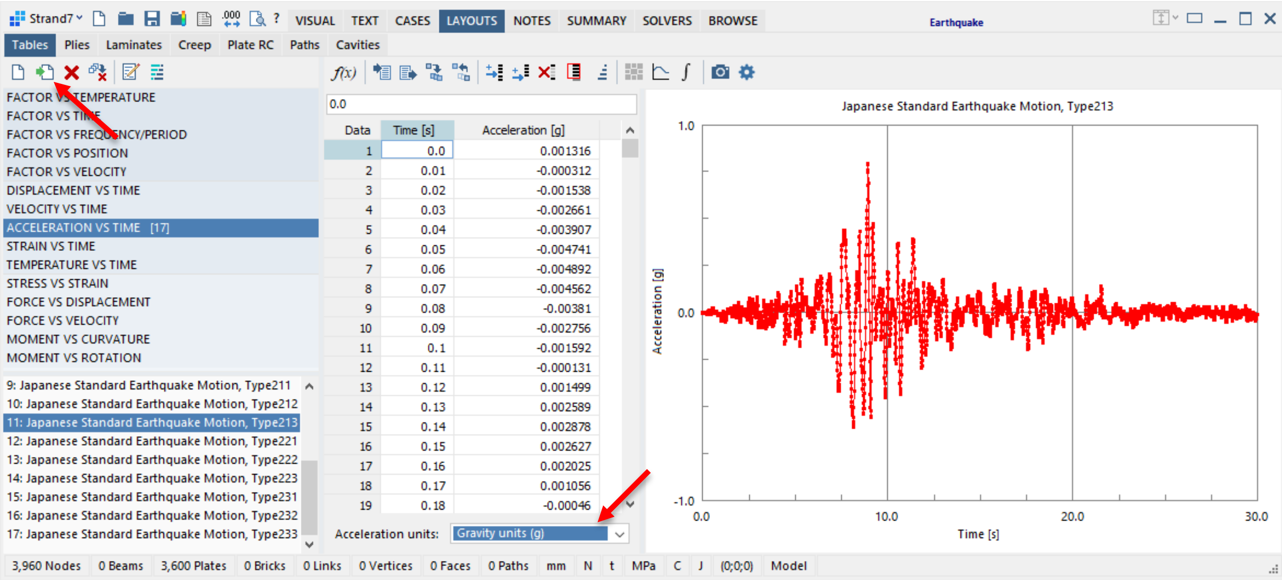
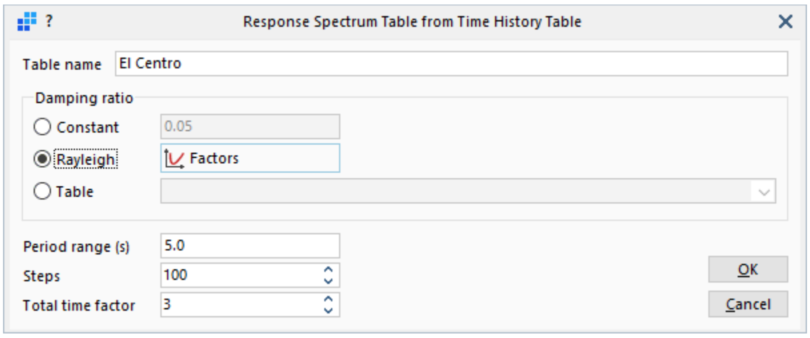


Table Units

In addition to the model units option (e.g., mm/s^2) Acceleration vs Time tables and Factor vs Frequency/Period tables can also be defined in units of *g* (acceleration due to gravity).

Convert Time History to Response Spectrum

In addition to the previously available constant damping ratio option, the damping for the conversion of acceleration time history tables to response spectrum tables in R3 can be specified as Rayleigh or via a Factor vs Frequency table.

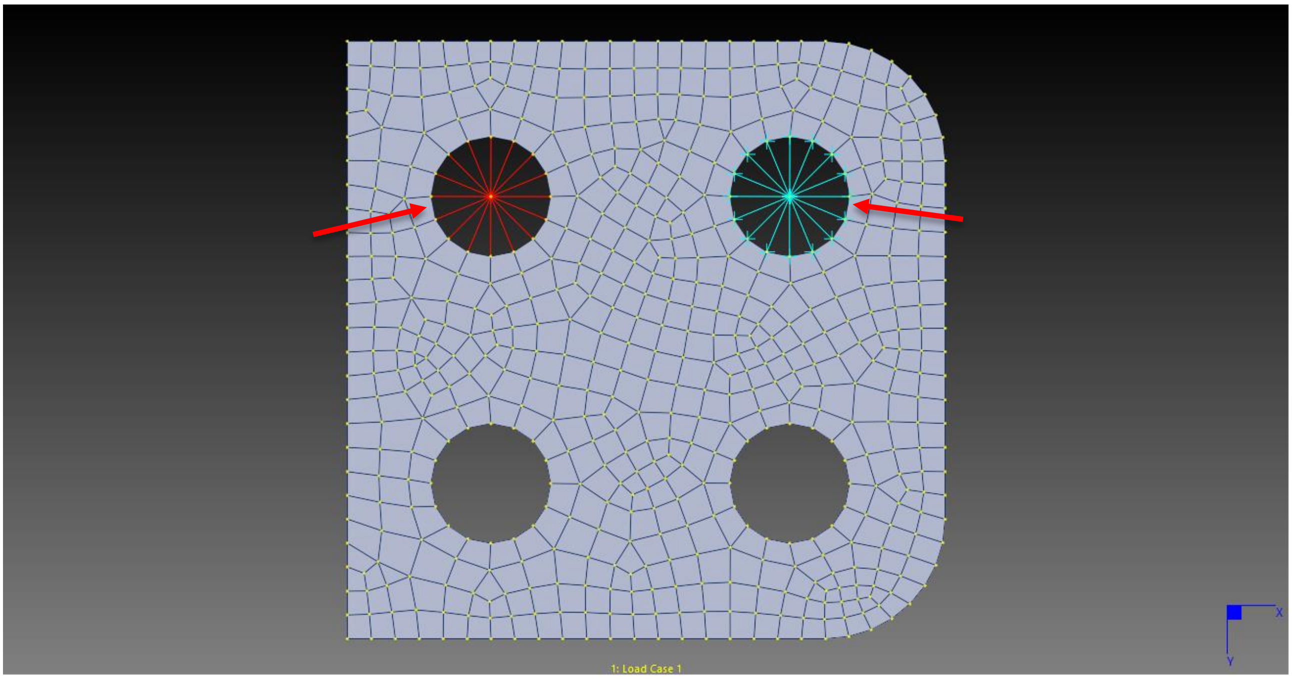


ELEMENTS

The R3 element library has been enhanced and improved compared with R24, and the majority of the elements will produce more accurate results compared with previous Strand7 releases. In addition, the efficiency of element routines is also generally improved compared with R24 (for example, the assembly of 370000 Quad8 shell elements might take 200 s in R24, but only 60 s in R3).

Multi Point Links (MPL)

In addition to the previously available two-node Master-Slave, Rigid and Pinned links, multi-point versions of these links are available in R3. These are useful because they replace clusters of multiple links (e.g., the 16 individual rigid links on the right-side hole below) with an equivalent single link (e.g., the single rigid MPL on the left-side hole below).

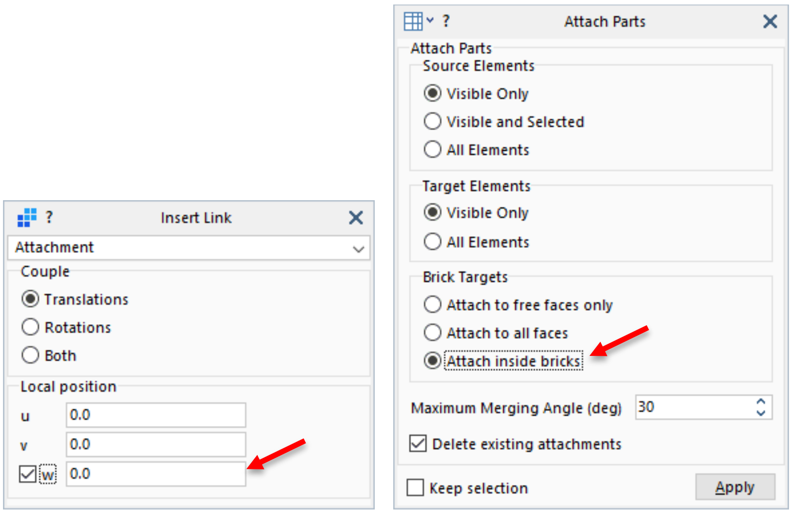


A tool is available to convert an MPL into the equivalent 2-node link cluster.

Attachment Link

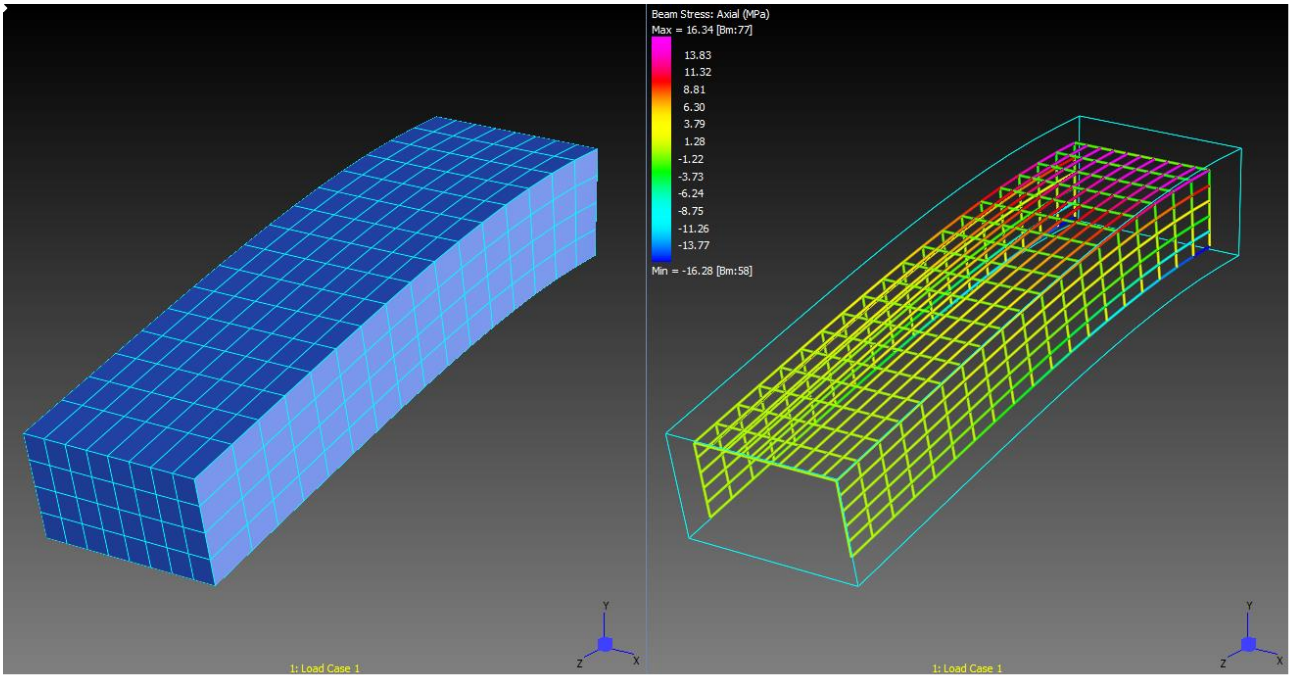
An attachment link connects a node to an element (i.e., along the length of a beam, on the surface of a plate, on the surface of a brick face). In R3, attachment links can also attach nodes to the *inside* of a brick element. This is achieved by defining a new *w* position parameter in addition to the previous *u* and *v* position parameters (the three parameters uniquely define a point in space inside a brick element).

Although attachment links can be inserted directly into a model, in most cases they are inserted indirectly via VISUAL/Tools/Mesh/Attach Parts, which uses beam, plate and brick attachment attributes to detect attachment areas. In R3, this tool offers the option to attach to the inside of brick elements for enclosed attachment points.



For example, this feature offers the possibility of easily connecting beam elements representing steel reinforcement to the inside of a brick mesh representing concrete, without the need for a compatible mesh between the beam and the brick elements.

The figure below shows a cantilever represented by a mesh of brick elements. Inside the brick mesh is a steel reinforcement arrangement represented by beam elements that do not connect to the nodes on the brick elements. The connection between beam ends and brick elements is attained by the use of internal attachment links. The deformation of the beam elements follows the brick mesh.



Another enhancement to the attachment link in R3 is that it considers the distance between the attached node and the target point on the element to which the node is attached. A single attachment link is equivalent to a rigid link connected from the node to the target element, not a master-slave link directly connecting the node to the target element. It is therefore not necessary to use pairs of rigid and attachment links in R3 as the attachment link automatically does the work of an implicit rigid link.

Reaction MPL

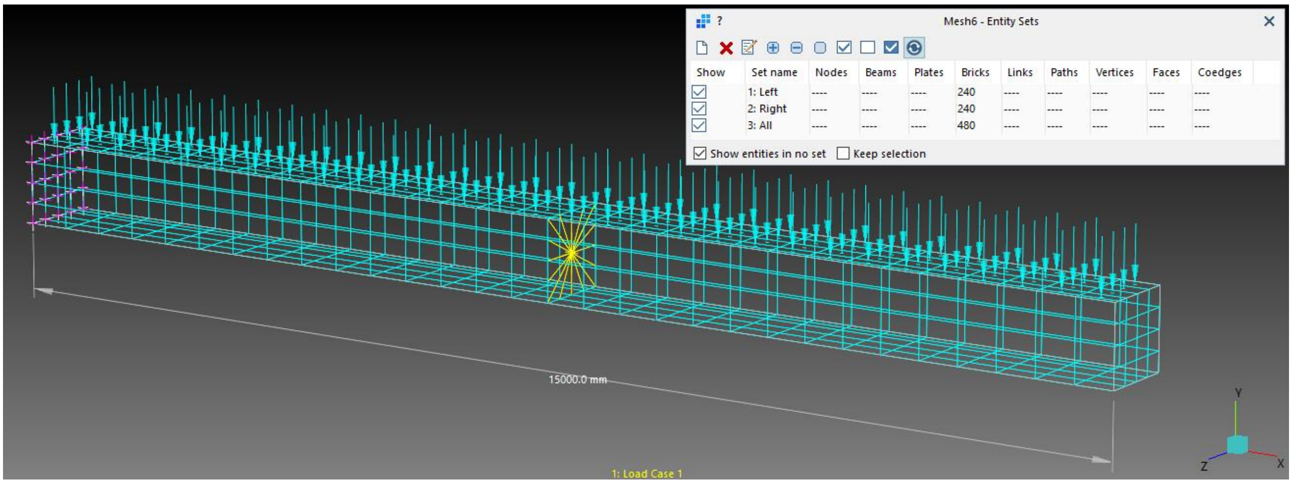
The new Reaction MPL is a special type of link that does not enforce any constraint relationship between nodes or degrees of freedom. Instead, the link is used to calculate force and moment resultants anywhere in the model. It sums the element nodal forces and moments (or flux for heat transfer analysis) at selected nodes of selected elements. This enables the extraction of results such as in-plane shear in a lift core at each floor of a building, the bending moment diagram of the lift core in a building, and the summation of flux/forces/moments about any point in space, to name a few.

To define the link, the nodes to be summed need to be selected. The elements that are to contribute to those nodes must also be selected. Both elements and nodes need to be selected in order to find non-trivial internal forces between components or parts of a model.

The nodes are selected explicitly by attaching them to the MPL itself, similarly to the way other links are defined. The elements are selected by associating an entity set containing the elements with the link.

This is illustrated in the following example.

A cantilever is modelled with brick elements. The cantilever carries a uniformly distributed pressure on the top surface. At the middle of the span, a Reaction MPL is attached to all nodes on the cross section. Three entity sets are defined. The brick elements to the left of the link are assigned to the *Left* entity set. The brick elements to the right of the link are assigned to the *Right* entity set. One other entity set that includes all of the bricks in the mesh, *All*, is also defined.



A linear static analysis is executed and the forces and moments at the Reaction MPL are extracted using **Peek**. Three cases are considered; in each case one of the entity sets is assigned to the MPL before extracting the results. The MPL forces are tabulated below.

Entity Set	FX	FY	FZ	MX	MY	MZ
Left	0	-7.5000E+06	0	0	0	-2.8100E+10
Right	0	7.5000E+06	0	0	0	2.8100E+10
All	0	0	0	0	0	0

The link sums the nodal force contributions from the elements in the associated entity set. When all elements at a node are summed, zero reactions are produced as expected. The difference between the *Left* and *Right* entity sets is just the sign of the results. When these two results are added, the total reactions again sum to zero.

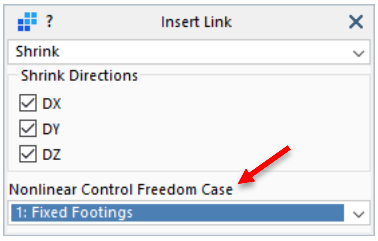
With this functionality, it is possible to extract free body diagrams of any part of a structure, or internal forces across any part or component of a structure. The sign of the result can be obtained as required by defining the set to contain elements on one side or the other side of the link.

The link may be used for equilibrated results (e.g., linear static) and for non-equilibrated results produced by spectral response SRSS and CQC solutions, and for combinations of these with linear static results.

The links may also be used with envelope result cases to obtain the envelope of each summation.

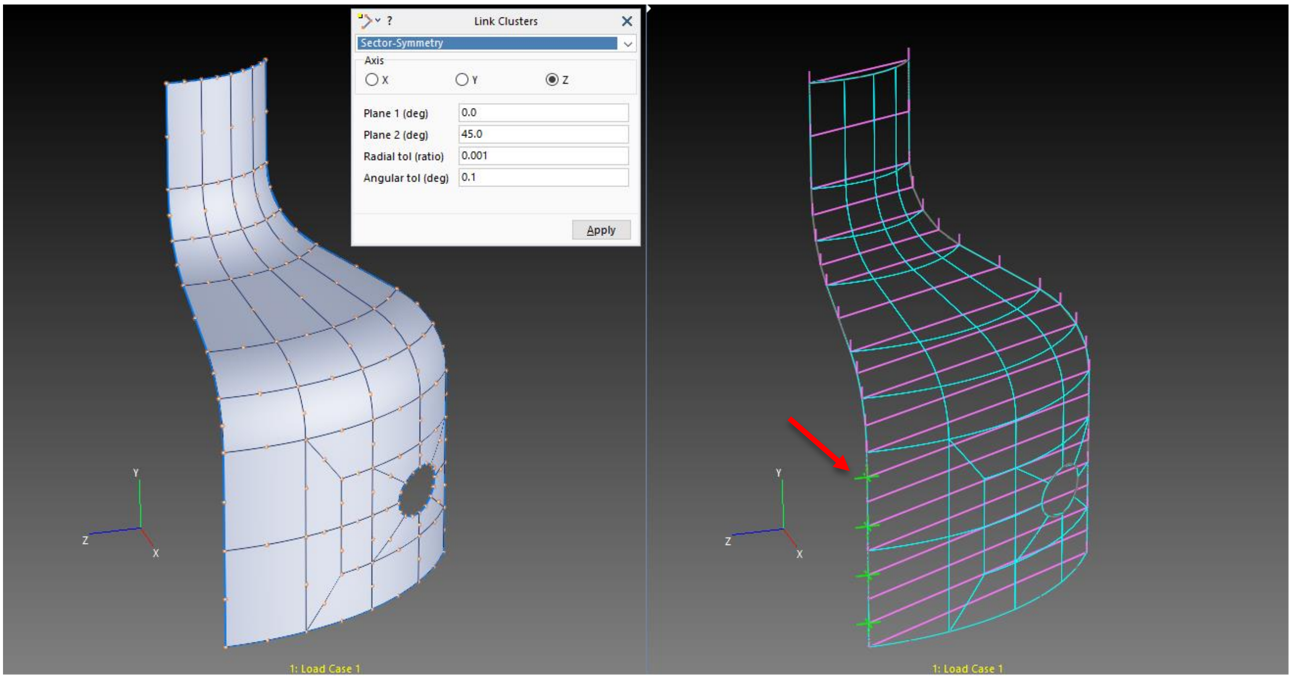
Links – Nonlinear Control Freedom Case

For links with constant terms (i.e., Shrink, 2-Point and User-defined MPL), R3 offers the option of associating a freedom case with the link to enable the link's constant term to be controlled in nonlinear analysis. This facilitates the scaled application of the constant term, such as to progressively increase the amount of shrink applied by a shrink link on an increment-by-increment basis.



Automatic Sector-Symmetry Links

The function **VISUAL/Insert/Multi-element/Link Clusters** offers a range of options for the automatic insertion of link clusters. For the **Sector-Symmetry** option, pairs of nodes with matching radius and axial coordinates should ideally be introduced in the mesh before inserting the sector-symmetry links. For meshes where matching nodes cannot be found, the R3 tool will automatically insert interpolating attachment links as required.



Beam Section Properties and Shear Stress

Shear stress on beam cross sections due to shear forces and torque are calculated for any cross section shape in R3. The calculation is a numerical one whereby the cross section is discretised and a 2D potential flow solution is pre-calculated using the finite element method to produce coefficients that relate shear forces and torque to shear stress at pre-defined positions on the cross section. Six coefficients are calculated at each point to define shear stress on the plane of the section as follows:

$$\tau_{31} = F_1\varphi_1 + F_2\varphi_2 + T\varphi_3$$

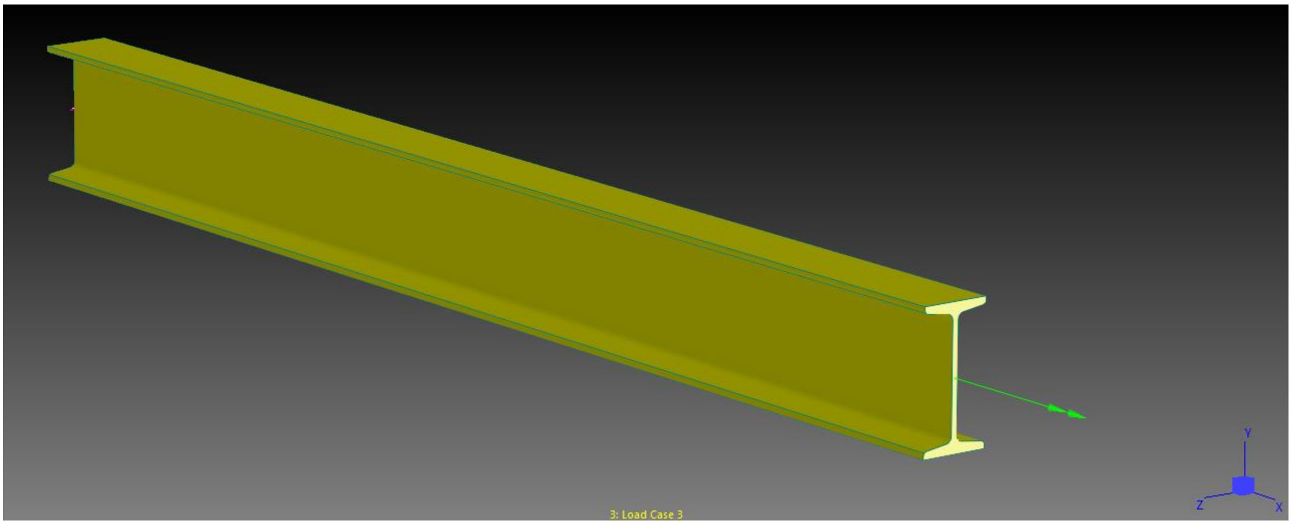
$$\tau_{32} = F_2\varphi_4 + F_1\varphi_5 + T\varphi_6$$

Here F_1 and F_2 are the shear forces in the principal directions of the cross section, T is the torque about the beam axis, and τ_{31} and τ_{32} are the shear stresses in the plane of the cross section. These stresses, together with the fibre stress (normal to the cross section), are used to calculate a range of new stresses on beams, including:

- Shear Stress due to Shear Force 1
- Shear Stress due to Shear Force 2
- Shear Stress due to Torque
- Shear Stress on Plane 1
- Shear Stress on Plane 2
- Shear Stress Magnitude Plane 1-2
- Minimum Principal Stress
- Maximum Principal Stress
- von Mises Stress
- Tresca Stress

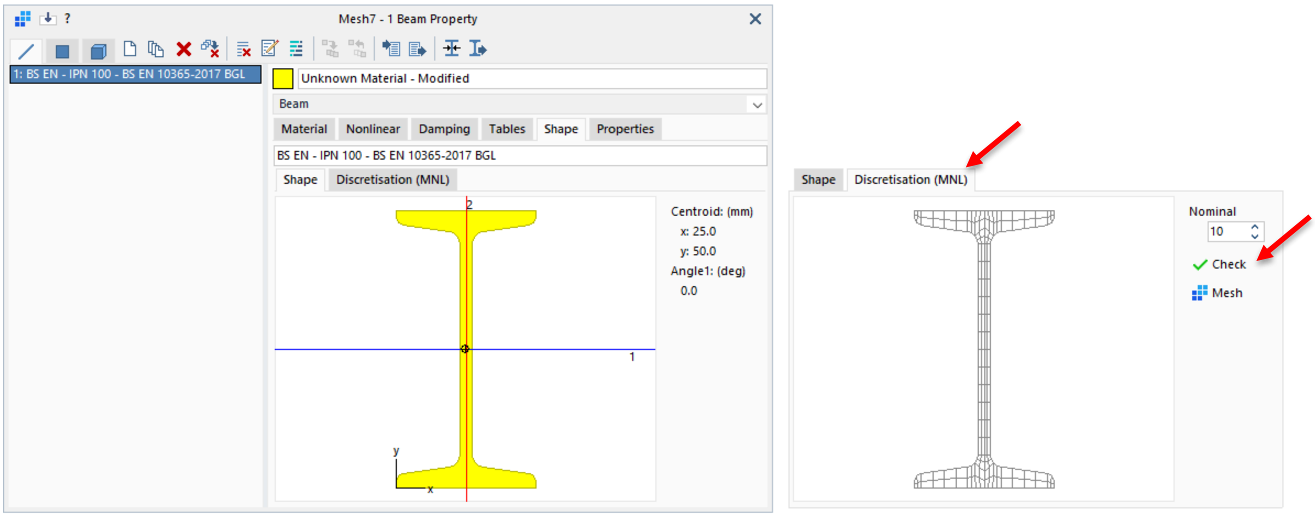
Some of these results are illustrated in the following example.

A cantilever beam is modelled with the cross section assigned from the R3 Beam Geometry Library (BGL - see below).

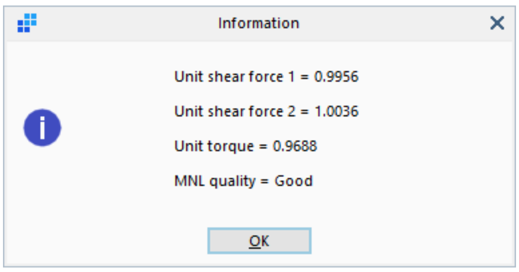


Three load cases are considered:

1. Shear force in the major axis direction.
2. Shear force in the minor axis direction.
3. Torque.



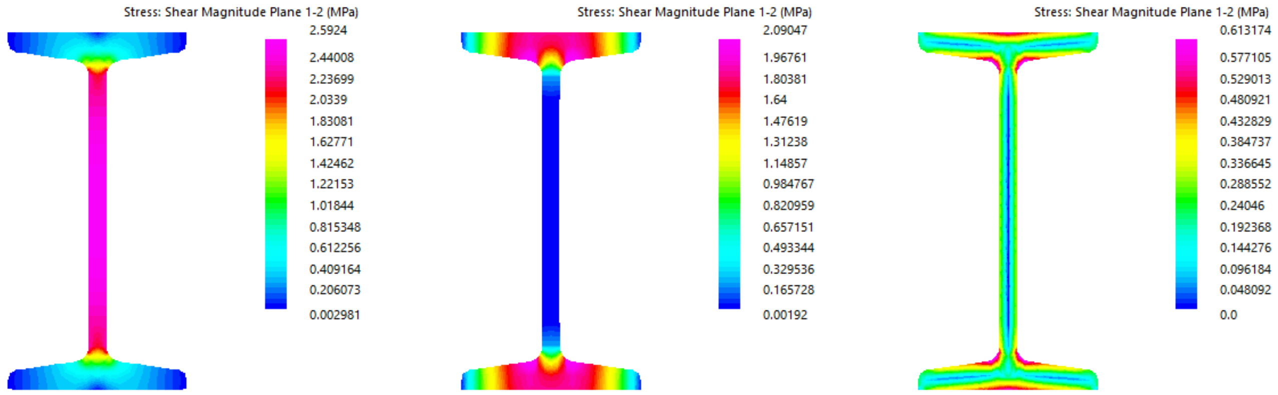
As the coefficients can only be calculated at a pre-defined number of points on the section, a balance needs to be found between accuracy of results and calculation time. This can depend on the section shape and the type of load applied. The **Shape/Discretisation** tab of the beam property dialog provides a tool to **Check** the quality of the mesh and the shear stress integration for different levels of discretisation.



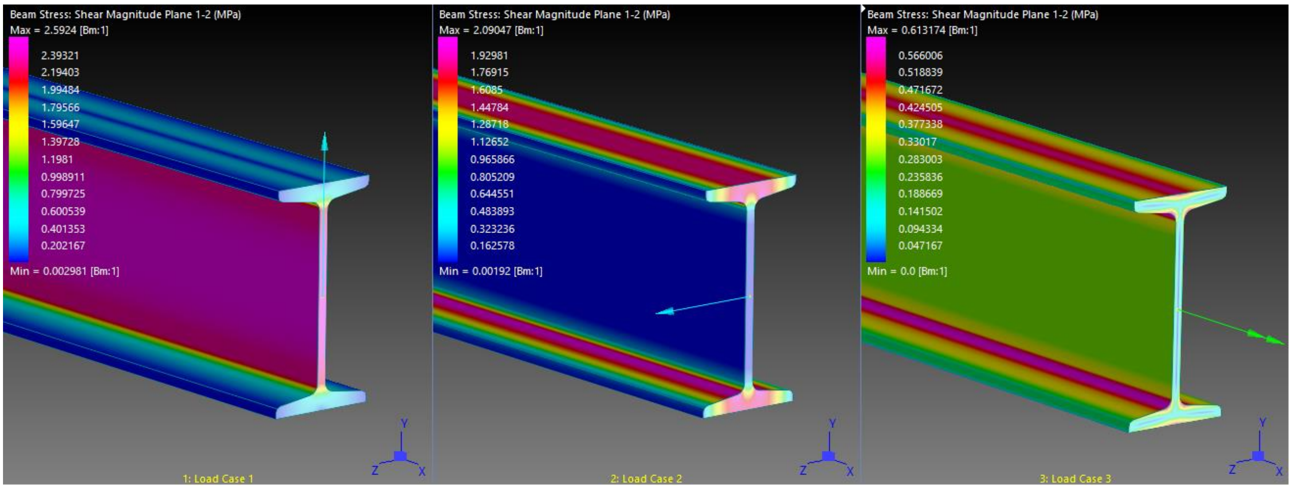
To produce the three quality measures presented on the dialog, the following procedure is employed for the selected discretisation:

- A unit shear force is applied in each of the principal 1 and 2 directions, the shear stresses are calculated at each point on the cross section, and the calculated shear stresses are integrated over the section to produce two forces, one in each of the principal directions. For an exact result, these integrals should both be equal to 1.
- A similar procedure is used to integrate the shear stresses due to torque, which are then integrated to return the torque integral.

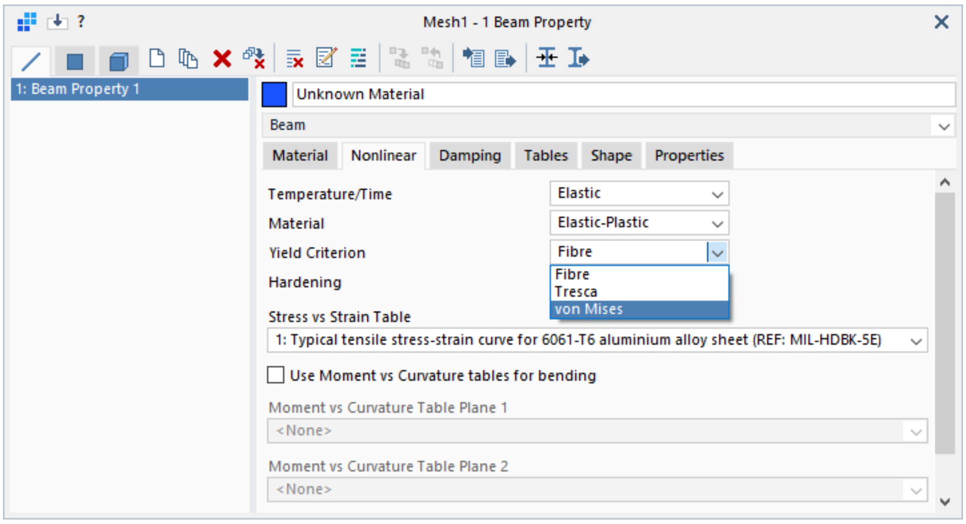
A good quality mesh will typically return values between 0.95 and 1.05 for each of the three components. The following image shows the shear stress magnitude calculated on the cross section for each of the three load cases.



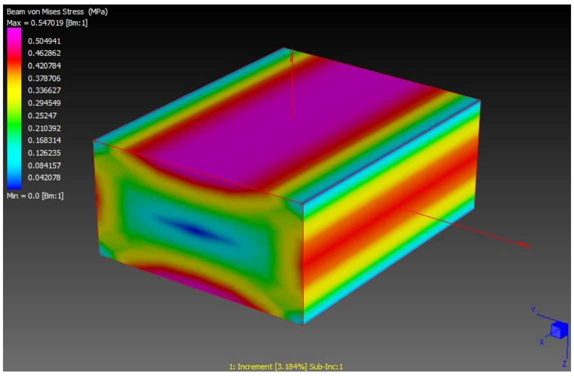
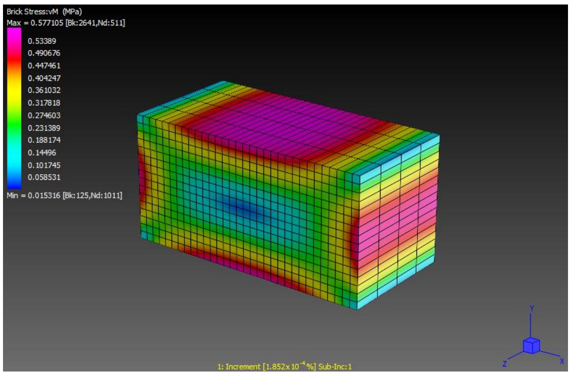
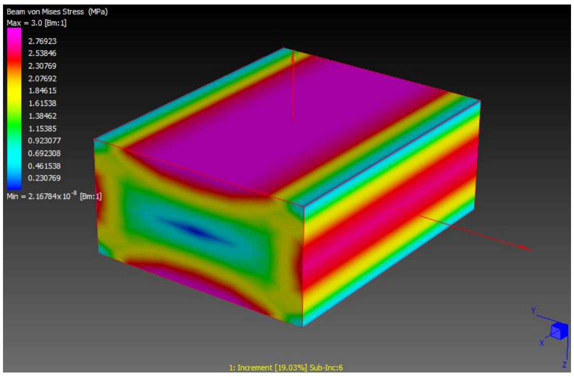
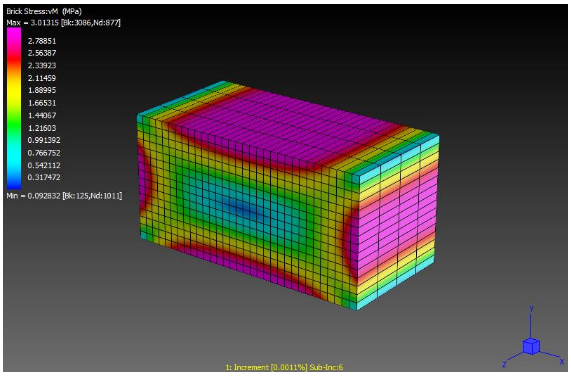
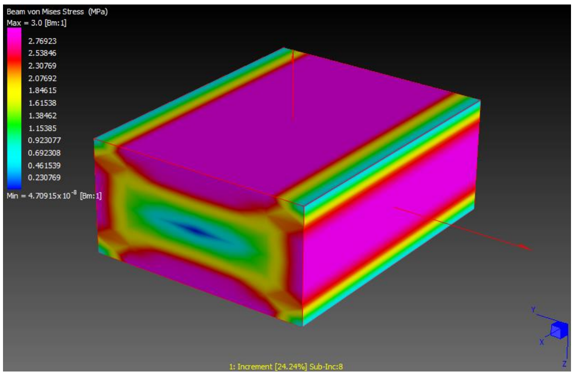
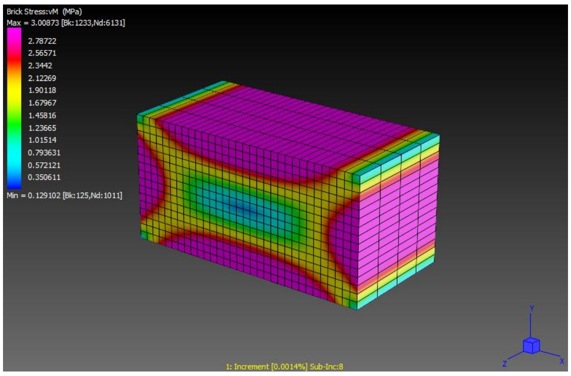
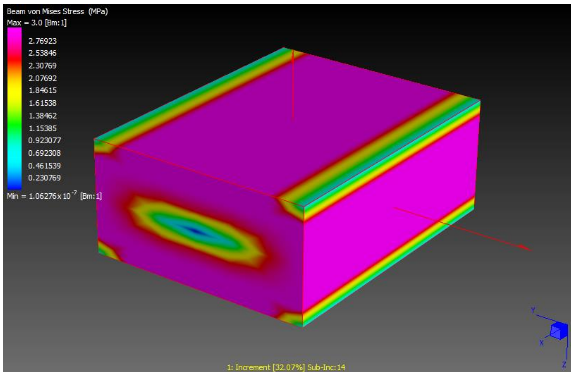
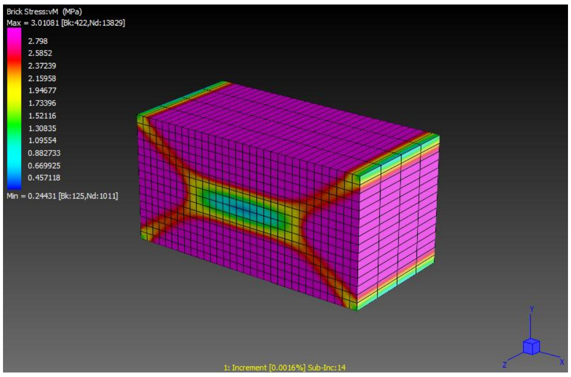
Shear 1 Shear 2 Torque



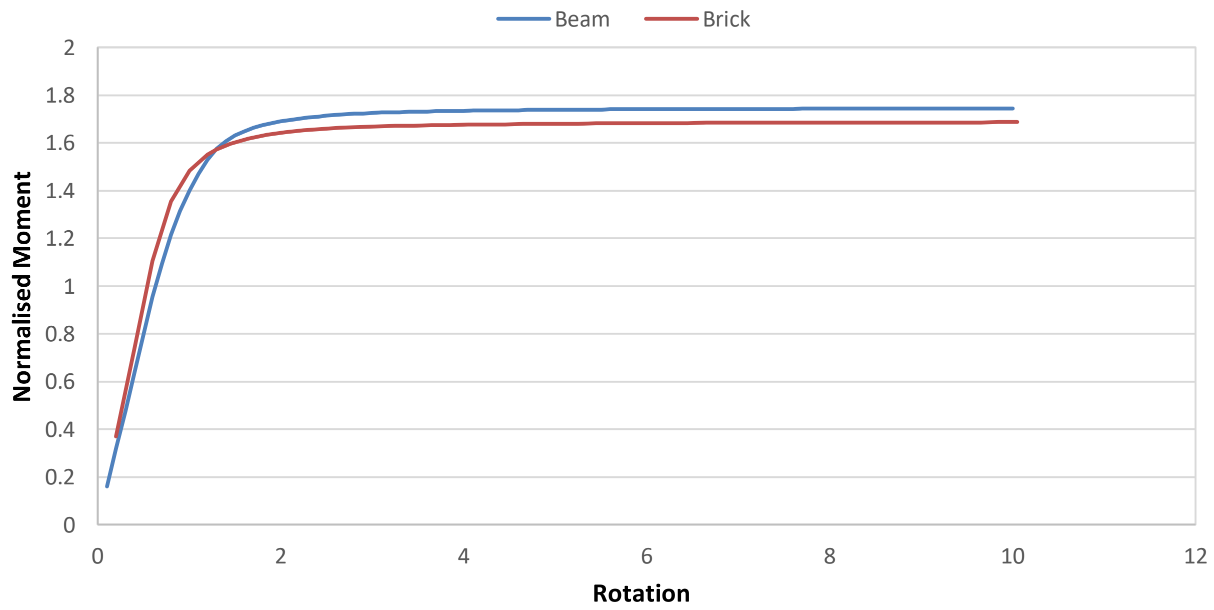
With the availability of shear stress on any cross section, R3 extends the previously available **Fibre Stress** yield criterion to include **Tresca** and **von Mises** criteria when a stress-strain table is assigned for material nonlinear analysis.



The use of the von Mises criterion in a material nonlinear analysis is demonstrated via the simple example below whereby the results of a single beam element with a rectangular solid cross section under torque, applied as an enforced rotation, are compared with an equivalent brick element model. The material is assumed to be elastic-perfectly-plastic.

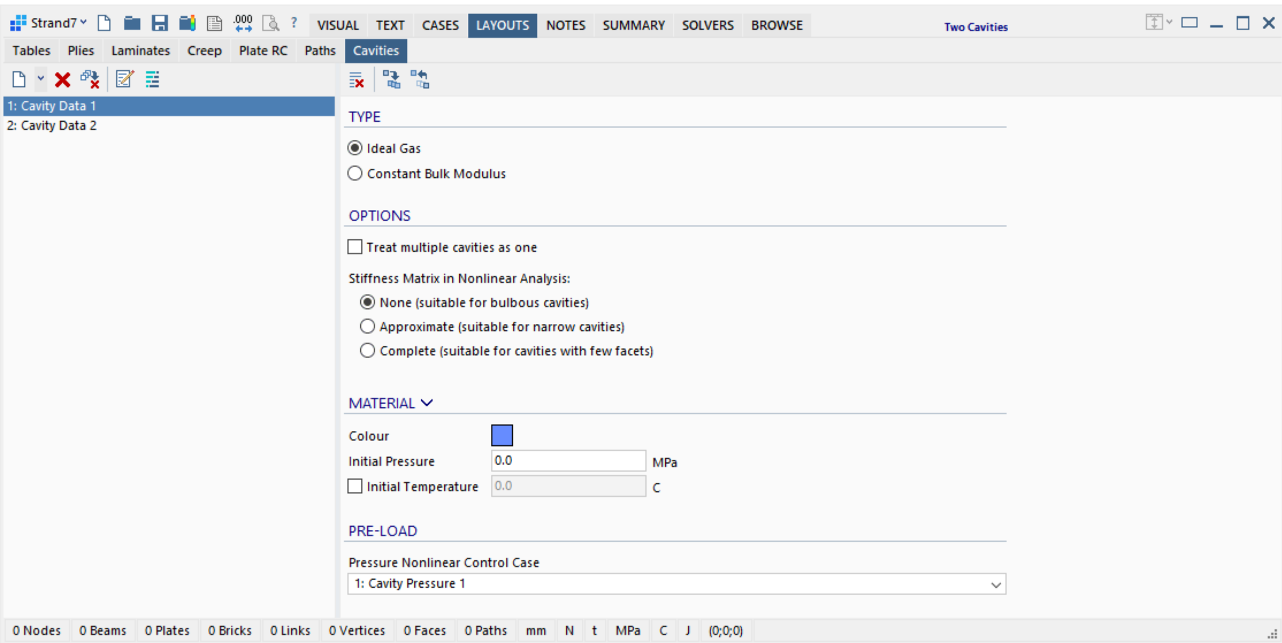
Twist deg	Beam model von Mises Stress	Brick model von Mises Stress
0.1		
0.6		
0.8		
1.4		

The normalised moment vs rotation result is compared in the following graph.



Cavity Attribute

The R3 cavity attribute fills a closed cavity (i.e., volume) defined by plate elements and/or brick element faces, with a fluid – e.g., the inside surface of a balloon, the inside surfaces of an IGU (Insulated Glass Unit), and so on.

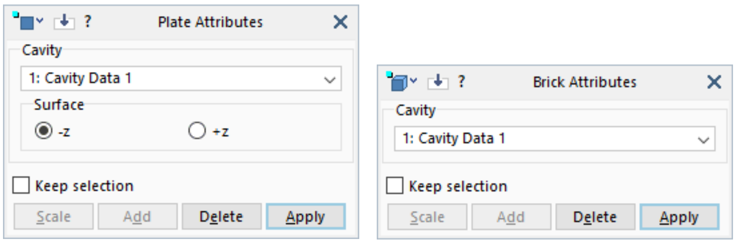


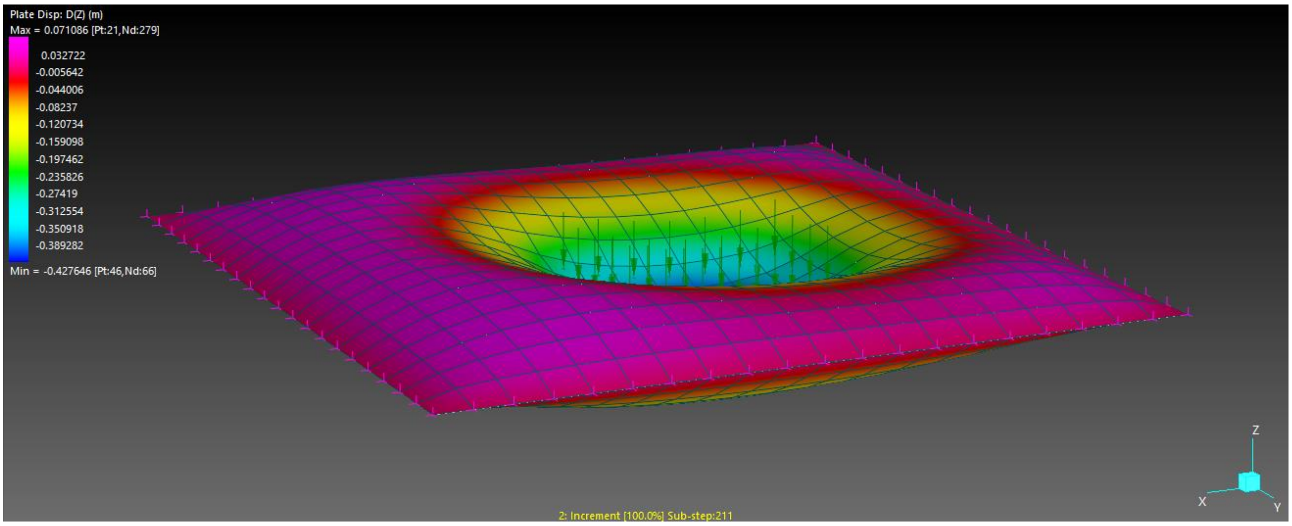
Two types of cavities are available and these are defined under LAYOUTS/Cavities:

1. **Ideal Gas** cavity, which enforces the ideal gas relationship between pressure, volume and (optionally) temperature (e.g., a helium-filled balloon).
2. **Constant Bulk Modulus** cavity, which enforces a constant relationship between pressure and volumetric strain by the definition of a constant bulk modulus (e.g., a water-filled balloon).

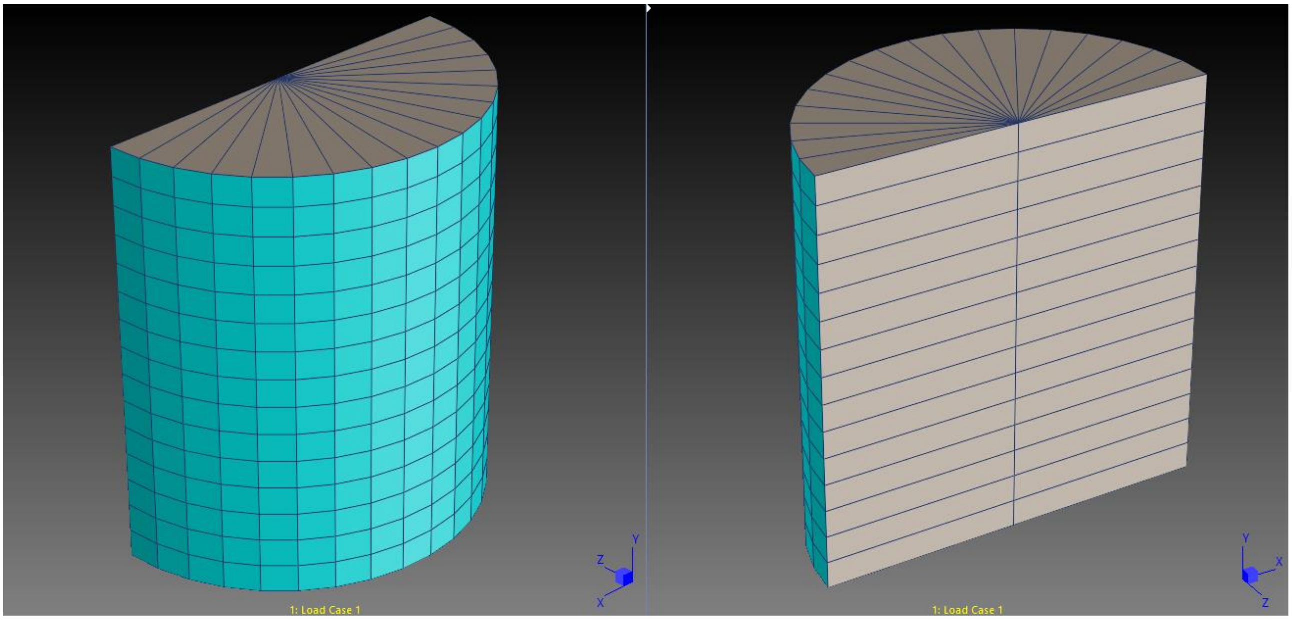
For both cavity types, the initial volume is defined by the cavity geometry (i.e., the user-defined mesh).

Cavity layouts are assigned to plate and brick surfaces as attributes.

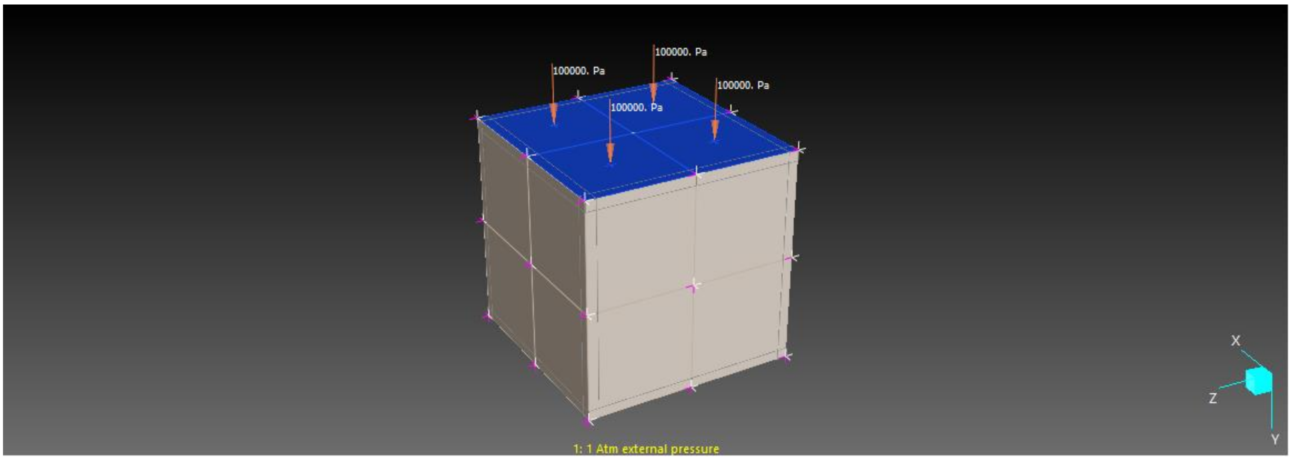




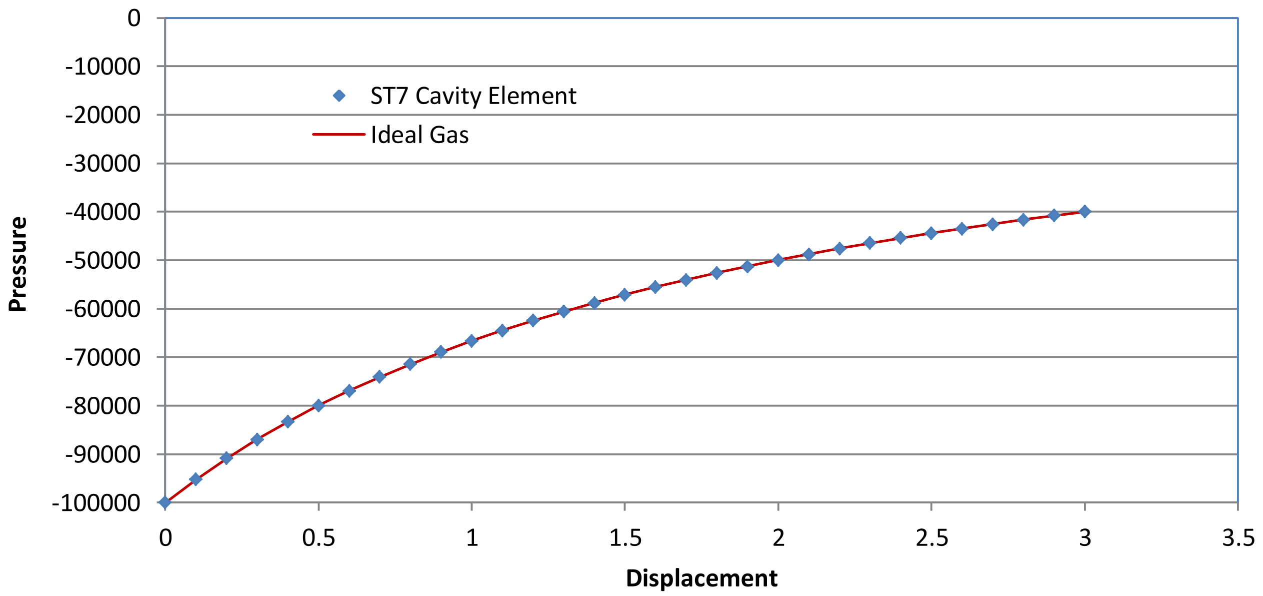
A valid cavity is one where the cavity volume is represented by a watertight (manifold) surface with all elements on the surface assigned the same cavity layout. For modelling symmetric halves or quarters of volumes, the symmetry planes need to be closed such that a closed cavity is defined. For example, in the half cylinder model below, plate elements of property type 0 are used to close the cavity. Those elements don't participate in the analysis; they are used simply to define the volume. Therefore, mesh refinement is not necessary for these elements; they just need to be compatible with the real mesh.



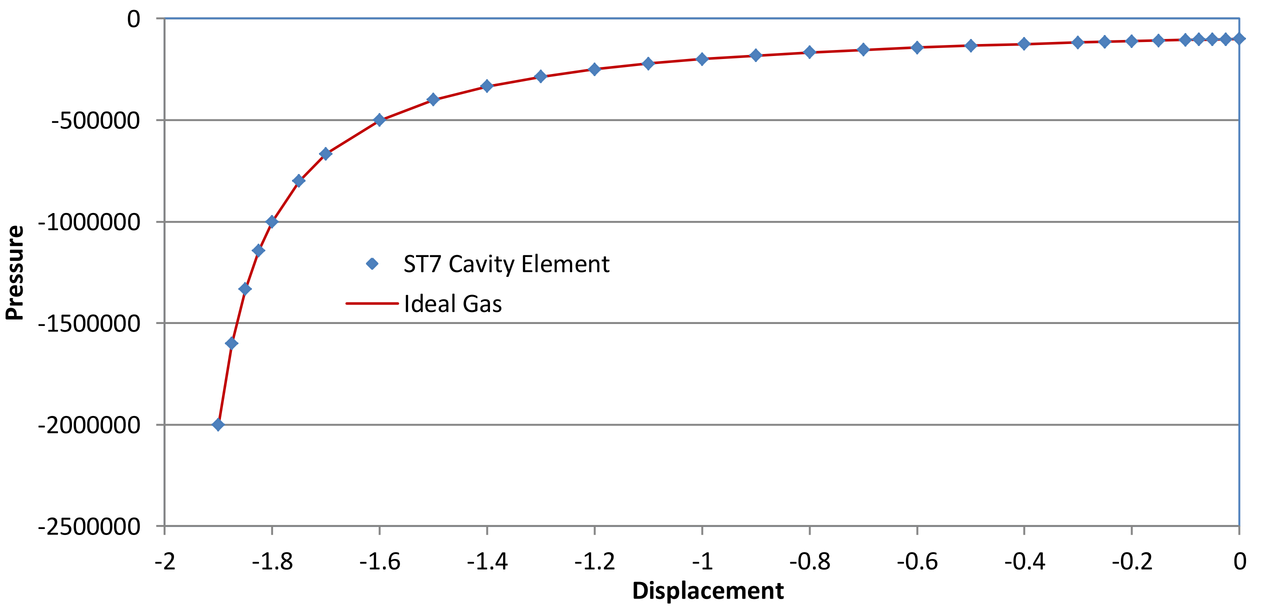
The following illustrates the simple example of an ideal gas piston-cylinder arrangement.



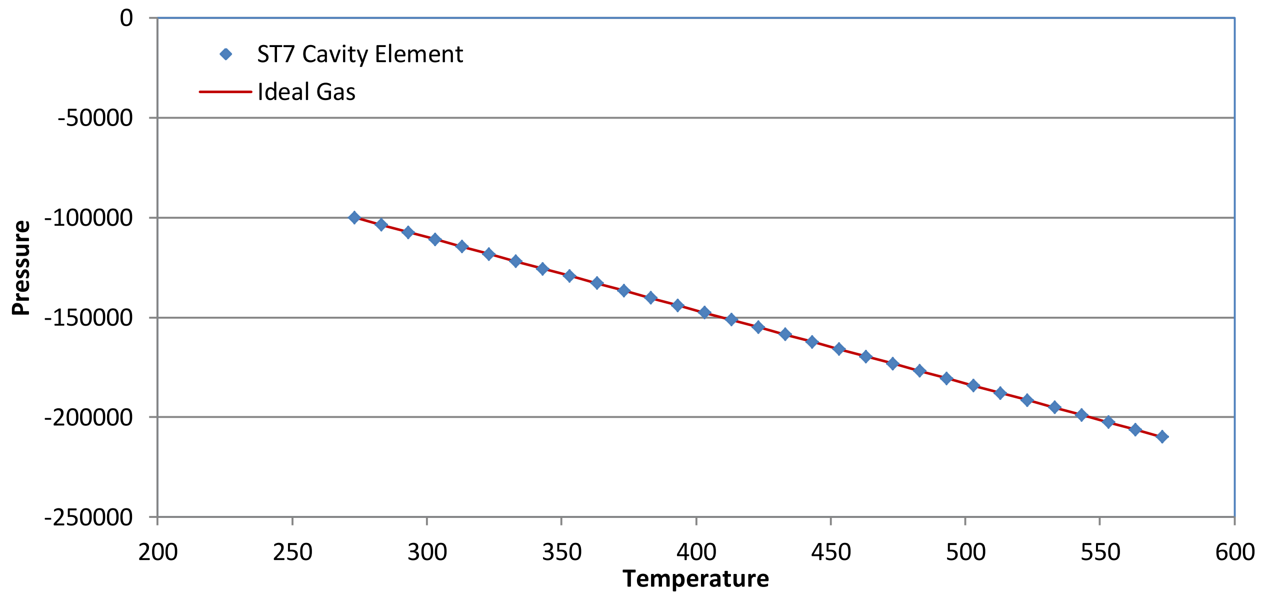
Case 1: Increasing the volume by enforced displacements and monitoring the cavity pressure.



Case 2: Decreasing the volume by enforced displacements and monitoring the cavity pressure.



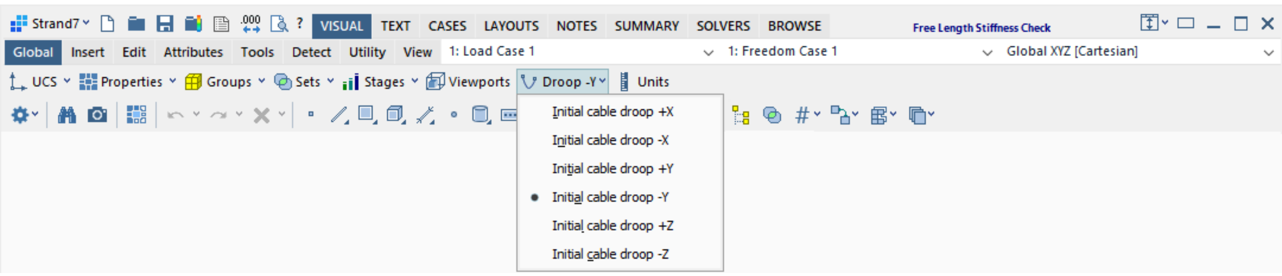
Case 3: Keeping the volume constant and increasing the cavity temperature.



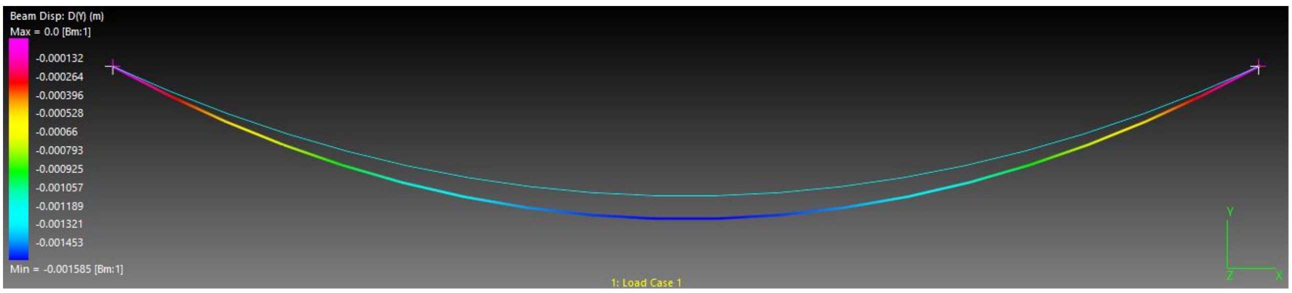
The above graphs report the pressure acting on the plate (or brick) surfaces. For an ideal gas cavity, the element surface pressure will range between zero (i.e., a vacuum) and some negative number (i.e., a compressive pressure on the surface).

Initial Cable Droop

The cable droop setting defines the initial shape and position of catenary cable elements before any loads are applied.



The initial shape is determined by solving the catenary equation for an inextensible cable of the prescribed length, drooped in the nominated direction. This then becomes the zero-displacement position in the analysis. Displacements produced by the various solvers are then assumed to be with reference to this initial shape (i.e., position) and therefore cable displacements, in addition to cable positions, can be determined even for the first load step of an analysis.

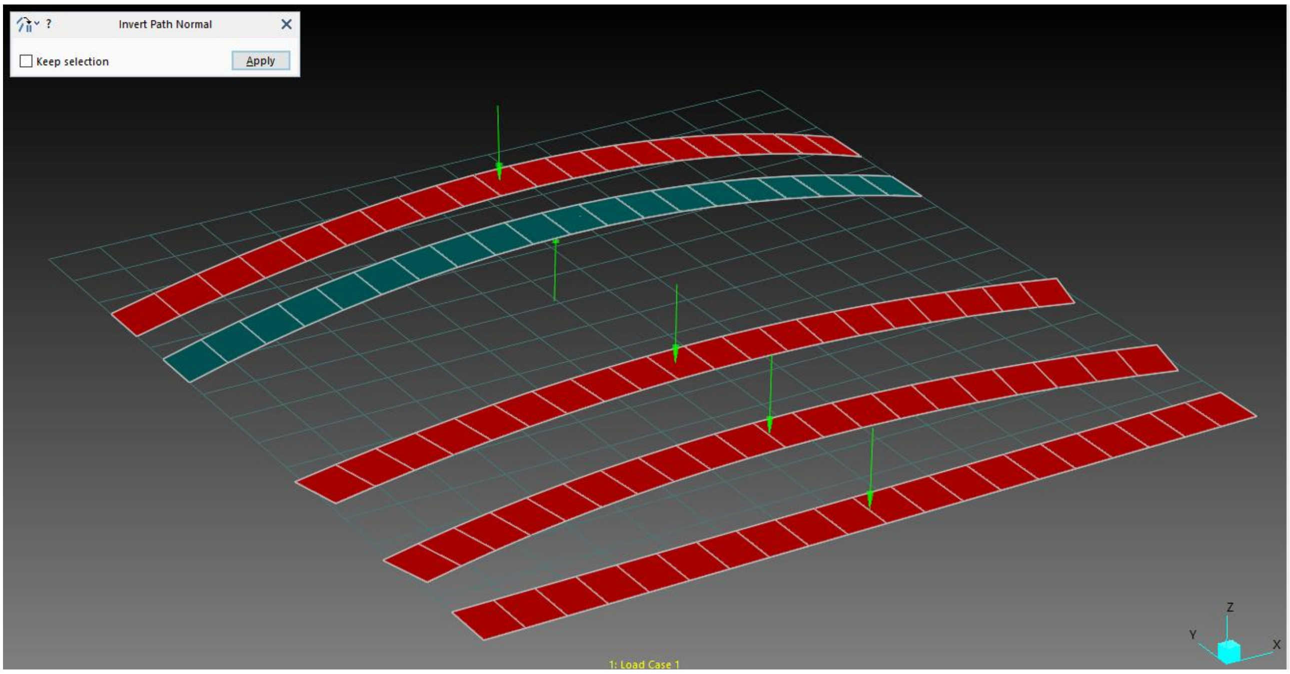


Weightless Cable

The R3 solver supports a weightless cable element without reverting to an equivalent truss element. This means that a cable free length attribute is supported not only by the standard catenary cable with inertia load (mass and gravity), but also by a massless cable that is not affected by inertia loads.

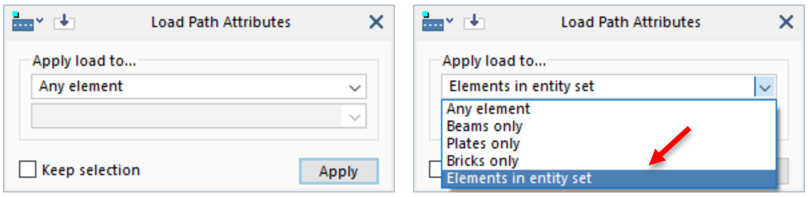
Load Path Normal

Load paths in R3 support an orientation flag allowing for the normal direction to be inverted without having to change the end connection order. This provides additional flexibility, especially for non-flat paths.



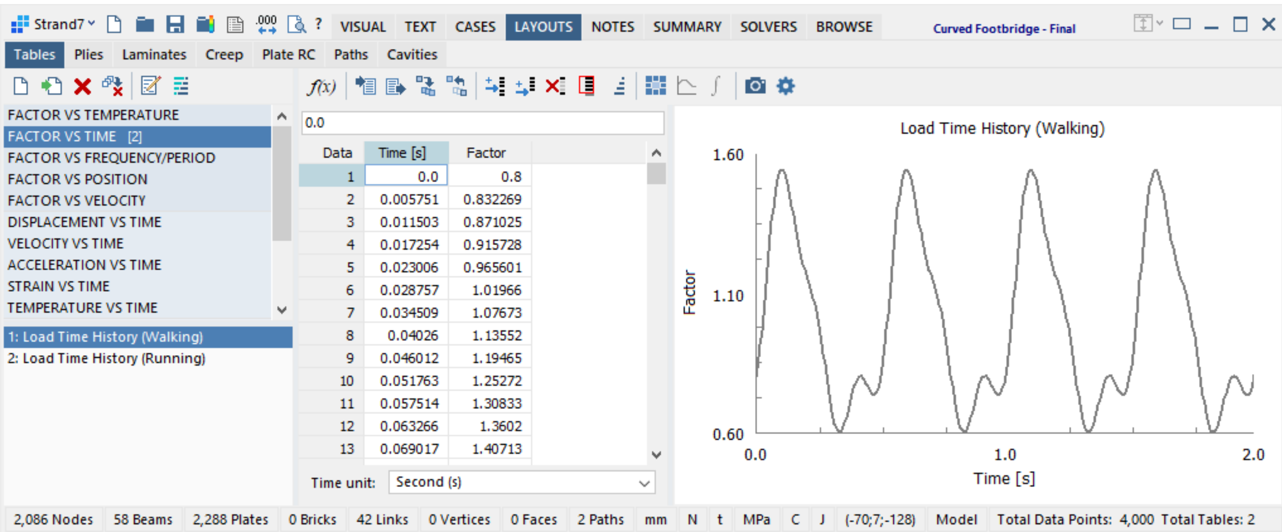
Load Path Sets

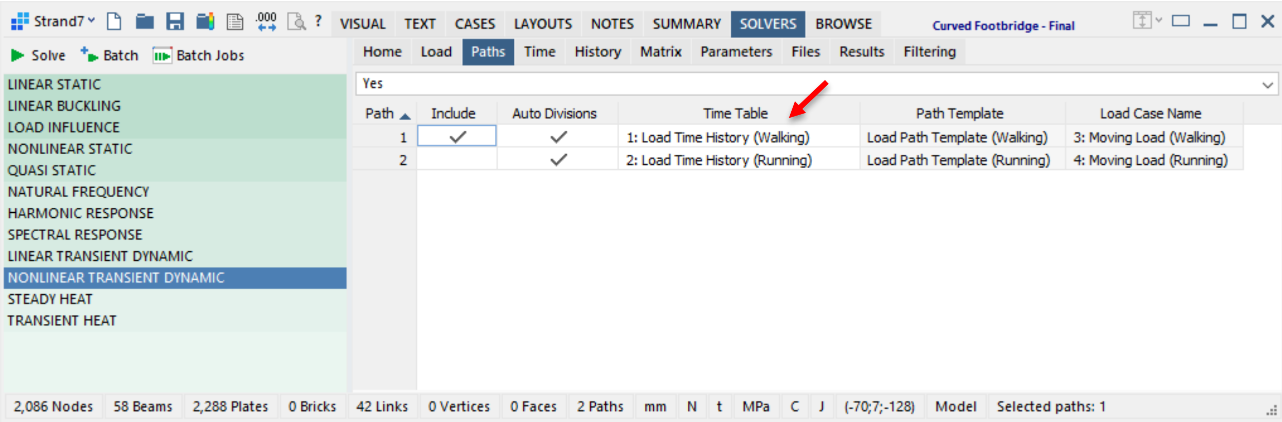
In addition to the previously available **Any element**, **Beams only**, **Plates only** and **Bricks only**, R3 offers the option of transferring the loads from a load path to only the elements contained in a specific entity set, **Elements in entity set**.



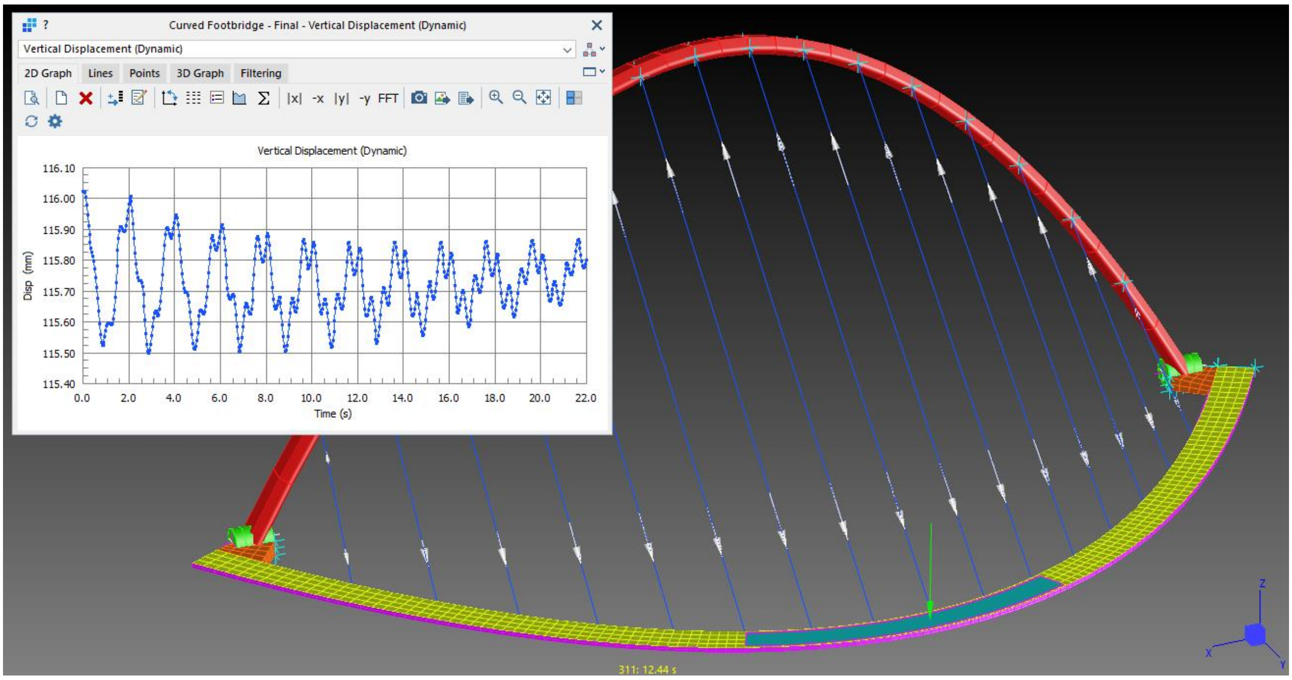
Load Path Time Tables

Loads on load paths may reference a Factor vs Time table in R3, enabling the modelling of loads whose position and magnitude (or direction) may both change with time.



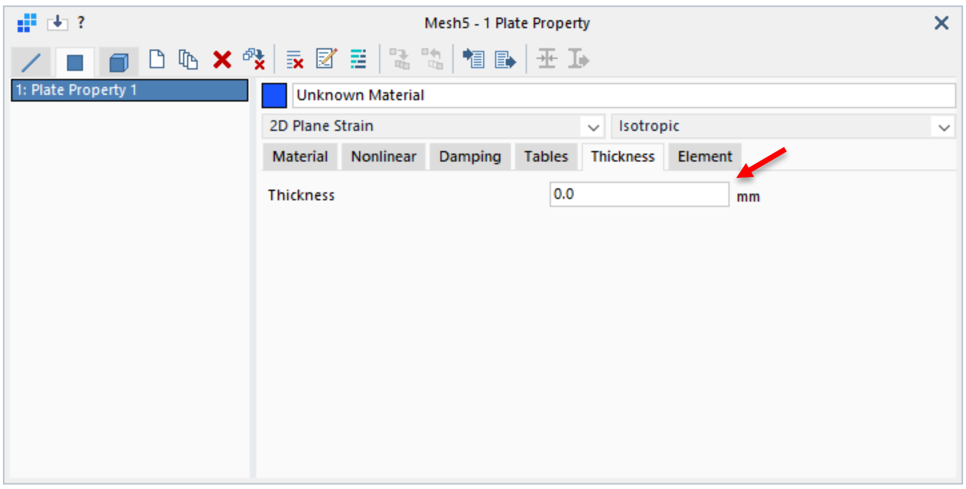


An example use of this feature includes the modelling of footfall on a bridge.



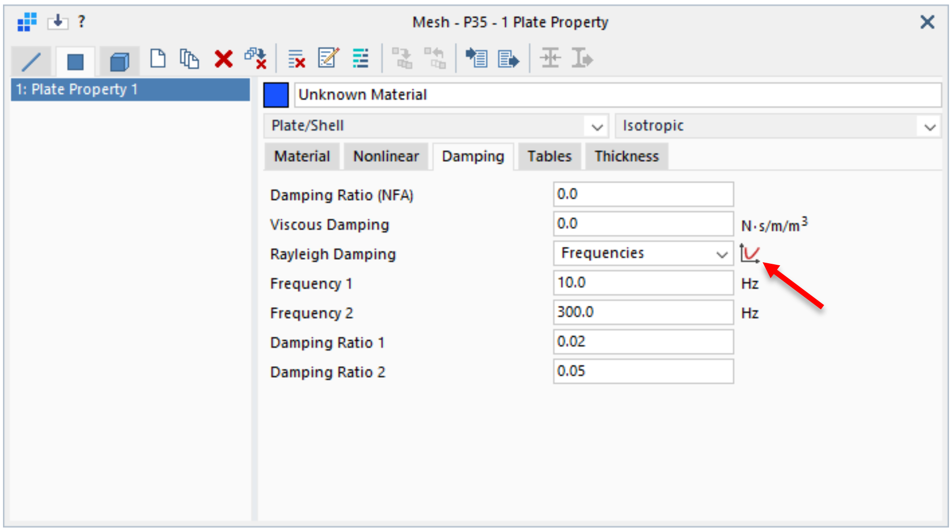
Plane Strain Thickness

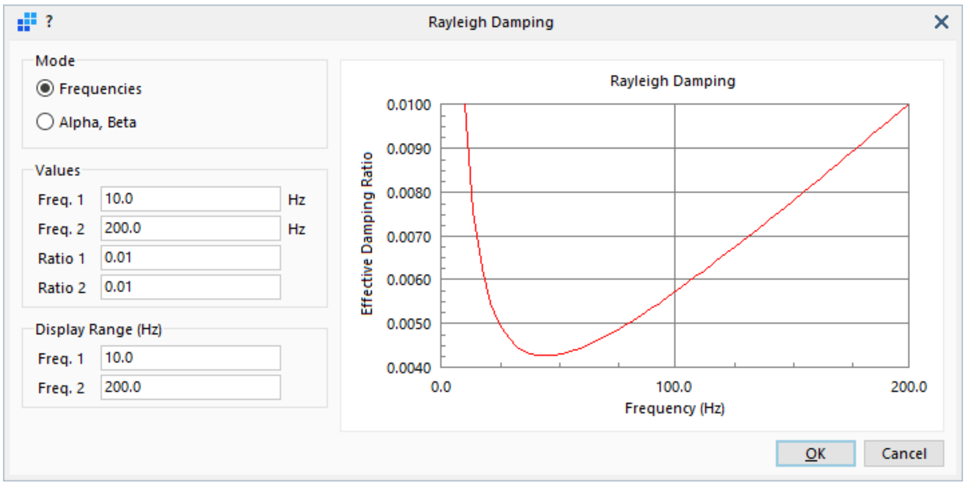
R3 supports a user-defined thickness for plane strain elements (R24 always considers plane strain as a unit-thickness element). The advantage of the user-defined thickness for plane strain is that it allows plane strain properties to be fully units-aware, ensuring that a units scaling operation does not change the physical problem.



Property-based Rayleigh Damping

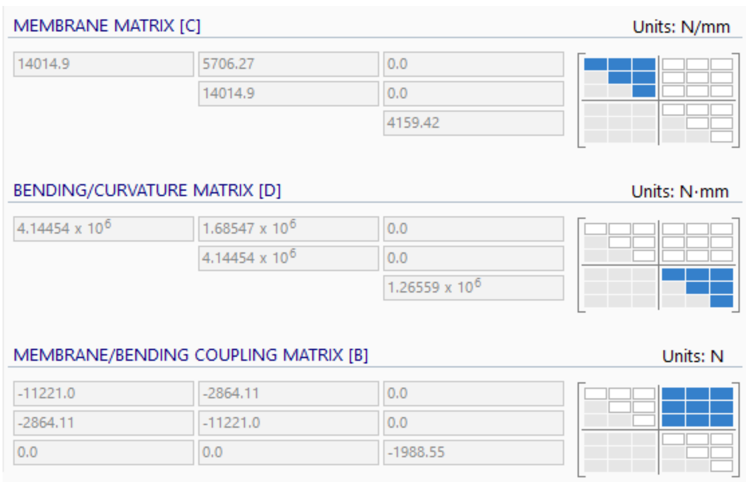
Rayleigh damping for linear and nonlinear transient dynamic analysis may be defined at the property level in R3. This applies to all of beam, plate and brick property types. This feature allows for different damping characteristics to be considered for different material types. The option to apply Rayleigh damping based on the global mass and stiffness matrices is still available.



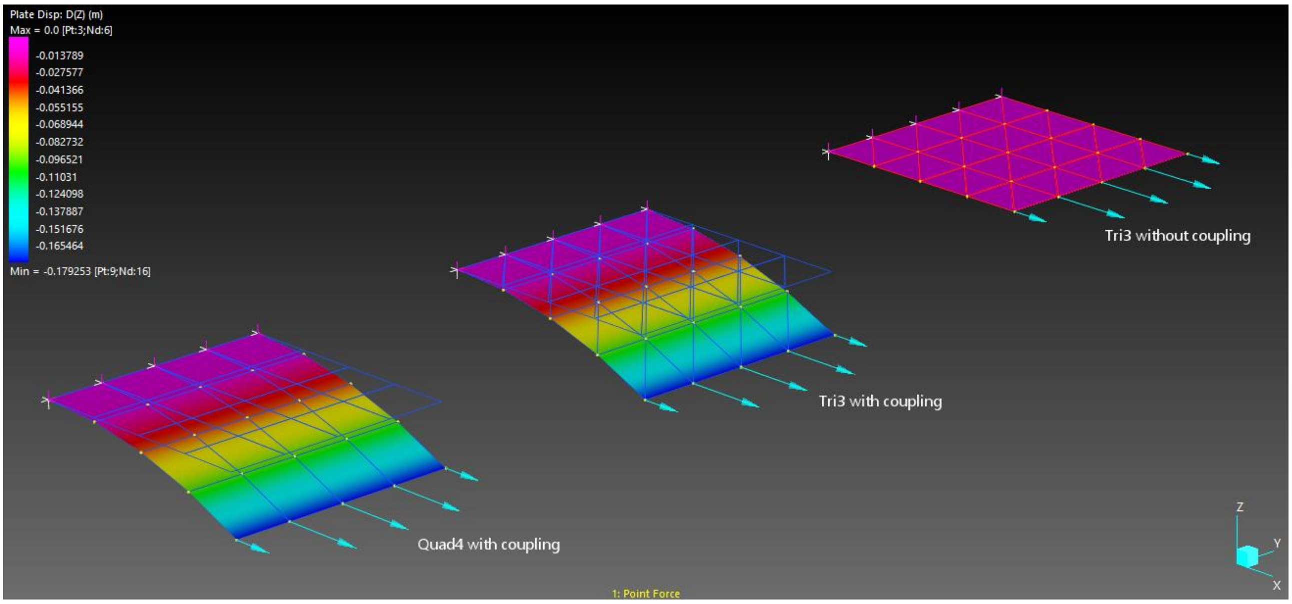


Tri3 Membrane-Bending Coupling

Tri3 shell elements in R3 support membrane-bending coupling. This is particularly useful for the analysis of laminated composites in mixed Quad4/Tri3 shell meshes.

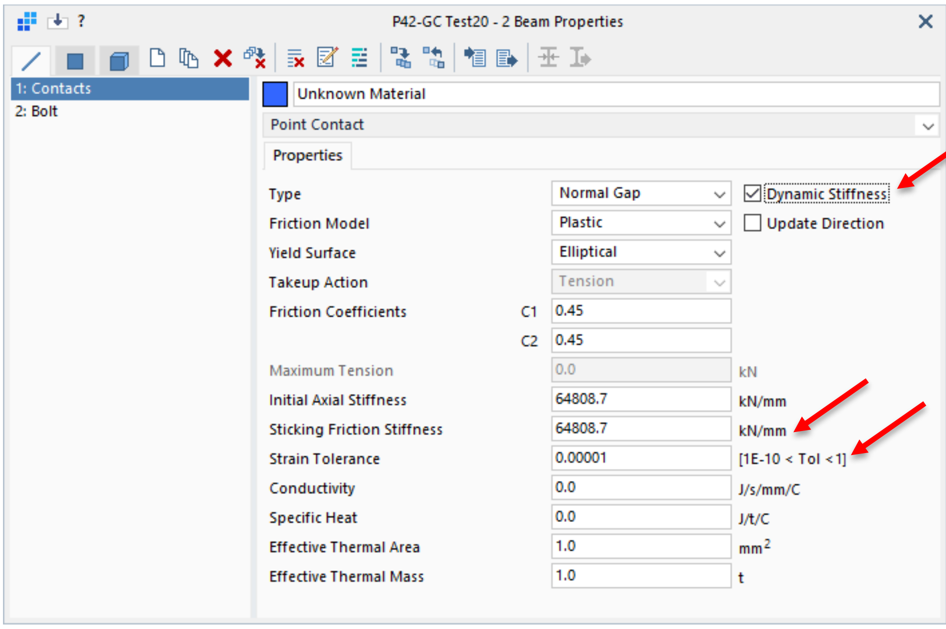


Without support for membrane-bending coupling, the [B] matrix is ignored and therefore a membrane force produces no out-of-plane bending, as illustrated below.



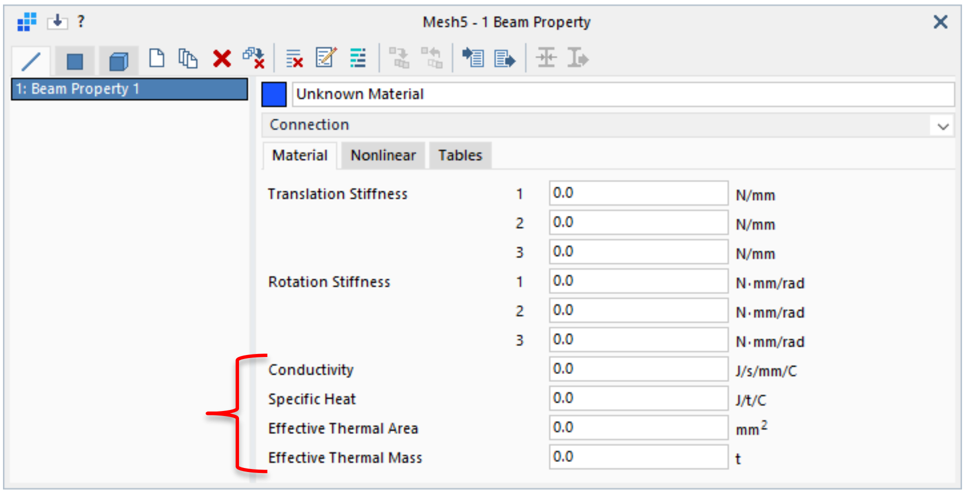
Point Contact Elements

In R3, point contact elements consider the Dynamic Stiffness parameter in nonlinear transient dynamic analysis (care needs to be taken when the stiffness is very high since this may require very small time steps to integrate the time equations accurately). In addition, the sticking friction stiffness is now explicitly set at the property level for each point contact type, and not defined as a fixed ratio of the initial axial stiffness via a solver parameter. The new **Strain Tolerance** parameter, analogous to the Contact Cutoff Strain solver parameter in R24, is also a property-level parameter in R3. This is useful to enable different factors to be used for relatively short contact elements compared with relatively long contact elements contained in the same model.



Heat Transfer for Line Elements

All line elements (beam properties) in R3, including Spring-Damper, Connection and Point Contact, may be used in heat transfer analysis and they can all produce a thermal stiffness matrix by the definition of equivalent thermal properties including Conductivity, Specific Heat, Effective Thermal Area and Effective Thermal Mass.



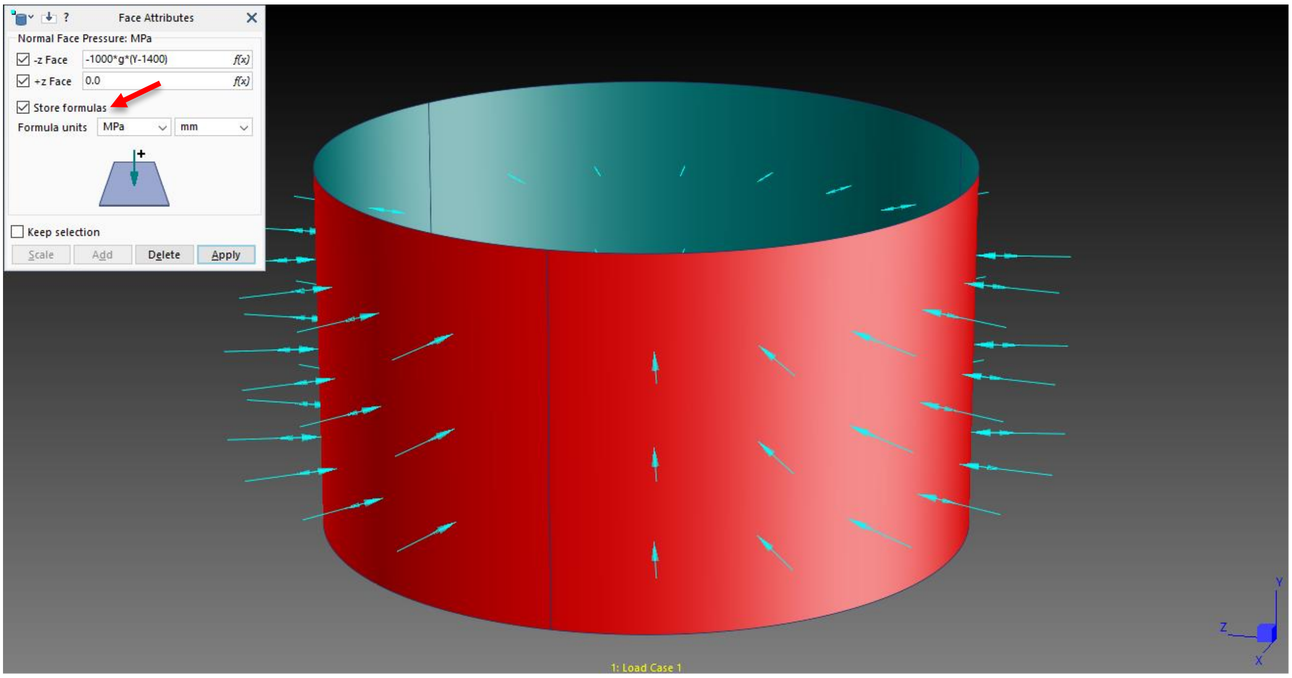
ATTRIBUTES – MODELLING FEATURES

Store Formulas

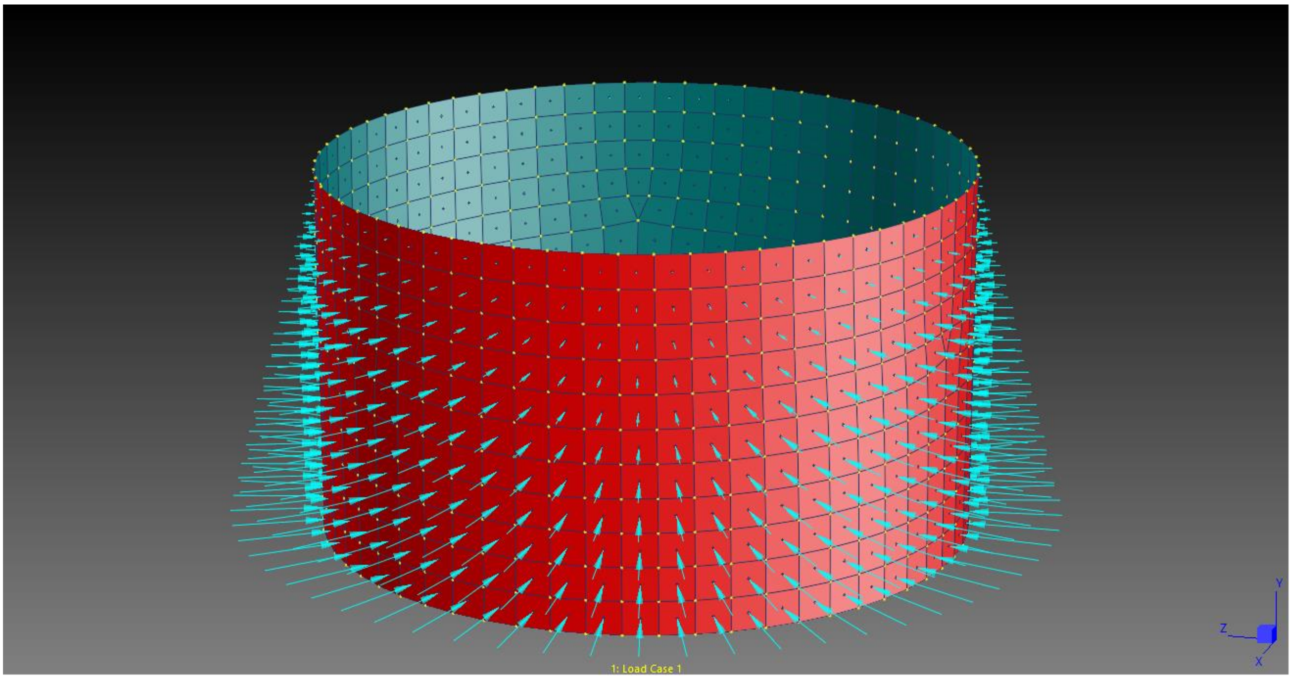
Strand7 has always provided the option to use parametric formulas for the application of attributes – for example, a hydrostatic pressure could be easily applied by selecting the plate elements and assigning a plate pressure attribute using a formula containing a position-dependent variable. When applied in this way, the numeric value is assigned but its association with the original formula is lost. If the mesh is refined, the pressure would need to be re-applied using the formula.

In R3, it is possible to store formulas with the attribute. These formulas are supported by both elements and geometry faces. Tools are available to edit, delete and re-evaluate the formulas.

In the example below, a hydrostatic pressure formula is applied to geometry faces and stored with the attribute.



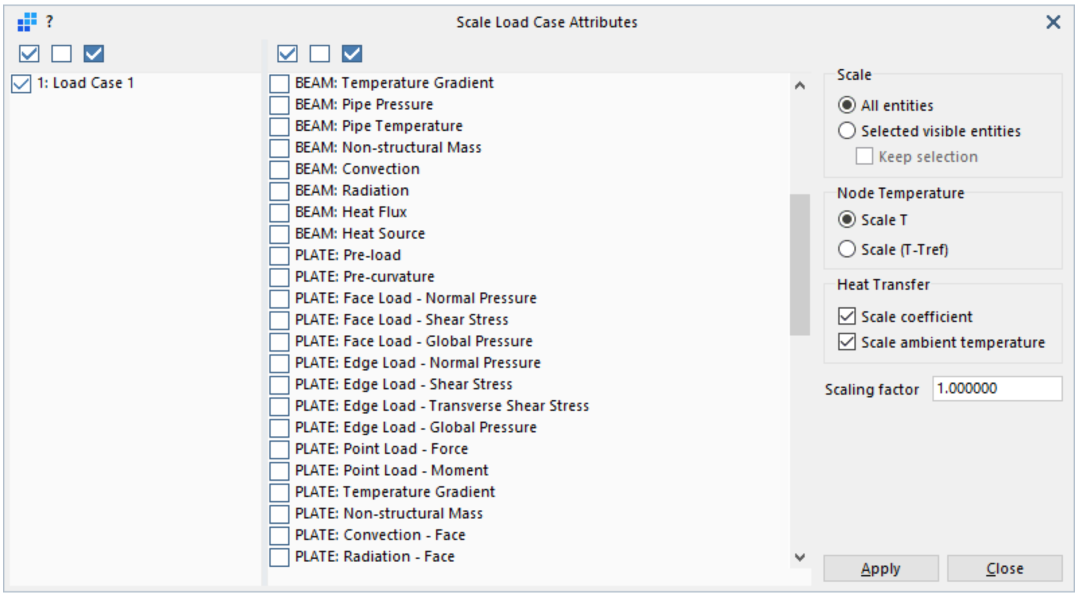
When the faces are meshed, the elements evaluate the formula so that each element has the correct value of pressure assigned.



This feature is applicable not only to geometry, but also to elements. If applied to plate elements, a subdivision operation will re-evaluate all of the attributes on the new elements.

Attribute Bulk-edit Functions

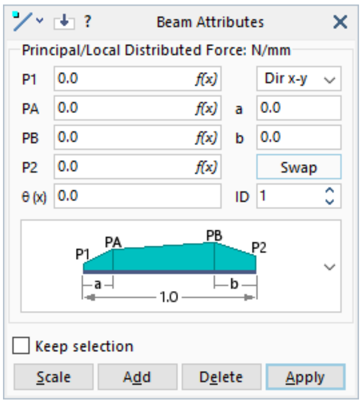
This tool provides a way to make bulk changes to attribute values in the model.



ATTRIBUTES – NEW OR ENHANCED

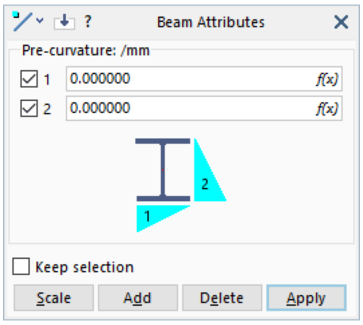
Beam Local Distributed Force

Distributed force attributes on beam elements may be assigned in the local x-y directions in addition to the previously available principal 1-2 directions.



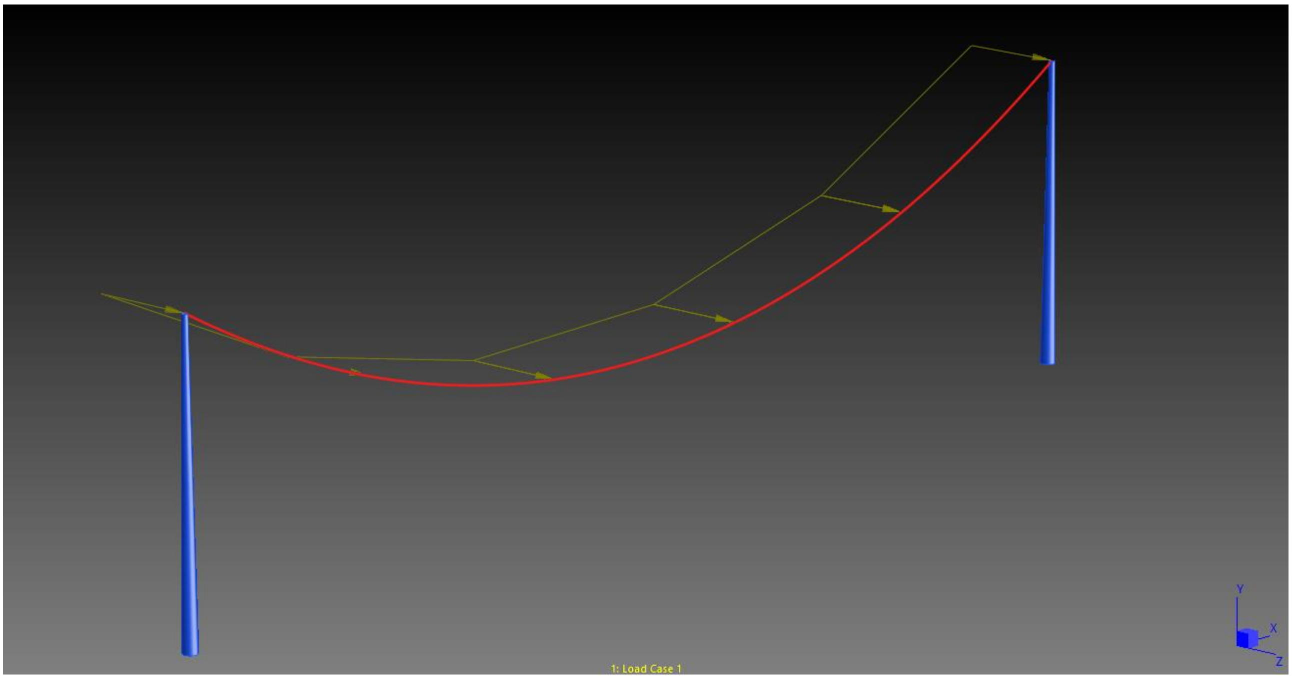
Beam Pre-Curvature

Beam elements in R3 support a pre-curvature attribute in each of the two principal axis directions.



Global Distributed Load on Cables

Cable elements in R3 support a uniform global distributed load attribute. Non-uniform distributions may be applied, and these are smeared over the length of the cable as equivalent uniform load.



Beam Side Attachment

In R3, attachment attributes at the ends of beam elements may be assigned in the principal (i.e., lateral) directions, in addition to the previously available axial direction. This enables beam elements to be attached to plate meshes via side attachment links as illustrated below.

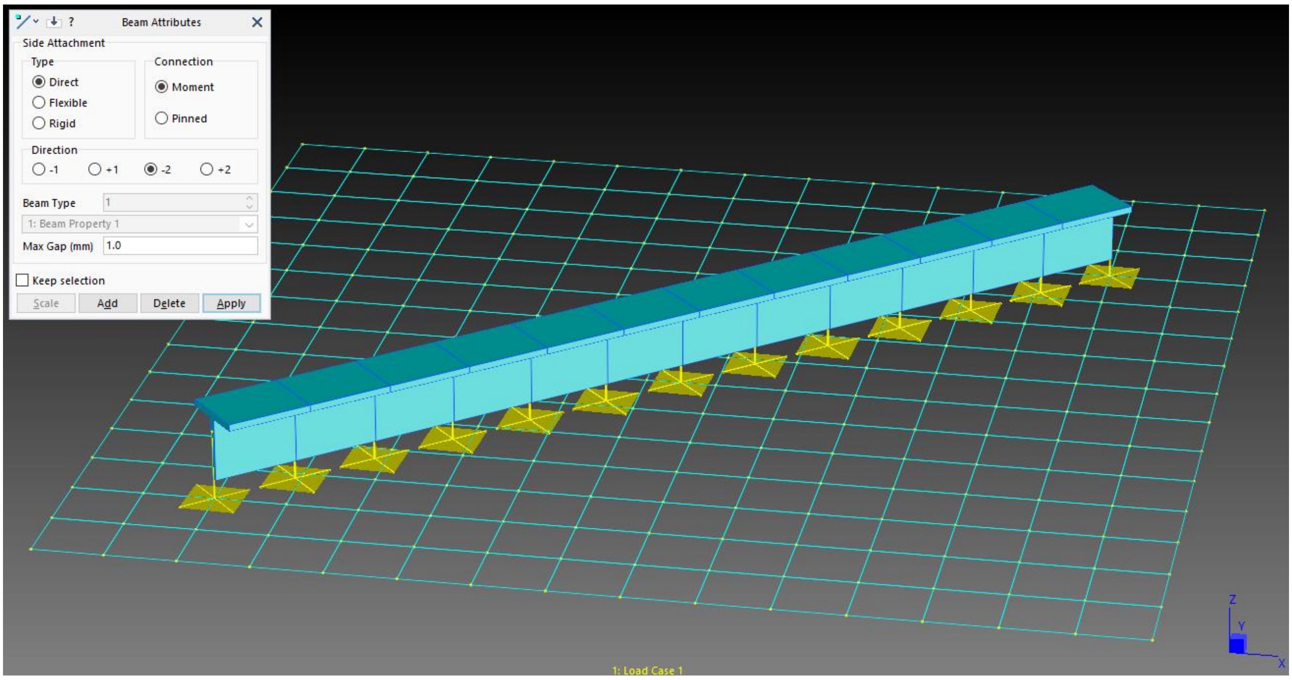
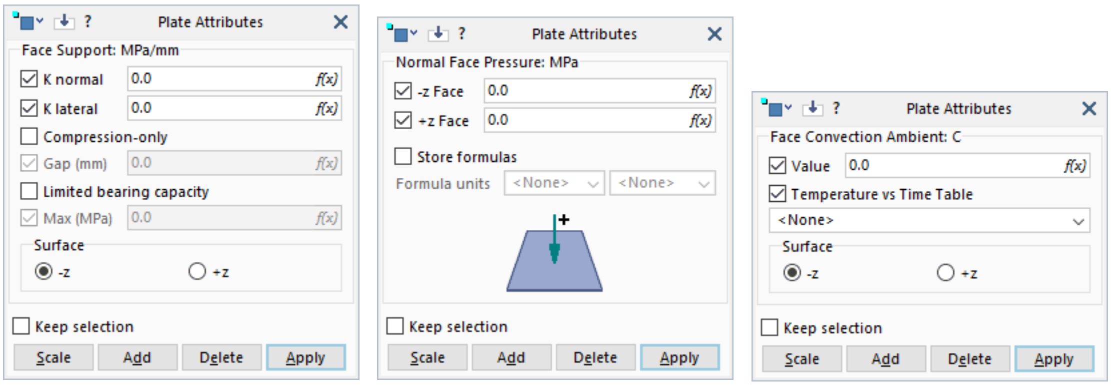


Plate Surface-dependent Attributes

Plate face support, plate face pressure and plate face heat transfer attributes in R3 explicitly reference the plate surface (i.e., -z or +z). These attributes can be assigned to either surface or to both surfaces at the same time.



Support Attributes – Lateral Stiffness, Gap, Bearing Capacity, Direction

Plate and brick support attributes offer additional parameters including a lateral (shear) stiffness and a bearing capacity limit. For compression-only supports, there is also the option to include an initial gap between the element and the support. The gap can be positive (i.e., the element has not yet made contact with its support) or negative (i.e., the element is pre-compressed into the support).

Beam support attributes in R3 also offer the gap parameter. In addition, beam support attributes may be applied in any of the principal axis directions. Multiple directions can be applied simultaneously if required.

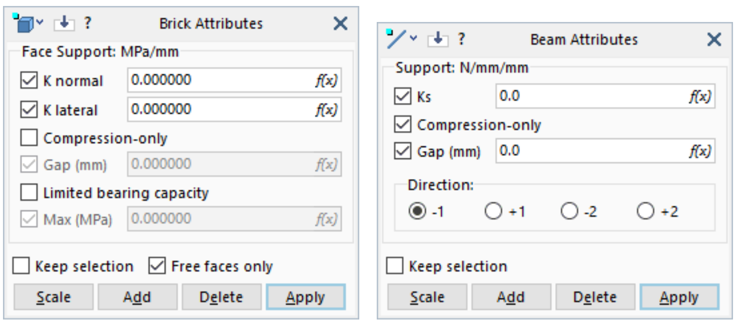


Plate Pre-Curvature

Plate elements in R3 support a pre-curvature attribute in each of the two local axis directions.

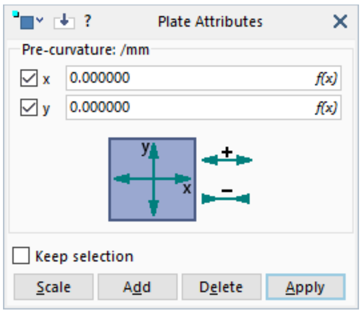


Plate Stiffness/Mass Factors

Similarly to the previously available beam Stiffness/Mass Factor attribute, plate elements in R3 also support a Stiffness/Mass Factor attribute. All of the components of the element material matrix can be independently scaled to produce a pseudo-orthotropic material matrix from the isotropic material parameters and scaling factors.

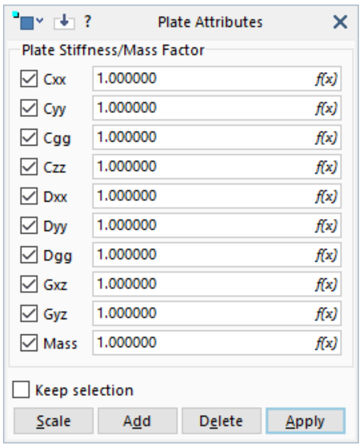
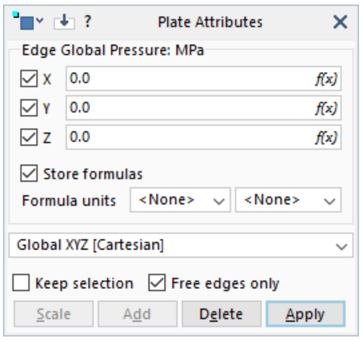


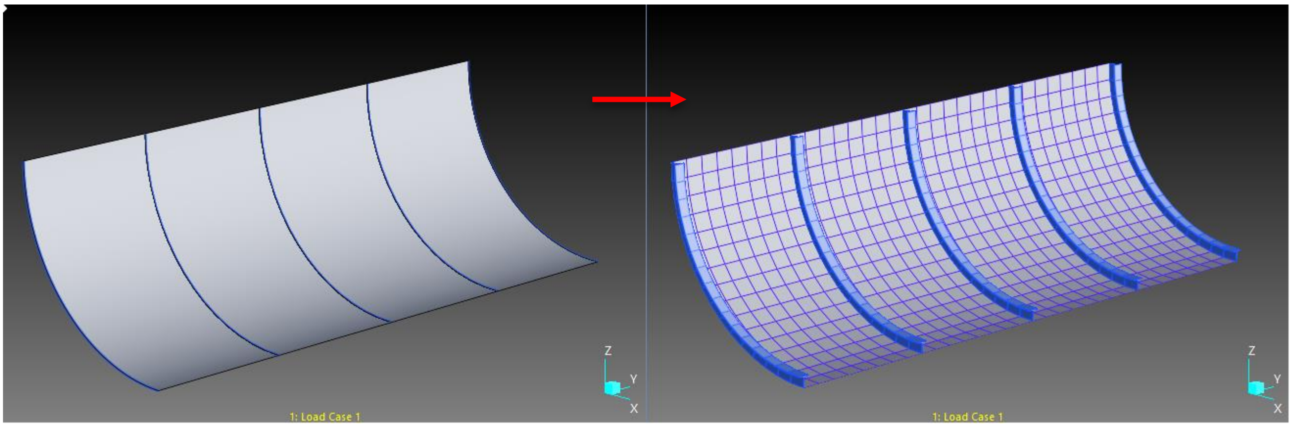
Plate Edge Global Pressure

Plate edge pressure attributes in R3 may be assigned with reference to the global Cartesian axis system or a UCS, in addition to the previously available local direction.



Geometry Edge Attribute – Edge Beam

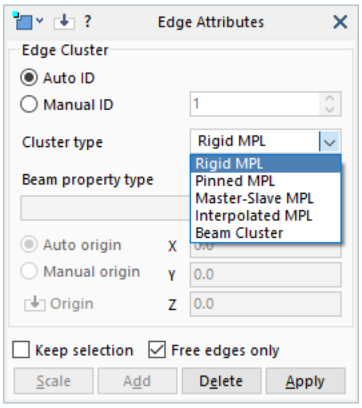
Geometry face edges may reference a beam property number in R3. When automeshing the face, a series of overlaying beam elements will be automatically inserted along the edge. These beam elements will be compatible with the generated plate mesh, automatically aligned, and will be connected between adjacent nodes along the edge. The attribute is typically used to automatically generate stiffeners modelled with beam elements on panels modelled with shell elements.

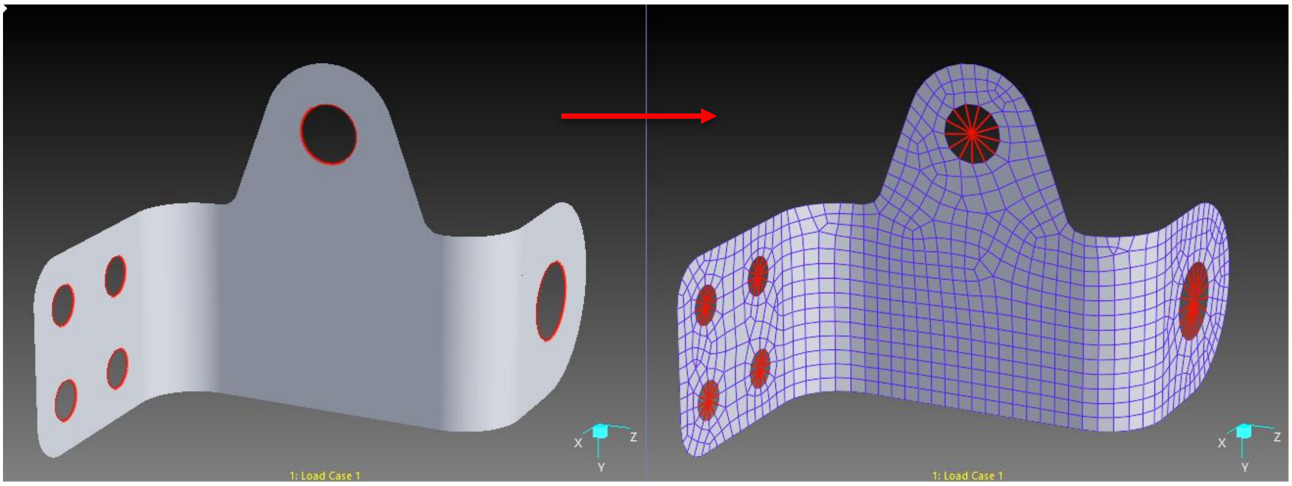


Geometry Edge Attribute – Edge Cluster

This attribute references a link type or beam property by the assignment of a cluster ID, which is used by the automesher. All nodes generated on edges with the same cluster ID are automatically connected to a cluster of the selected type by automeshing. The cluster consists of a slave node connected to every node on the edges.

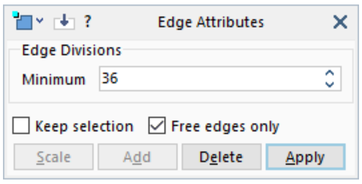
The Auto ID option facilitates the bulk assignment of cluster IDs on multiple disconnected loops. Cluster types may be Rigid MPL, Pinned MPL, Master-Slave MPL, Interpolated MPL or beam elements.

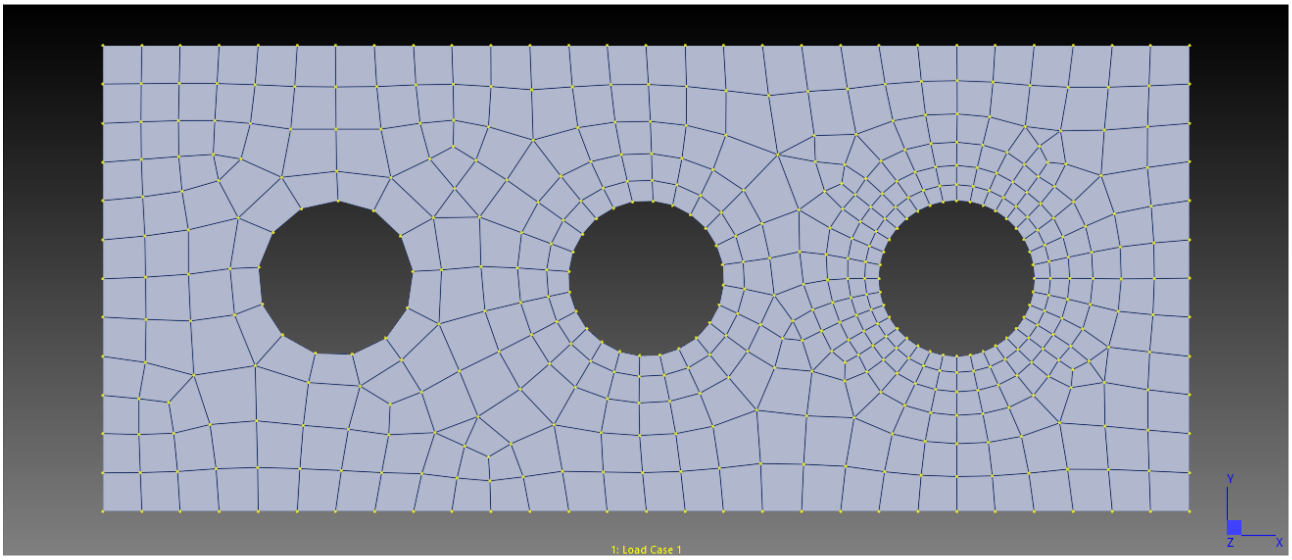




Geometry Edge Attribute – Divisions

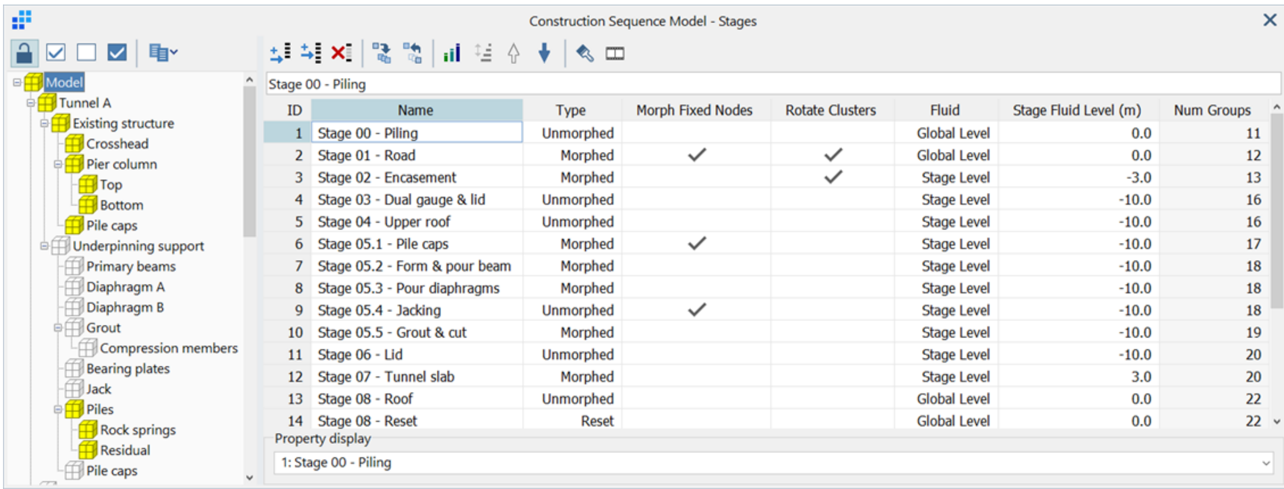
This attribute is used by the automesher and augments the previously available vertex mesh size attribute. Whereas the vertex mesh size attribute assigns a length-based division at a vertex, from which edge divisions are deduced, the edge divisions attribute specifies the required number of divisions directly. In the image below, three holes in the same face are meshed with different edge divisions.





STAGING

In addition to the redesigned stage editor shown below, the stage functionality in R3 offers new options.



Stage-dependent Soil Fluid Level

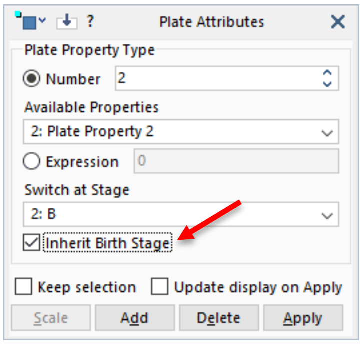
Fluid level can be set on a stage-by-stage basis to facilitate the modelling of dewatering and other events that change the fluid level.

Stage Type – Reset

In addition to the previously available **Morphed** and **Unmorphed** stage types, stages can now also be of the **Reset** type. A reset stage clears all element and node history and the analysis starts again as a completely new analysis at that stage. Subsequent staging events can follow the reset stage.

Property Switch with Birth Stage Inheritance

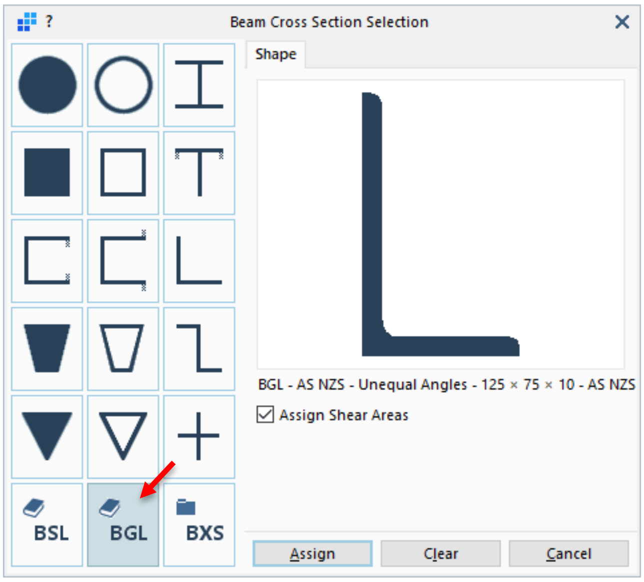
A property switch is available to change an element from one type to another at a given stage. For example, a soil element can be switched to a concrete element. By default, the switched element's birth stage is set as the stage at which the switch occurs. A new option, **Inherit Birth Stage**, enables the switched element to take on the birth stage of the element it replaces. This can be useful when the replaced element was morphed into place at its birth stage, and the switched element is required to have the same initial, not current, dimensions of the element it replaces.



FEATURES

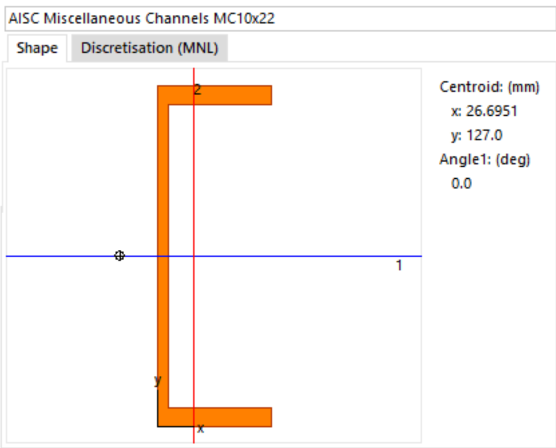
Beam Geometry Library (BGL)

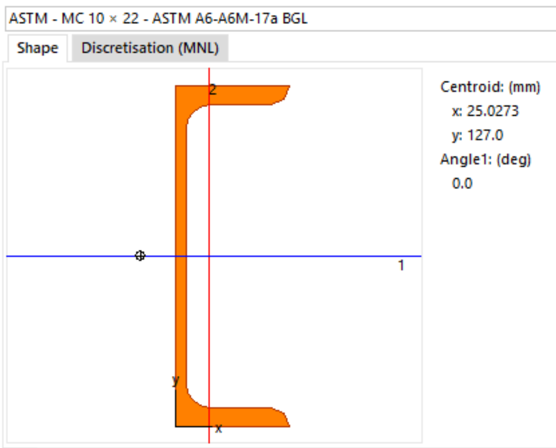
In addition to the previously available Beam Section Library (BSL), which is itself greatly expanded in R3 with new collections of sections from around the world, a new type of cross section library, the BGL, is introduced in R3.

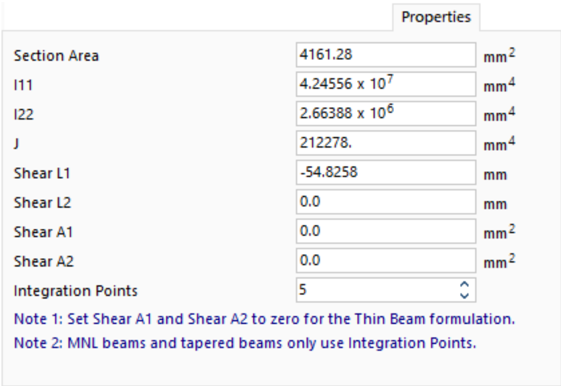


Cross sections in the BGL consider a more detailed geometry than the simplified six-parameter definition of section shapes in the BSL. BGL sections take into account the root and toe radii, flange tapers and more. These allow for a better representation of the geometry for more accurate evaluation of shear stress and nonlinear material behaviour.

The following images compare the BSL and BGL versions of the same cross section.







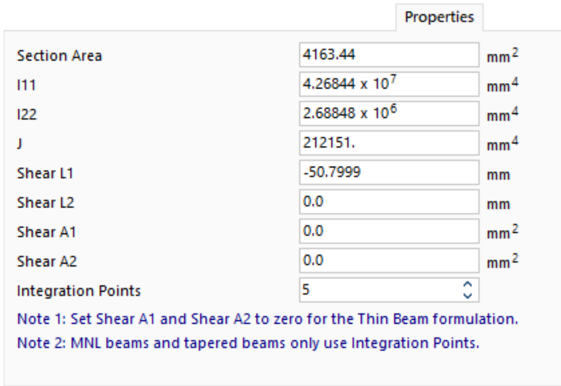
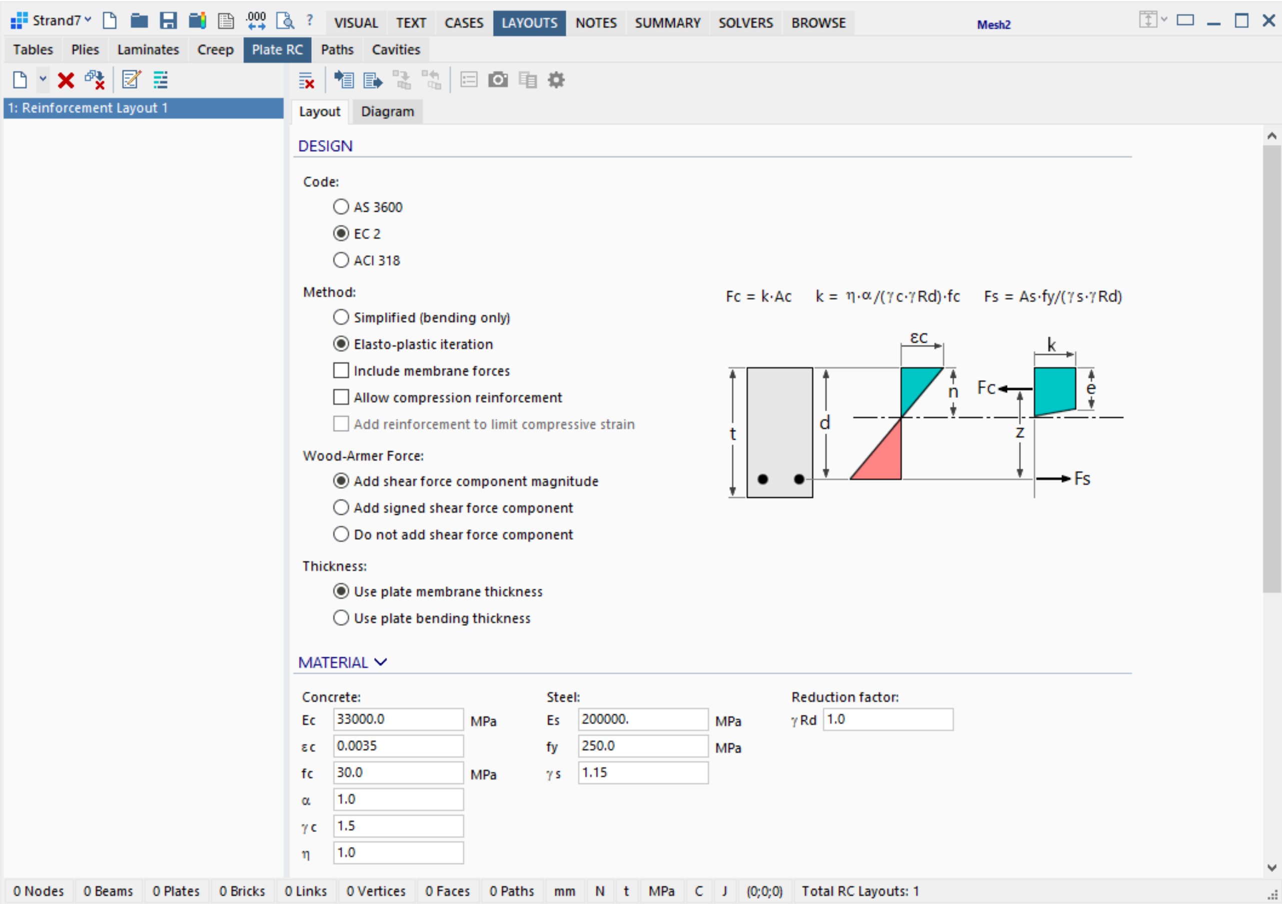
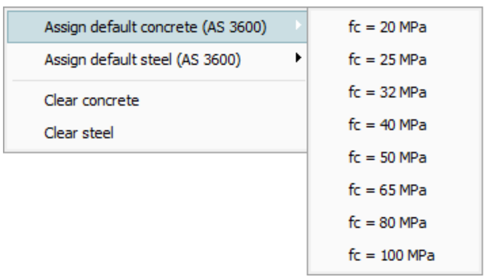


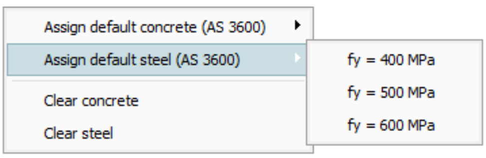
Plate Reinforced Concrete Module

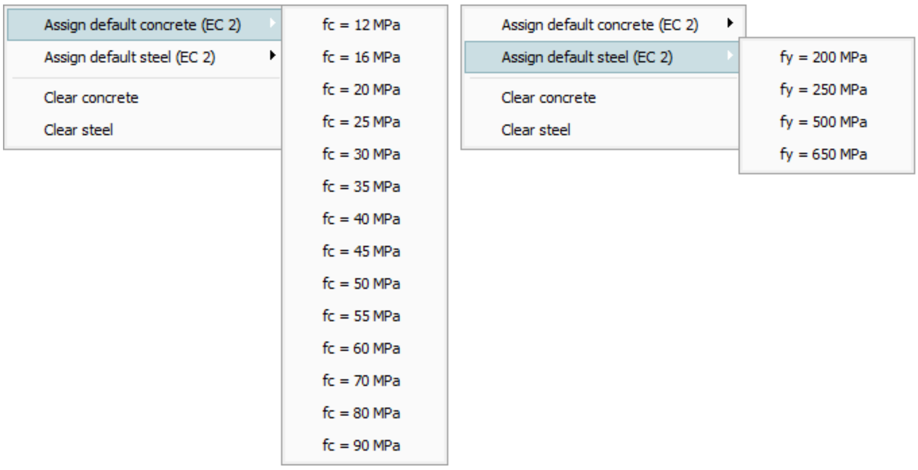
Compared with previous Strand7 releases, the Plate RC module in R3 provides significantly greater performance of calculation when using the Elasto-plastic iteration method. It also provides support for ACI 318 in addition to the previously available AS 3600 and EC 2 parameter sets.

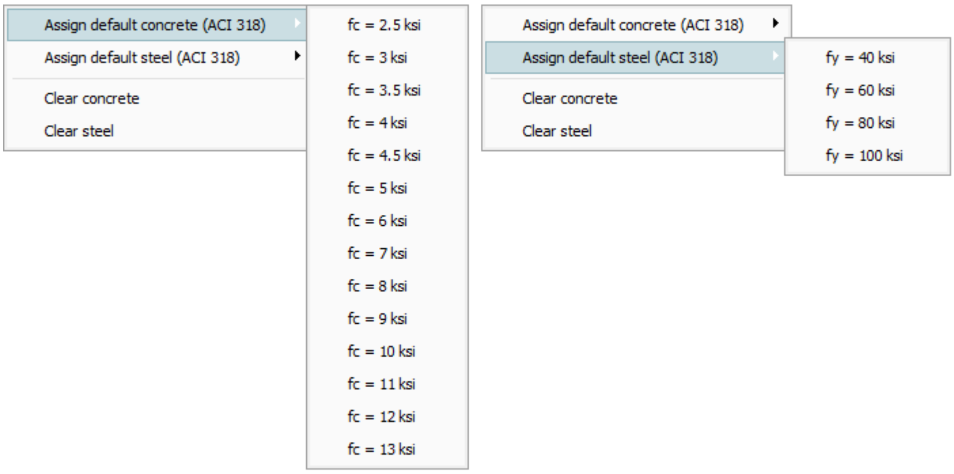


The MATERIAL section offers a popup menu with a range of standard steel and concrete properties, depending on the selected code.

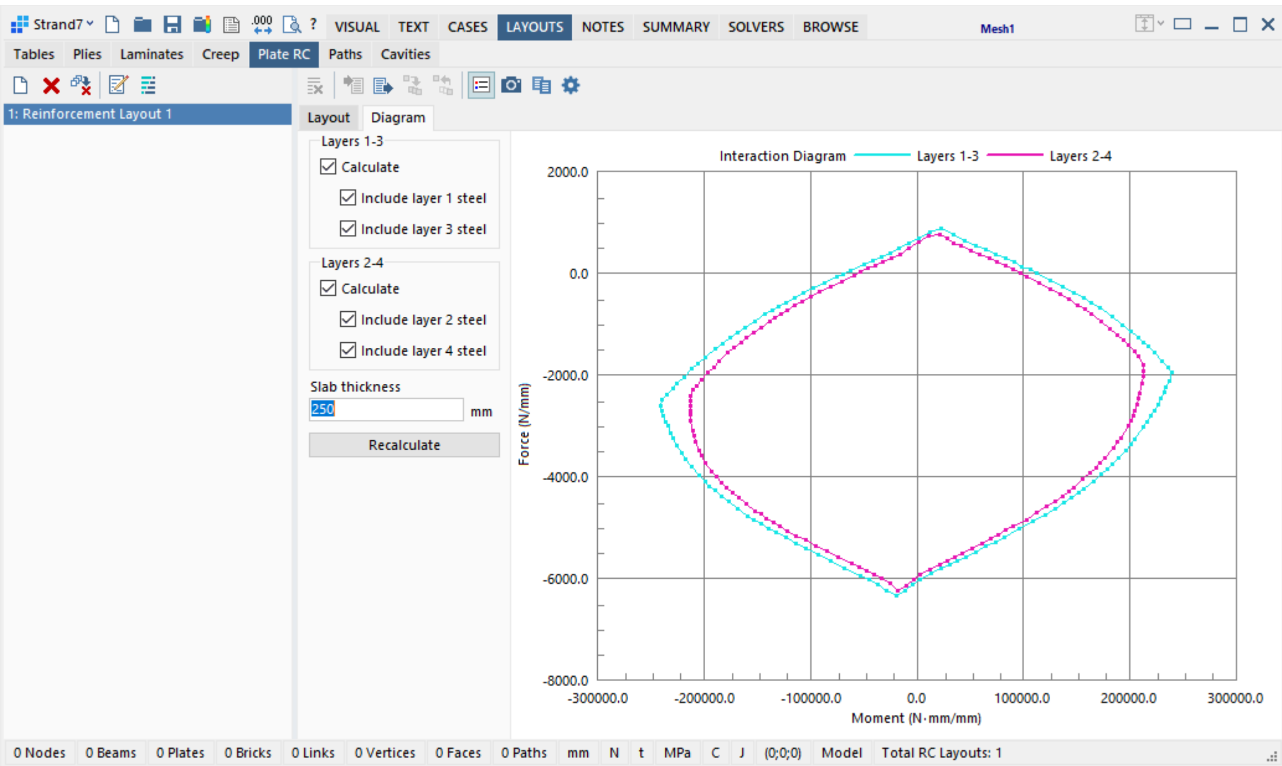








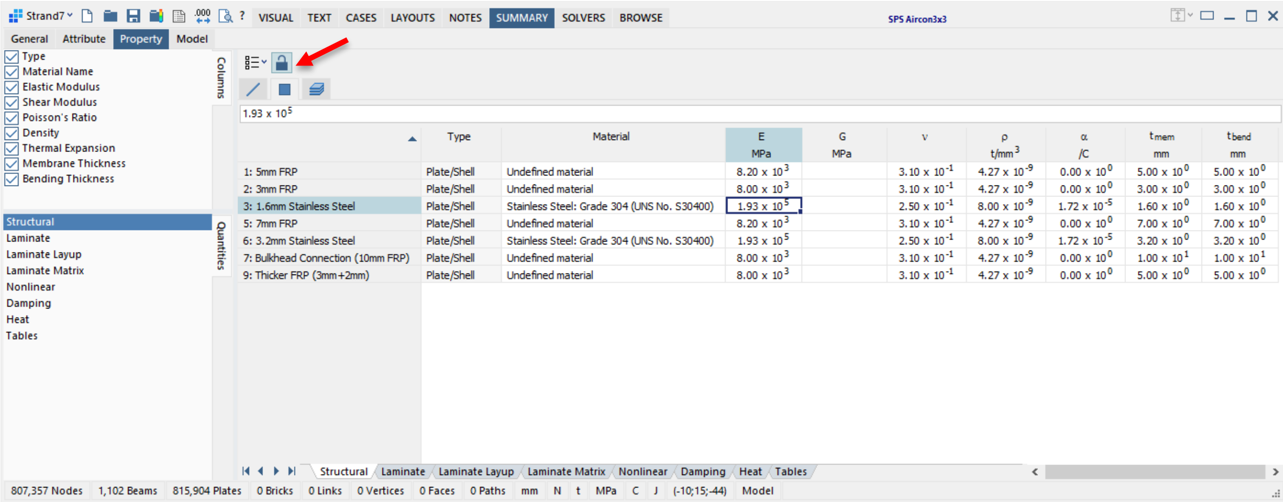
The R3 Plate RC module introduces the **Diagram** tab, which offers a useful design tool in the form of an interaction diagram calculator and visualiser.



OTHER PRE-PROCESSING

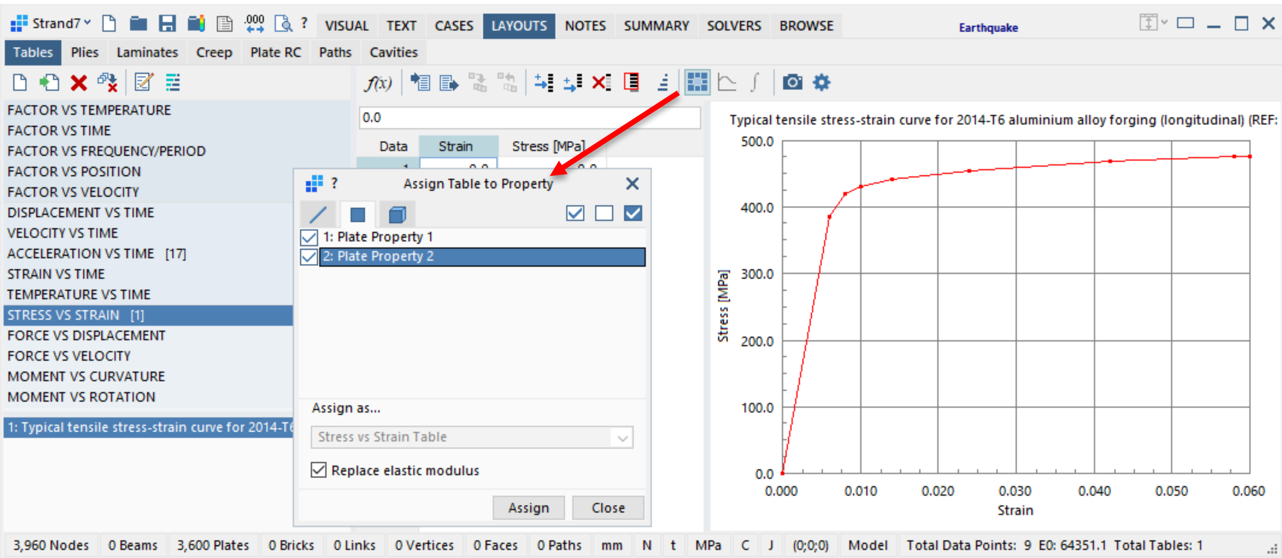
SUMMARY/Property – Bulk-edit Mode

The **SUMMARY/Property** tab offers an edit mode for bulk editing of beam, plate, brick and ply property data in a grid.



Assign to Property

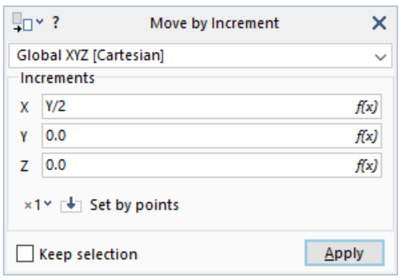
Tables, laminates and creep data can be assigned to the respective properties via the **LAYOUTS** tabs.



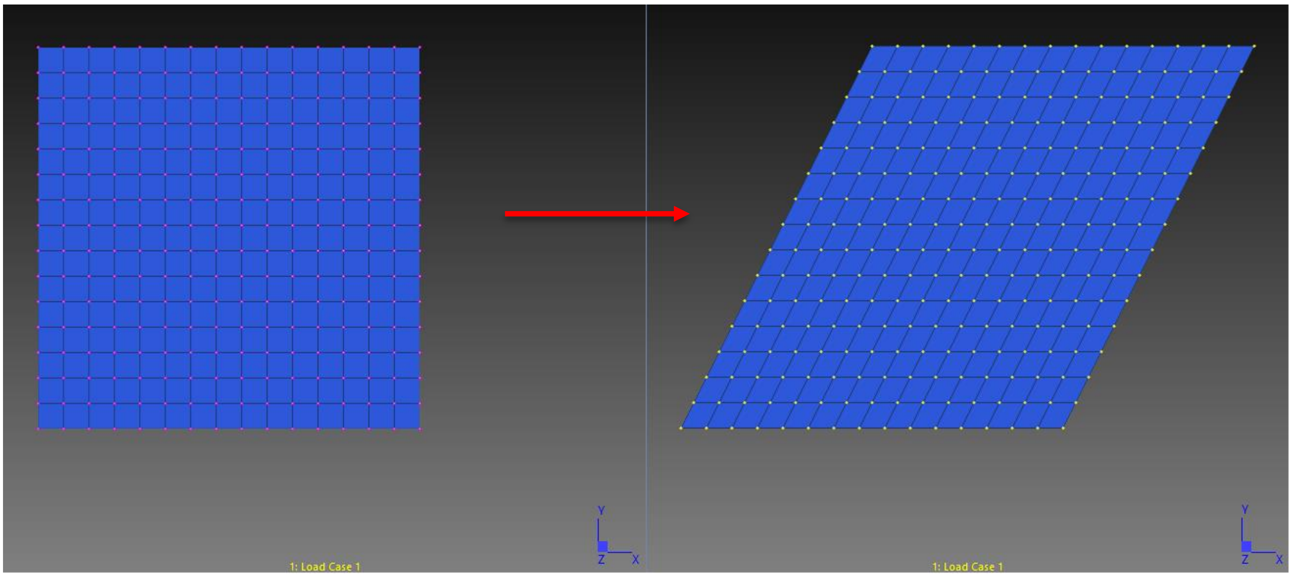
TOOLS AND RELATED

Variables

Variables may be used in tool edit boxes such as in **Copy**, **Move** and **Extrude by Increment**. For example, setting "Y/2" in the X edit box,

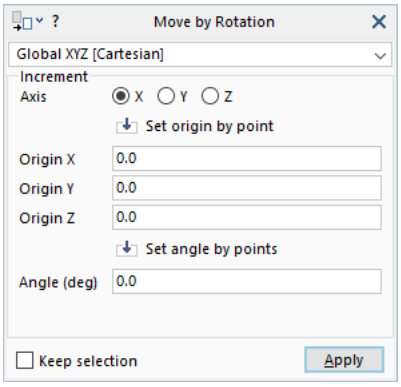


relates the X movement of a node to its Y coordinate.



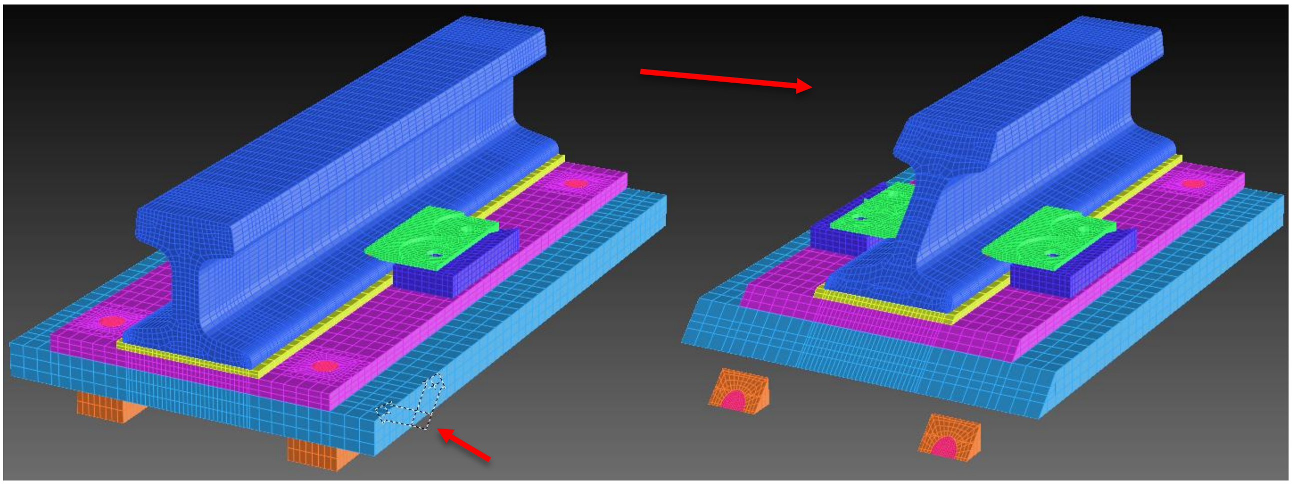
by Rotation

This is available for **Copy**, **Move** and **Extrude** tools. This obviates the need to pre-define a cylindrical UCS to operate by a rotation about an axis.



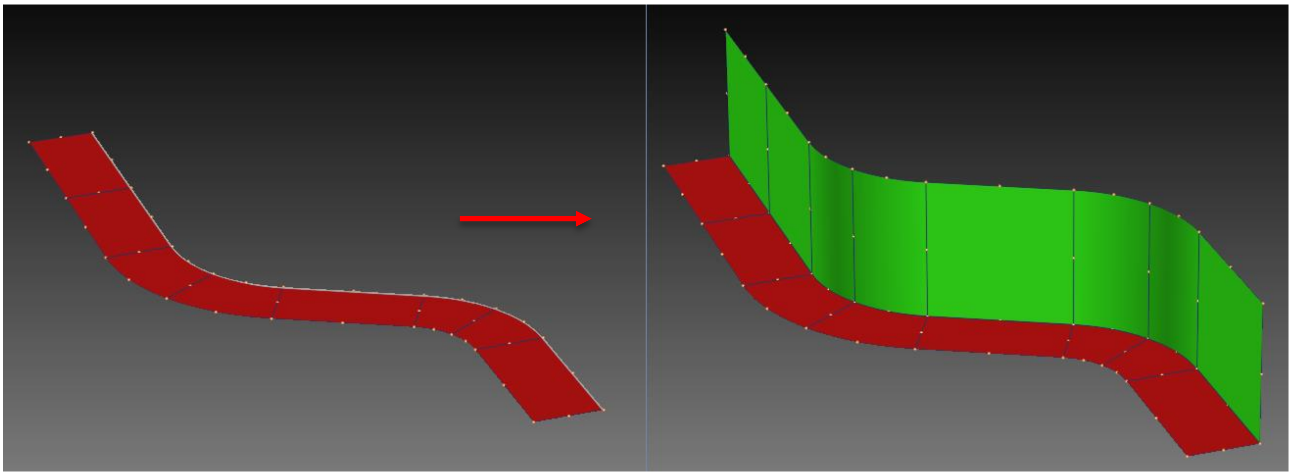
Cut Elements

This tool augments the previously available **Cut Beams and Plates** tool to enable the cutting of brick elements.



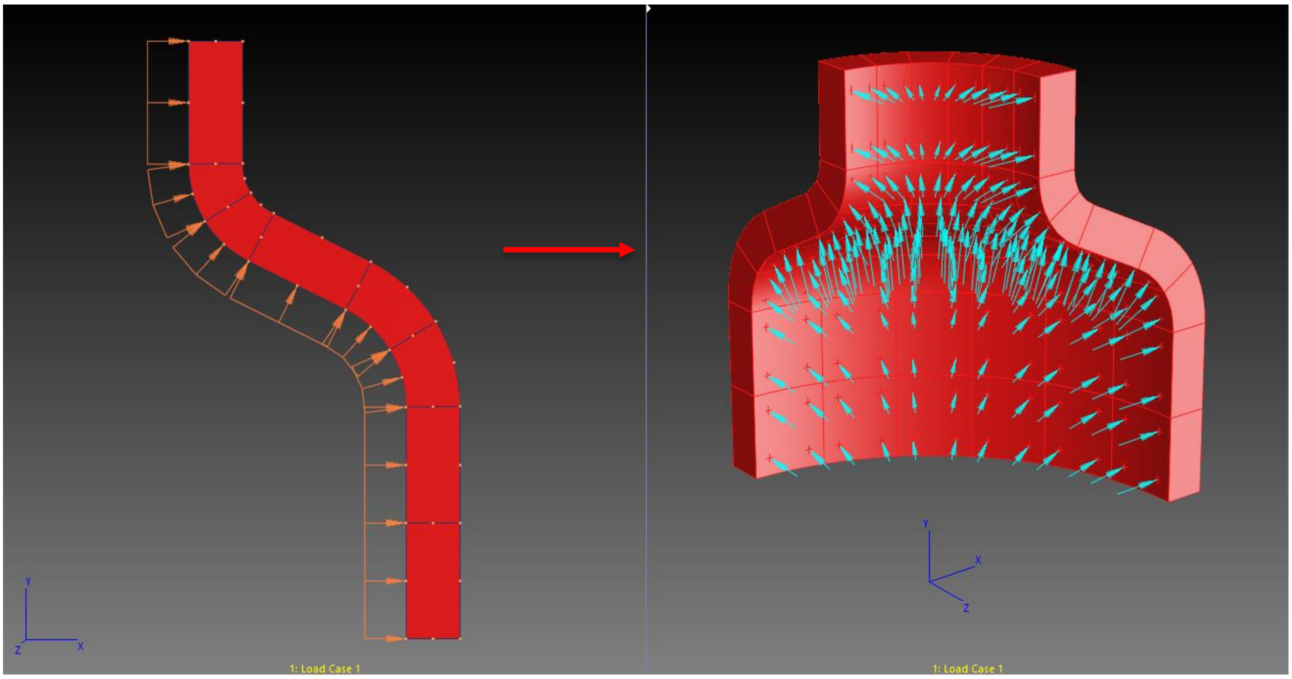
Extrude Plate Edges

The extrusion tools offer the option to extrude selected plates into brick elements, or selected plate edges into other plate elements.



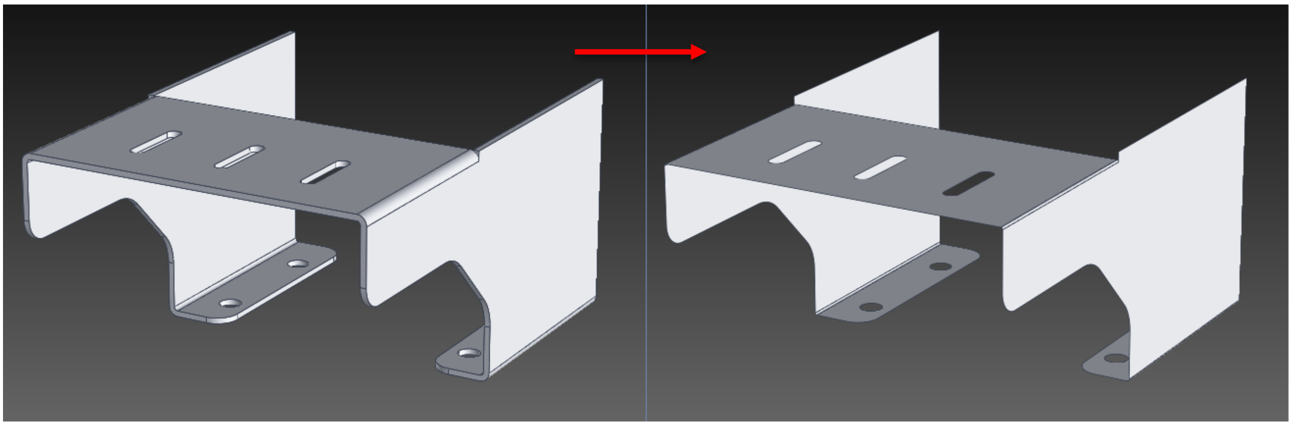
Extrude Plate Edge Attributes

When extruding plates into bricks, a new option is available to map existing plate edge attributes into equivalent brick face attributes. For example, plate edge pressure attributes become brick face pressure attributes. Where the resulting brick property does not yet exist, it is automatically created from property data of the source plate element.



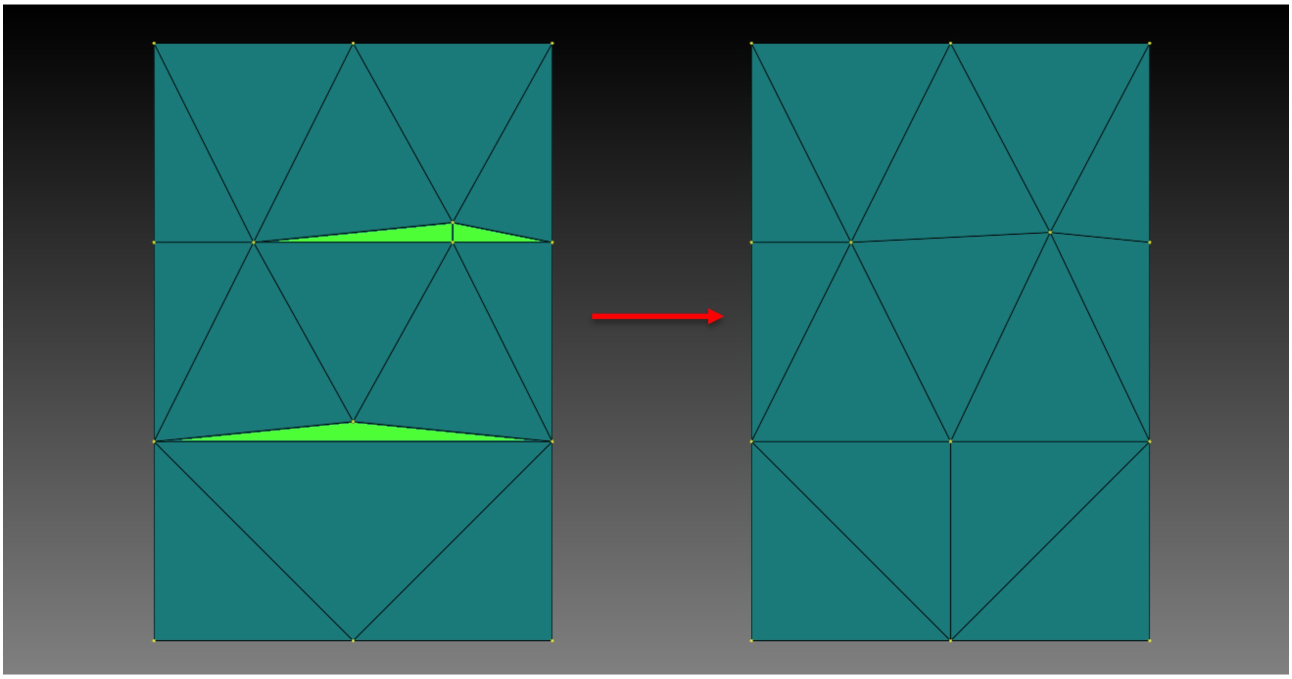
Mid-plane Thin Solids

The tool generates the mid-plane faces of selected geometries that resemble thin solids, and assigns a thickness attribute to the newly generated mid-plane faces. Selected geometry faces that are separated by a relatively small gap are regarded as valid thin solids. Each solid must be defined in a separate group, which is typically already the case if the solid geometries are imported from a well-defined CAD model. The selected faces do not need to form a closed solid manifold; the tool can still extract mid-plane surfaces in many cases. Not all thin solids can be mid-planed by this tool, however.



Repair Tri3 Mesh

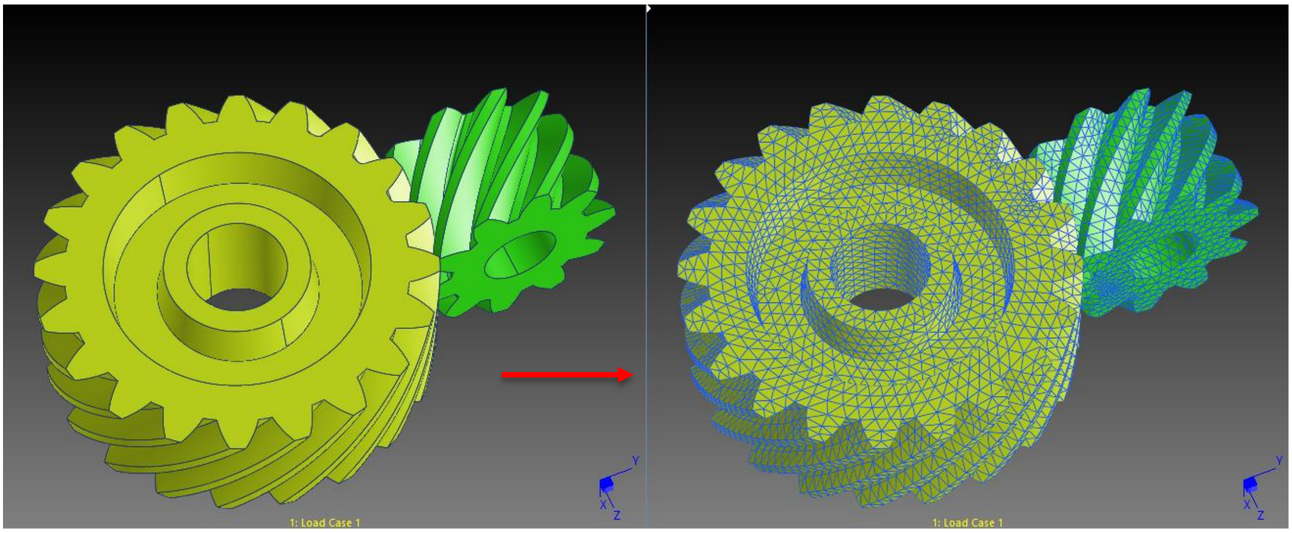
Specifically intended for triangular meshes typically imported from STL files, the tool provides additional functionality to repair such meshes, beyond what can be done using the mesh cleaning tools.



Direct Solid Mesh

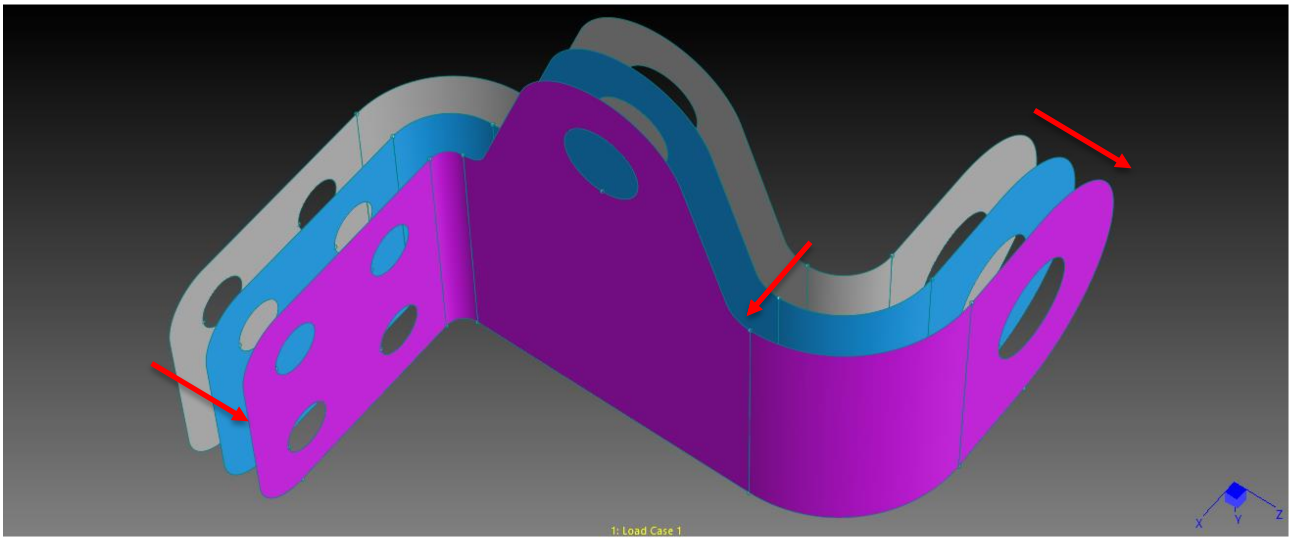
The tool automatically generates a solid mesh of tetrahedral brick elements from geometry face boundaries defining the surfaces of closed volumes. This function consolidates the two-step process of **Surface Automesh** followed by **Solid Automesh from Plates** to generate brick elements directly from geometry faces. This can speed up the process of solid meshing, and it can also avoid problems that may occur with the two-step process when plate elements are inadvertently detached or zipped, particularly plate elements representing adjoining faces of separate solids.

The geometry faces must define a watertight (i.e., manifold) surface for each solid. Geometry faces containing free edges and/or T-junctions cannot be used for direct solid meshing. Multiple solids can be meshed at once and they can all be in the same group. Internal cavities are detected and can be treated as cavities or as separate solids.



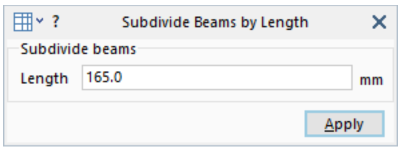
Geometry Copy or Move by Thickness

This tool uses the face normals to define the direction in which the face will be copied or moved. The tool is also useful for mid-planing geometry that cannot be mid-planed automatically.



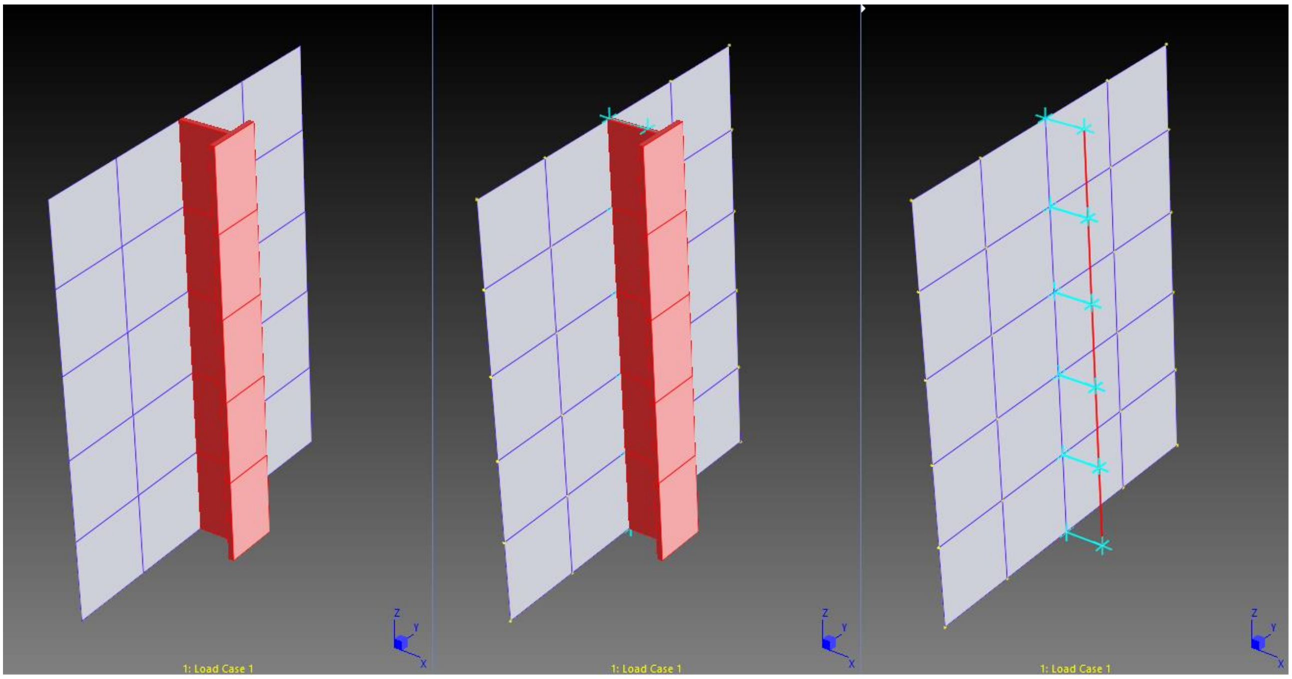
Subdivide Beams by Length

For models containing beam elements of different lengths that need to be refined into approximately equal length elements, this tool can be used. It avoids having to select beams of different length separately to subdivide independently.



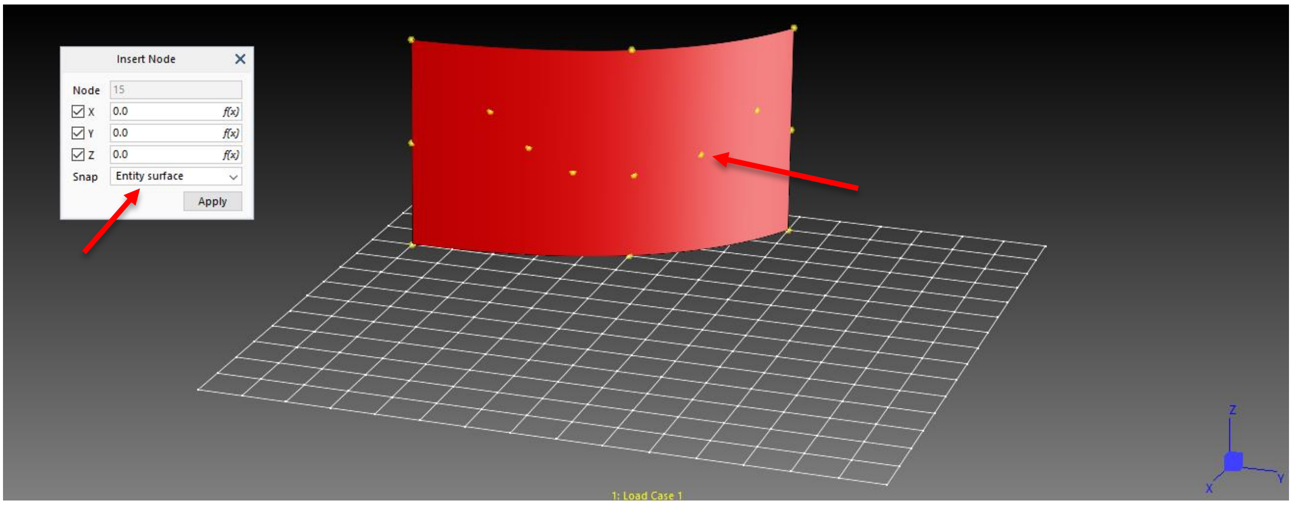
Convert Beam Offsets to Rigid Links

In a geometry linear analysis, a beam offset applied to a beam element produces identical results to physically offsetting the beam elements and attaching them with rigid links. For geometrically nonlinear analysis, the results are virtually identical for small to medium levels of deformation. For very large levels of deformation, convergence can sometimes be easier when explicit rigid links are used in place of the beam offset attributes. For those situations, this tool can be used to automatically detach the beams, offset their nodes and reattach them via rigid links.



Insert Node on Entity

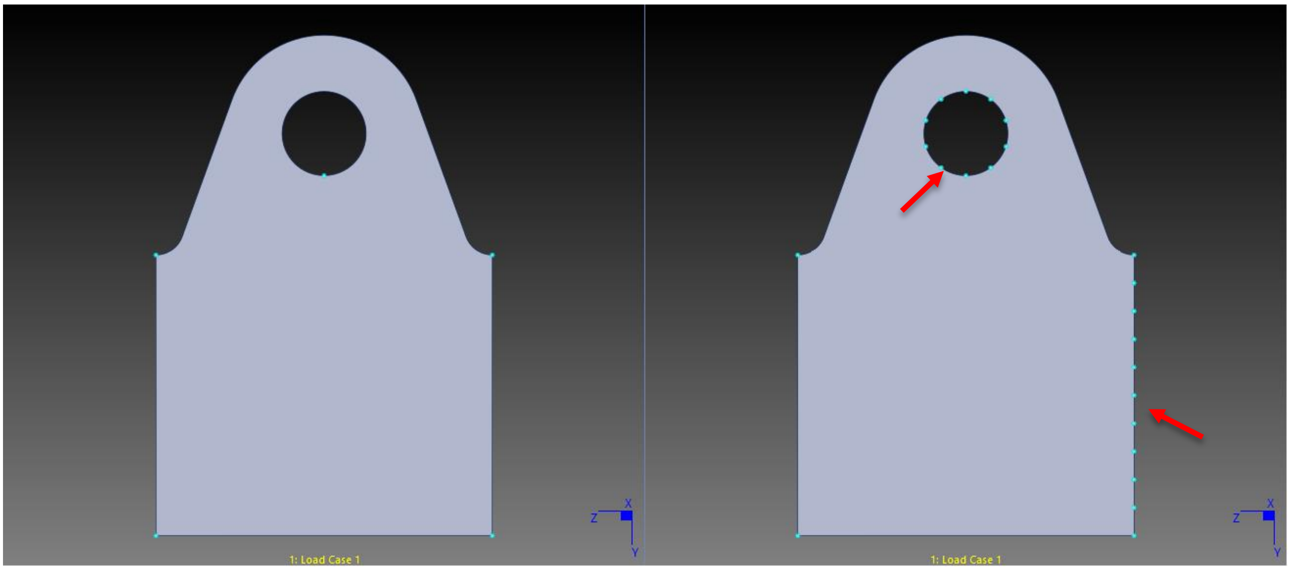
The function **VISUAL/Insert/Node** can be used to insert a node into a model via a range of options, such as by defining the coordinates or by clicking a grid point. The **Snap: Entity surface** mode enables the insertion of a node by clicking directly onto the surface of an entity, such as a geometry face or element. The new node will be generated by transforming the clicked pixel location on the screen to model coordinates and projecting the point onto the clicked entity surface.



This tool also offers an **Edit** option that allows for an existing node to be moved from its current location onto a grid point, a vertex or another node: click to pick up the source node and click to drop the node onto the other entity.

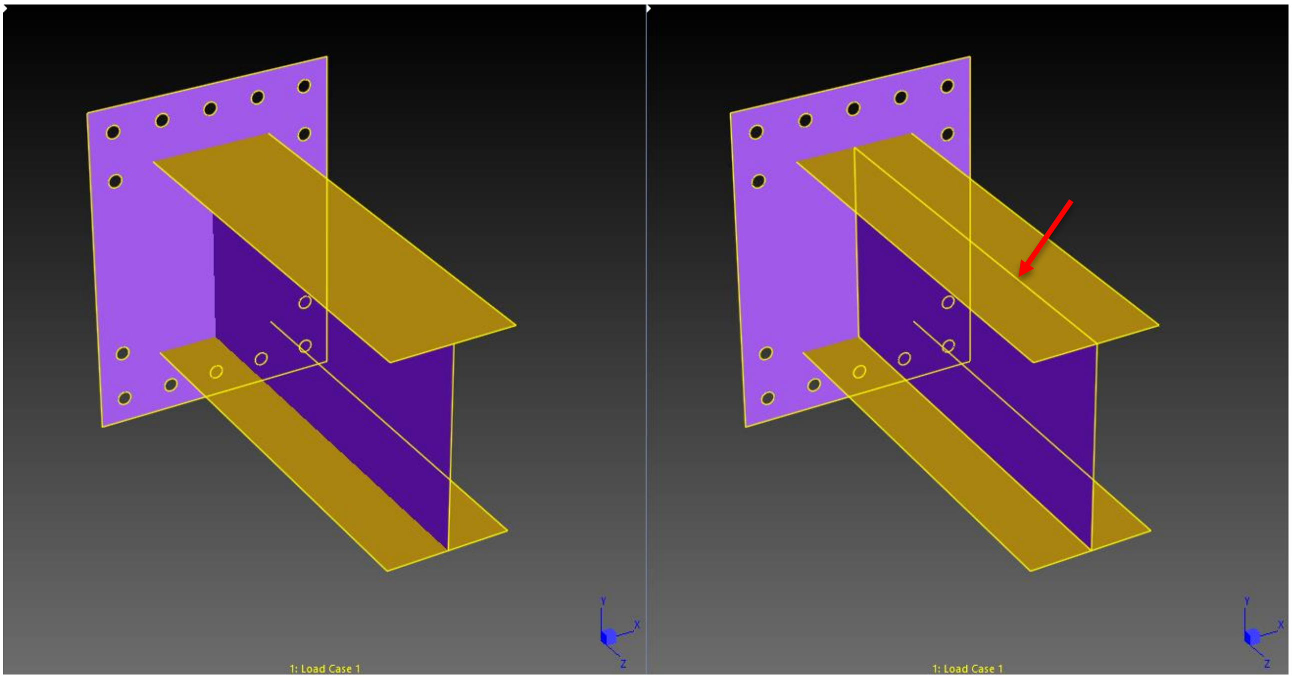
Geometry Subdivide Edges

This new tool effectively splits a geometry edge by inserting either fixed or free vertices at equally spaced points along the edge. This offers another way to dictate the mesh density generated along an edge.



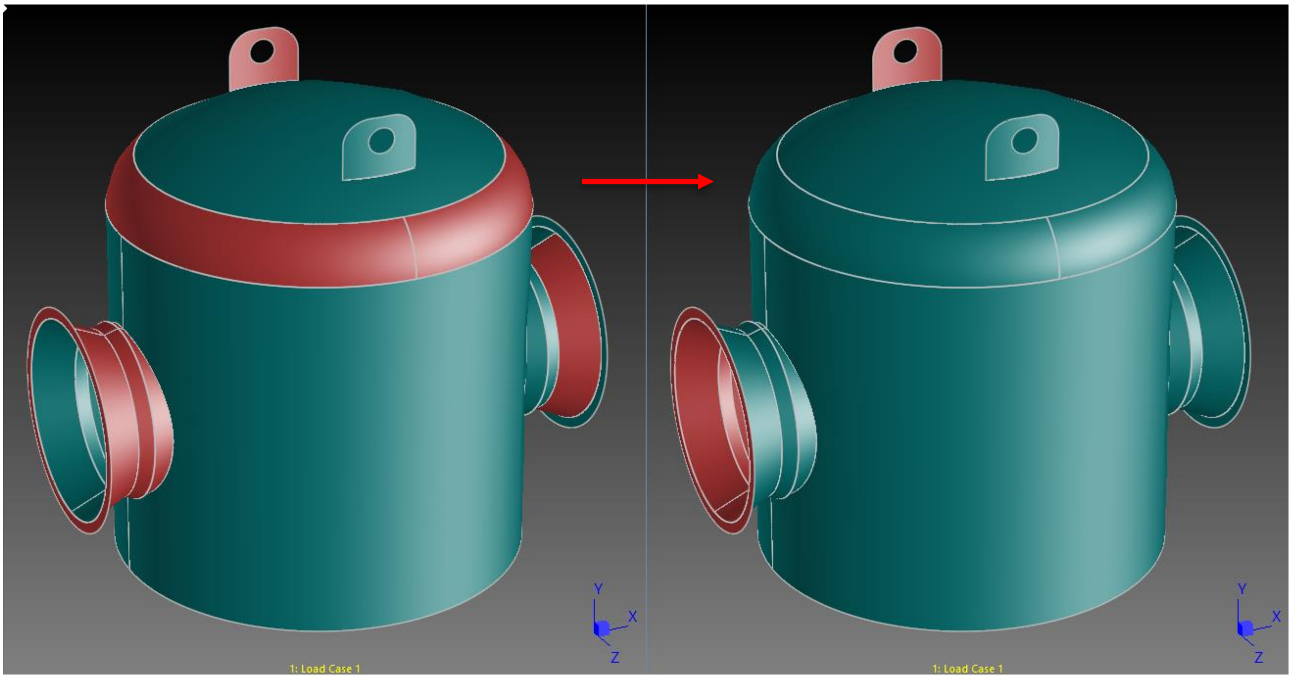
Geometry Detach Faces

Similarly to the **Detach Elements** tool, the **Detach Faces** tool can be used to disconnect a face or a collection of faces from their co-edges. This feature can be useful in situations where a geometry clean operation has used a tolerance that is too large, and it is desired to re-attach the faces with a smaller tolerance or to perform other operations on the geometry.



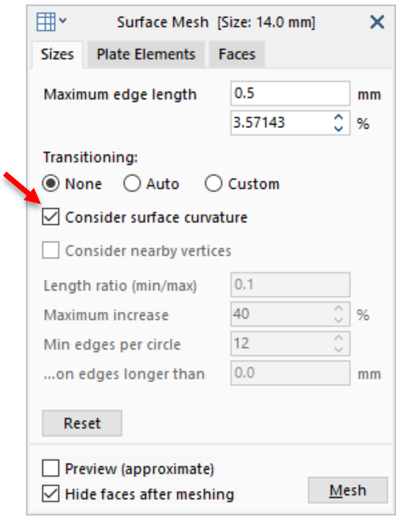
Align Face Normals

Similarly to the alignment of the normals of connected plates, the Align Face Normals tool enables the automatic alignment of faces to produce adjoining faces with consistent normals.

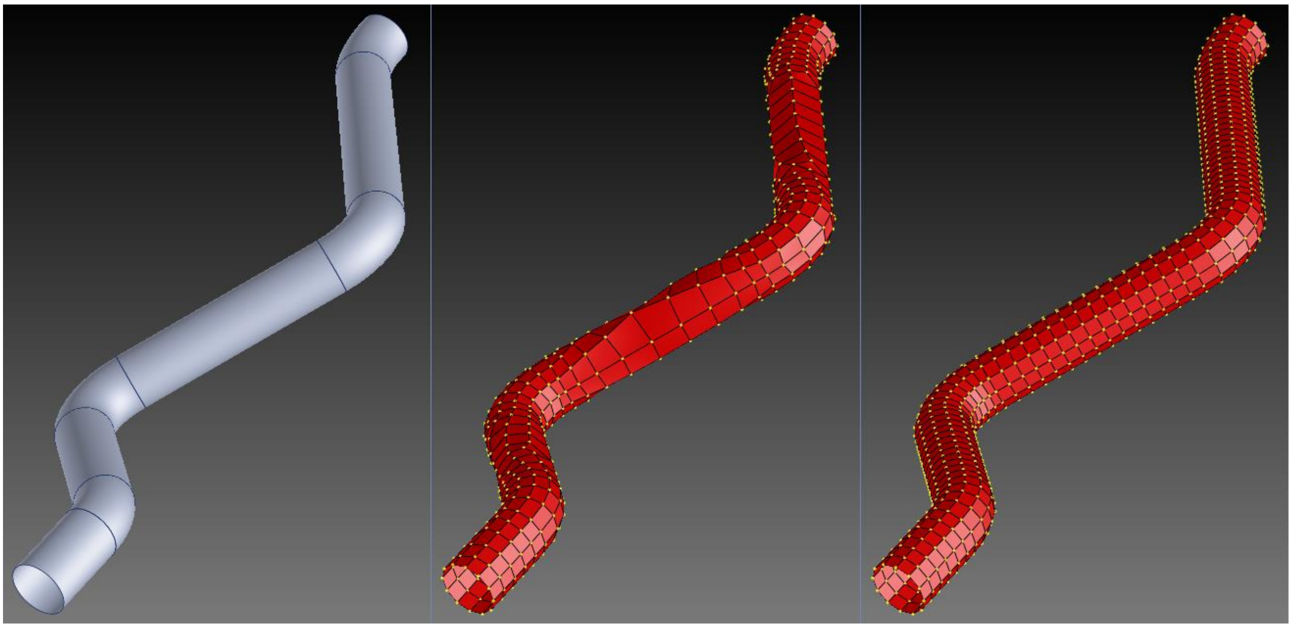


Surface Mesher

The surface automesher will provide significantly better runtime performance compared with R24, as well as generally improved mesh quality. In addition, the **Consider surface curvature** option can be used to automatically refine mesh sizes on faces based on their radius of curvature. This feature is similar to the automatic refinement of edges based on edge curvature, but applies to the whole face.



The following image compares the plate mesh produced with and without this option for the same nominal mesh size.

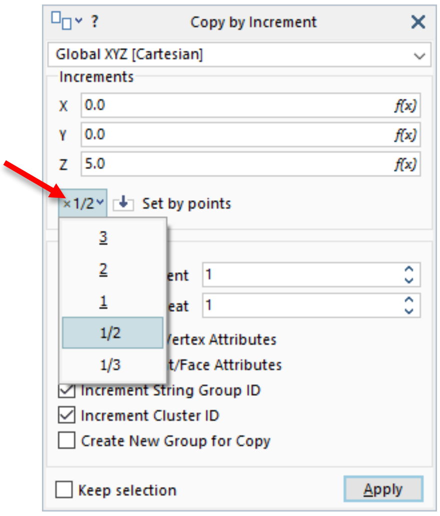


Geometry | Consider surface curvature = NO | Consider surface curvature = YES

Tools – Productivity Features

Increment Multiplier

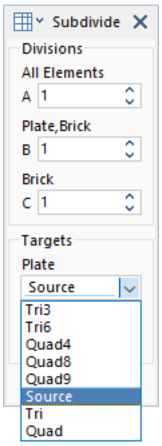
The tool dialog offers a useful feature that automatically scales the increments retrieved from the model window via the **Set by points** function. For example, to copy an entity by half a thickness, set the scaling factor to ½ and retrieve the full thickness distance using **Set by points**.



Subdivide Source / Tri / Quad

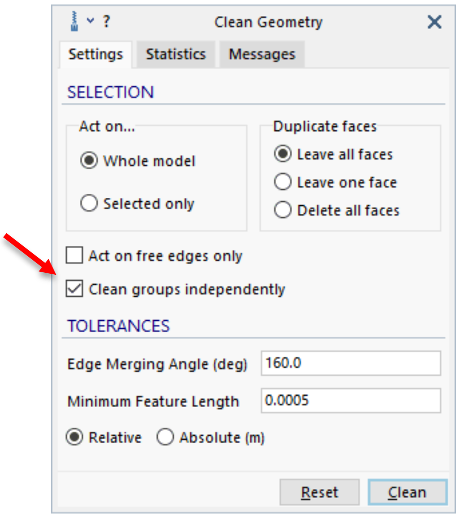
For meshes consisting of elements with different orders (e.g., Quad4 and Quad8) the **Source** option allows for the subdivision of elements according to the source type, thereby avoiding the need to perform the operation twice (once on the Quad4 elements and once on the Quad8 elements).

The **Tri** and **Quad** options are used to subdivide elements into triangular or quadrilateral elements based on the source (i.e., a Quad4 becomes a Tri3 whereas a Quad8 becomes a Tri6).



Clean Geometry – Clean Groups Independently

This new option prevents faces or bodies belonging to different groups from being zipped together during a geometry cleaning operation.



Solvers and Related

The R3 solver is a substantially new development, and not just a 64-bit version of the R24 solver. Consequently, the two solvers are not expected to produce numerically identical results in most cases. Whilst some of the underlying element formulations in R3 are the same as those in R24, the implementation is different.

Cable Inertia Case

If the model contains cable elements, the cable inertia case dropdown list becomes available for the linear solvers. All load cases in the model will be listed; one case is selected as the inertia case. The selected case is used to define the catenary, and hence stiffness matrix, of the cables for assembly into the global stiffness matrix.

This is required because the stiffness of a cable element depends on its catenary shape, which is determined by the applied inertia loads acting on the mass of the cable, together with the stiffness of the structure to which the cable ends are attached.

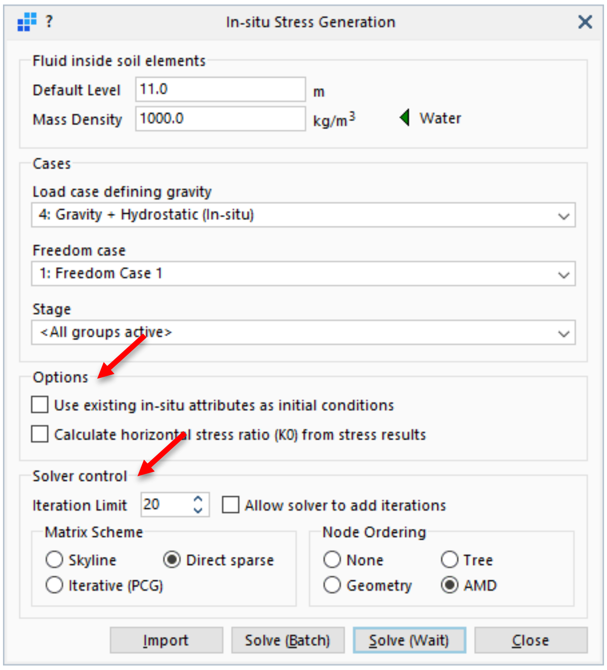
In reality, the cable element is a geometrically nonlinear element, and solving for this nonlinearity becomes part of any geometric nonlinear analysis, so no special considerations are required in those solvers. But if geometric nonlinearity is not considered because the analysis in a linear one, a cable stiffness has to be calculated by using some assumptions. In R3 the assumptions are that the cable shape (catenary) is set by the inertia load applied in the selected load case, based on the assumed cable free length and mass. The cable pre-load attributes in the selected load case will affect the cable free length.

The different linear solvers will offer different options for the cable inertia, while the linear static solver also offers the option to **Form stiffness matrix for every load case**. If the option is set, each load case is separately solved. In each case, the inertia load specified in the load case is used to define the stiffness matrix of the cable elements. This option is mainly useful if the load cases contain different inertia loads. In the more common case of one load case being used to apply the inertia load (e.g., gravity load) and other load cases being used to apply other load types, the cable stiffness should be calculated based on the one inertia load case.

Soil In-situ Options

The soil in-situ stress tool, found under **VISUAL/Utility/Soil In-situ**, offers the following new features.

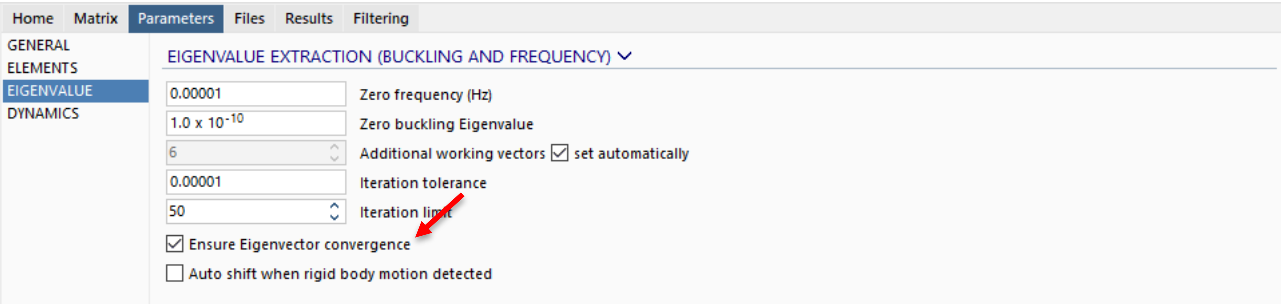
- Two run modes are provided, the **Wait** mode and the **Batch** mode. The Wait mode is as in R24 whereby the solver is executed while the GUI is paused until completion. The Batch mode launches the solver as a separate process with the GUI immediately becoming responsive. At the completion of a Batch mode execution, the resulting in-situ stresses can be imported by reading the relevant ISA file via the **Import** button on the dialog.
- The **Calculate horizontal stress ratio (K0) from stress results** option is available to assign the average lateral stress by the calculation of K0.
- The **Solver control** group offers direct control of the solver iteration parameters for the in-situ calculation.



Eigenvector Convergence for Natural Frequency and Linear Buckling Analysis

The Eigenvalue solver in R3 is used to calculate natural frequency and linear buckling modes of a structure. This calculation is an iterative one involving a convergence tolerance. In R24, the convergence tolerance is applied only to the Eigenvalue (e.g., the frequency); when the Eigenvalue is converged, the corresponding Eigenvector (i.e., the mode shape) is assumed to have also converged. In R3, convergence of the Eigenvectors can be requested in addition to convergence of the Eigenvalue, by setting the solver option **Ensure Eigenvector convergence**.

Eigenvector convergence is particularly important when performing modal superposition analysis such as spectral response analysis.



Mode Participation in Natural Frequency Analysis

In addition to the previously available **Mode Participation Mass Ratio** for mode *i*, m_i, given by the following:

$$m_i = \frac{(\varphi_i^T M d)^2}{d^T M d}$$

where d is a direction vector (the excitation direction), φ_i is the mode shape vector for mode *i*, and M is the mass matrix, a new quantity, the **Mode Participation Factor** for mode *i*, f_i, is given by the following:

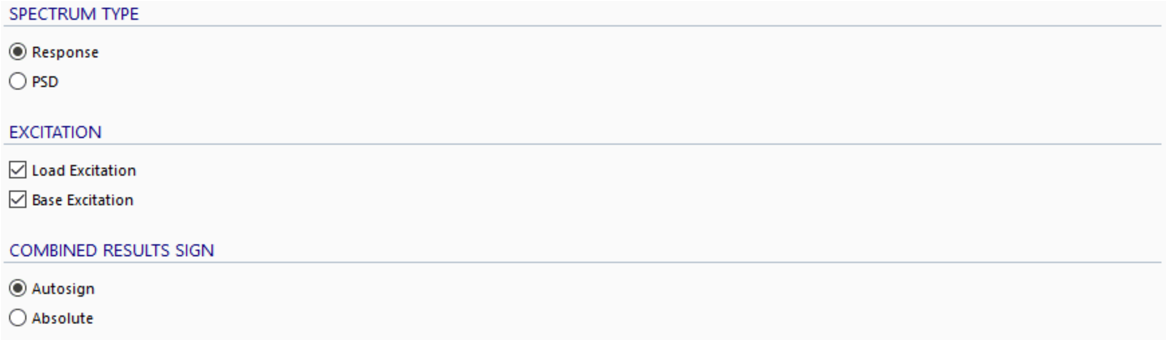
$$f_i = \frac{\varphi_i^T M d}{\varphi^T M \varphi} = \frac{\varphi_i^T M d}{1} = \varphi_i^T M d$$

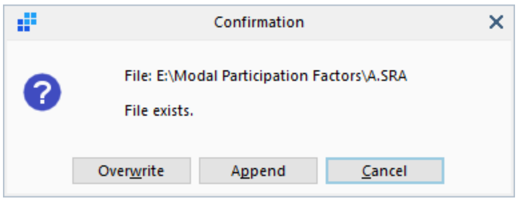
These are tabulated in the solver log file as illustrated in the log file excerpt below.

```
MODE PARTICIPATION FACTORS FOR TRANSLATIONAL EXCITATION
 Mode      Frequency         P-DX          P-DY          P-DZ          P-RX          P-RY          P-RZ
              (Hz)        SQRT(N/mm)    SQRT(N/mm)    SQRT(N/mm)    SQRT(N.mm)    SQRT(N.mm)    SQRT(N.mm)
    1   1.466385E+00  1.349967E+01 -3.509512E-02  2.096231E+01  3.118482E+05  1.338942E+04 -2.019440E+05
    2   2.061489E+00  2.518423E+01  4.155024E-01 -1.565447E+01 -2.463401E+05  3.193650E+05 -3.913983E+05
    3   2.789282E+00  7.869350E+00 -1.651454E-01  1.451089E+01  2.480814E+05 -2.798480E+05 -1.411296E+05
    4   4.510150E+00  5.635333E+00  5.257864E-02  9.112563E+00  1.333854E+04 -2.681165E+04 -8.362777E+03
    5   7.475686E+00  2.500738E+00 -2.103694E-01  2.542251E+00  1.082266E+04  3.117561E+04 -9.645800E+03
    6   7.844652E+00  1.201965E+01 -1.978382E+00 -8.393168E+00 -1.023215E+03  1.669594E+05 -4.753503E+04

 TOTAL MASS (MX,MY,MZ) (t)     : (1.207507E+03,1.206685E+03,1.207507E+03)

 MODE PARTICIPATION MASS RATIOS FOR TRANSLATIONAL EXCITATION
 Mode      Frequency     Modal Mass    Modal Stiff       P-DX          P-DY          P-DZ          P-DX     P-DY     P-DZ
              (Hz)          (t)          (N/mm)          (%)           (%)           (%)           (%)      (%)      (%)
    1   1.466385E+00  2.651808E+02  2.251117E+04  1.509234E+01  1.020704E-04  3.639055E+01     15.092    0.000   36.391
    2   2.061489E+00  5.826701E+02  9.775622E+04  5.252521E+01  1.430715E-02  2.029491E+01     52.525    0.014   20.295
    3   2.789282E+00  1.968148E+02  6.045086E+04  5.128474E+00  2.260161E-03  1.743807E+01      5.128    0.002   17.438
    4   4.510150E+00  2.490158E+02  1.999717E+05  2.629963E+00  2.290999E-04  6.876882E+00      2.630    0.000    6.877
    5   7.475686E+00  2.519965E+02  5.559765E+05  5.179012E-01  3.667511E-03  5.352383E-01      0.518    0.004    0.535
    6   7.844652E+00  7.633347E+02  1.854480E+06  1.196449E+01  3.243594E-01  5.833944E+00     11.964    0.324    5.834
 ----------------------------------------------------------------------------------------    -------------------------
 TOTAL TRANSLATIONAL MASS PARTICIPATION (%)       8.785838E+01  3.449254E-01  8.736960E+01     87.858    0.345   87.370

 MODE PARTICIPATION MASS RATIOS FOR ROTATIONAL EXCITATION
 Mode      Frequency     Modal Mass    Modal Stiff       P-RX          P-RY          P-RZ          P-RX     P-RY     P-RZ
              (Hz)        (t.mm^2)       (N.mm)          (%)           (%)           (%)           (%)      (%)      (%)
    1   1.466385E+00                              3.377146E+01  7.311858E-02  1.047697E+01     33.771    0.073   10.477
    2   2.061489E+00                              2.107335E+01  4.159862E+01  3.935606E+01     21.073   41.599   39.356
    3   2.789282E+00                              2.137233E+01  3.194102E+01  5.116935E+00     21.372   31.941    5.117
    4   4.510150E+00                              6.178456E-02  2.931916E-01  1.796696E-02      0.062    0.293    0.018
    5   7.475686E+00                              4.067535E-02  3.964004E-01  2.390288E-02      0.041    0.396    0.024
    6   7.844652E+00                              3.635775E-04  1.136909E+01  5.804985E-01      0.000   11.369    0.580
 ----------------------------------------------------------------------------------------    -------------------------
 TOTAL ROTATIONAL MASS PARTICIPATION (%)          7.631996E+01  8.567144E+01  5.557233E+01     76.320   85.671   55.572
```

Spectral Response Analysis Multi-case and Append Options

The spectral response solver in R3 allows for both Load Excitation and Base Excitation to be solved together with one execution of the solver, with results for both excitation types stored in the same result file. In addition, the append option enables the appending of different spectral response solutions into the same result file. This could be useful, for example, to compare the spectral results of a structure with different restraint conditions – as each restraint condition requires a separate natural frequency analysis to determine the mode shapes, a separate spectral response analysis is therefore also required for each natural frequency analysis.



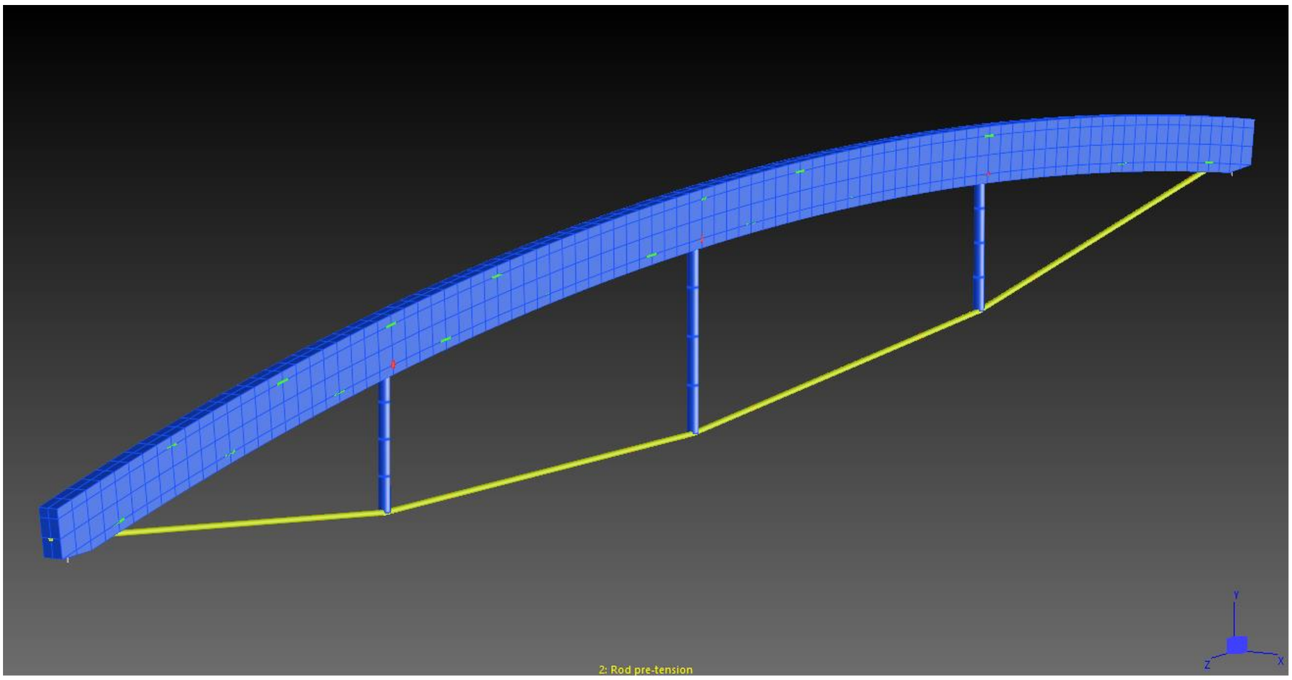


Linear Buckling Analysis – Constant + Variable Case

The linear buckling solver calculates factors applied to a load case to cause the structure to buckle. The load case is referred as the **Variable Case** in R3 and the buckling factors scale the loads in that load case.

R3 also offers the option to select a **Constant Case** together with the variable case. If a constant case is selected, that result case is assumed to be applied to the structure with a constant load factor of 1.0. The linear buckling solver then calculates the load factor that scales only the variable case to cause the structure to buckle while the load factor on the constant case remains as 1.0.

An example application of this feature is illustrated in the tied-arch model shown below. The roof load required to cause the structure to buckle is to be determined given that the structure already has pre-tension in the ties and self-weight due to gravity. The buckling factors to be calculated should not scale the gravity or pre-tension loads, only the roof load.



The model is set up with three load cases: gravity, pre-tension and roof load. A linear static analysis calculates the results for these load cases. A combination case is defined to combine gravity and pre-tension into one result case. The linear buckling analysis then selects the combination case as the **Constant Case** and the roof load case as the **Variable Case**. The buckling factors calculated scale the roof load while the other loads are applied unscaled.

INITIAL CONDITIONS

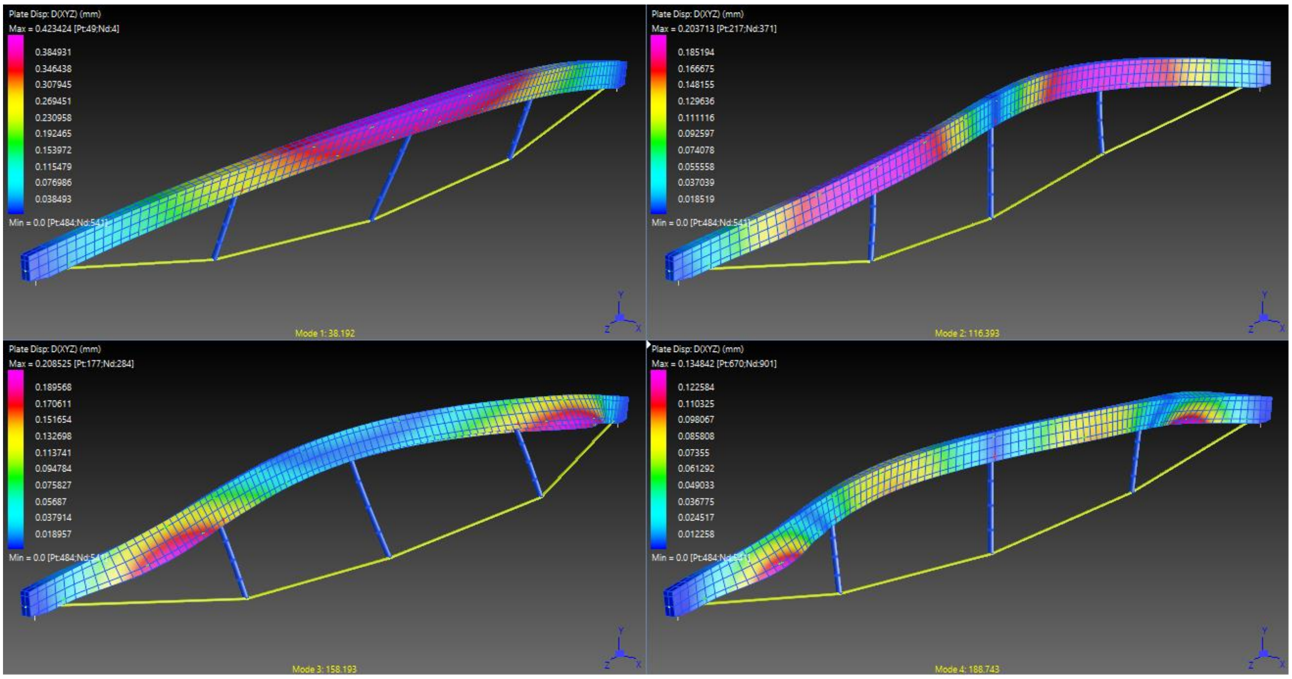
Initial Conditions File
E:\Buckling\Buckling of Tied Arch Roof.LSA

Variable Case
3: Imposed roof load

Constant Case (Load Factor=1.0)
5: [1: 1.2 G + Pre-stressing] [Combination]

☐ Include follower load [KG] (local distributed lateral loads only)

Warning: The constant case should be checked separately to ensure it does not cause buckling by itself.



Linear Buckling and Natural Frequency Follower Load [KG]

A new option is available in the linear buckling and natural frequency solvers: **Include follower load [KG]**.

INITIAL CONDITIONS

Initial Conditions File
C:\St7 Models\Ring with Plate Edge Follower Load Kg.LSA

Variable Case
1: Load Case 1

Constant Case (Load Factor=1.0)
<None>

☑ Include follower load [KG] (local distributed lateral loads only)

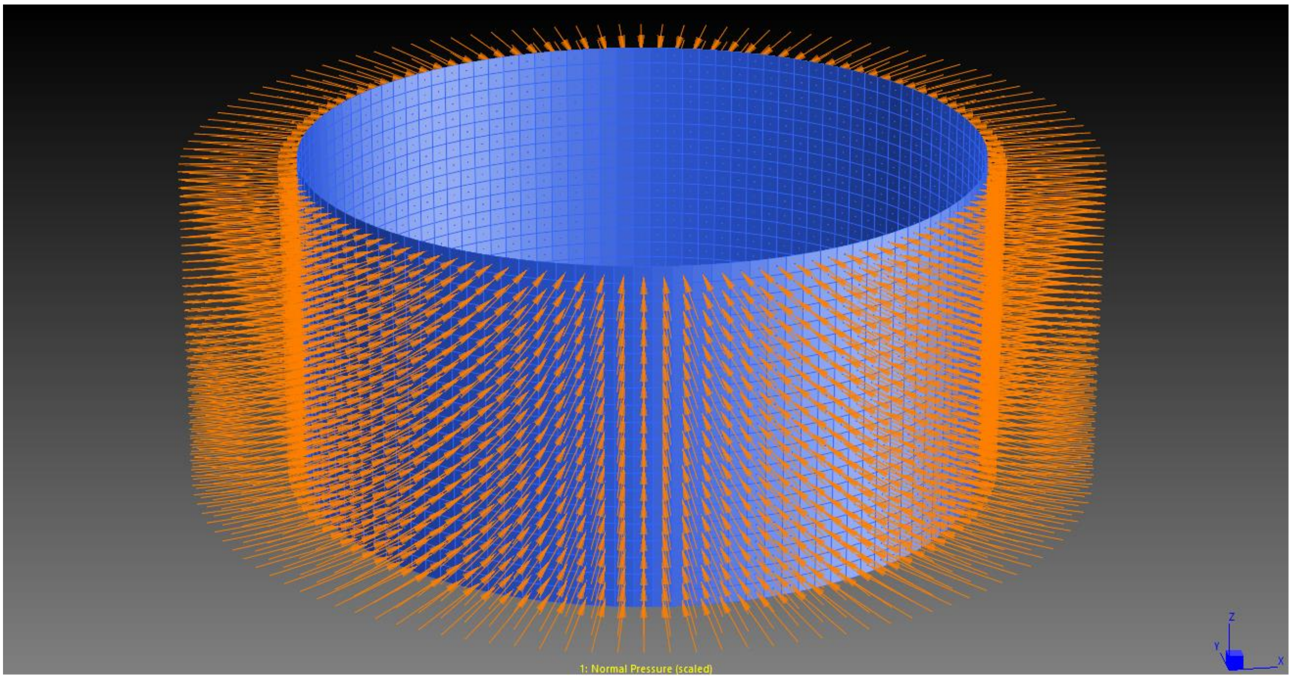
If the option is set, the geometric stress stiffness matrix used by the linear buckling solver, and optionally used by the natural frequency solver, will be augmented by an additional load stiffness matrix that depends on the change in tangent stiffness that occurs when an applied load changes direction as the structure deforms. Only some load types will change direction as the structure deforms; these are the so-called follower loads, which in Strand7 include the following element attributes:

- Beam Distributed Force applied in the principal lateral directions of Beam, Truss, Cutoff Bar and Spring elements;
- Plate Edge Normal Pressure applied to Plate/Shell, 3D-Membrane, Plane Stress, Plane Strain and Axisymmetric elements;
- Plate Face Normal Pressure applied to Plate/Shell and 3D-Membrane elements;
- Brick Face Normal Pressure (all brick elements).

A Plate Global Face Pressure is not a follower load because its direction does not change as the plate element deflects or rotates.

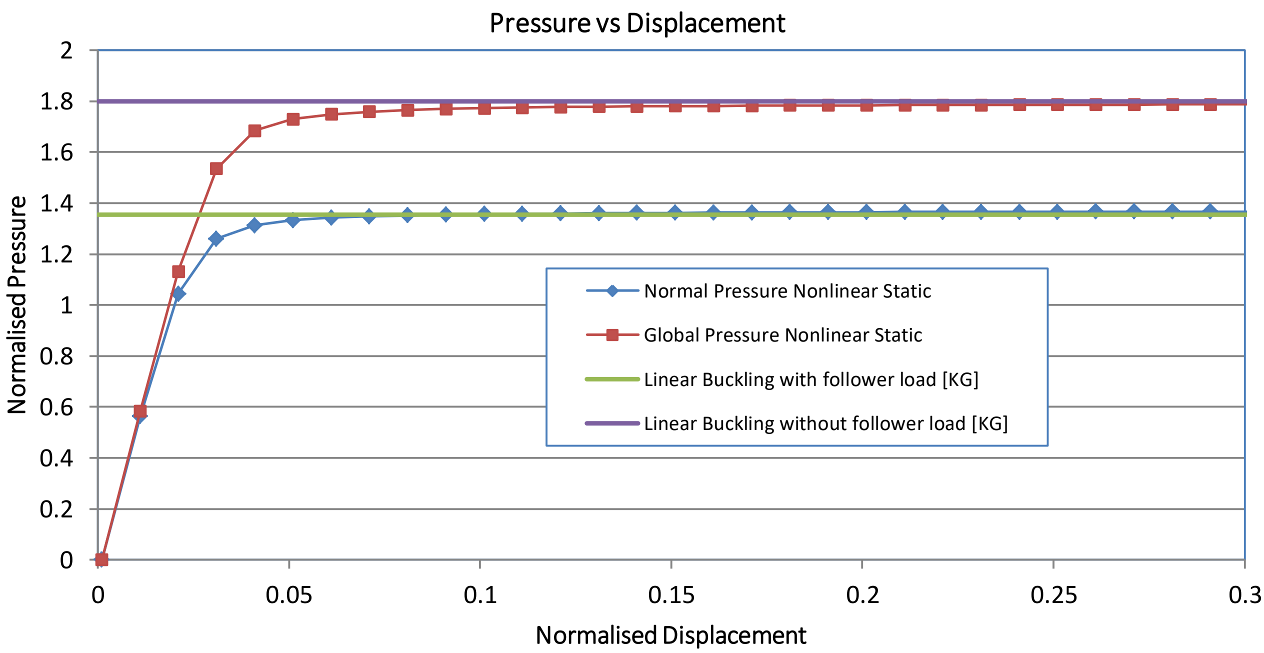
The following simple example illustrates the use of this feature in a linear buckling analysis.

A cylinder with a uniform external pressure is analysed using the nonlinear static solver considering geometric nonlinearity. The pressure is increased beyond the initial buckling response. The analysis is conducted twice: firstly, with the external pressure applied as plate normal pressure attributes, and secondly, with the external pressure applied as plate global pressure attributes.



In addition to the nonlinear static analysis, two linear buckling analyses are conducted using the load case with normal pressure attributes: one analysis includes the follower load [KG] whilst the other does not.

The following graph compares all four solutions. It is clear that the buckling analysis with follower load [KG] closely resembles the geometric nonlinear analysis with the normal pressure (i.e., follower load) attributes. Similarly, the buckling analysis without the follower load [KG] closely resembles the geometric nonlinear analysis with the global pressure attributes.



Note that in general, the load stiffness matrix is non-symmetric. However, a symmetric approximation has to be included in linear Eigenvalue solvers that work only with symmetric matrices. The accuracy and applicability of this option should be checked with the GNL solver as in the example above.

Position Offset vs Time Tables – Quasi-static and Nonlinear Transient Dynamic Analysis

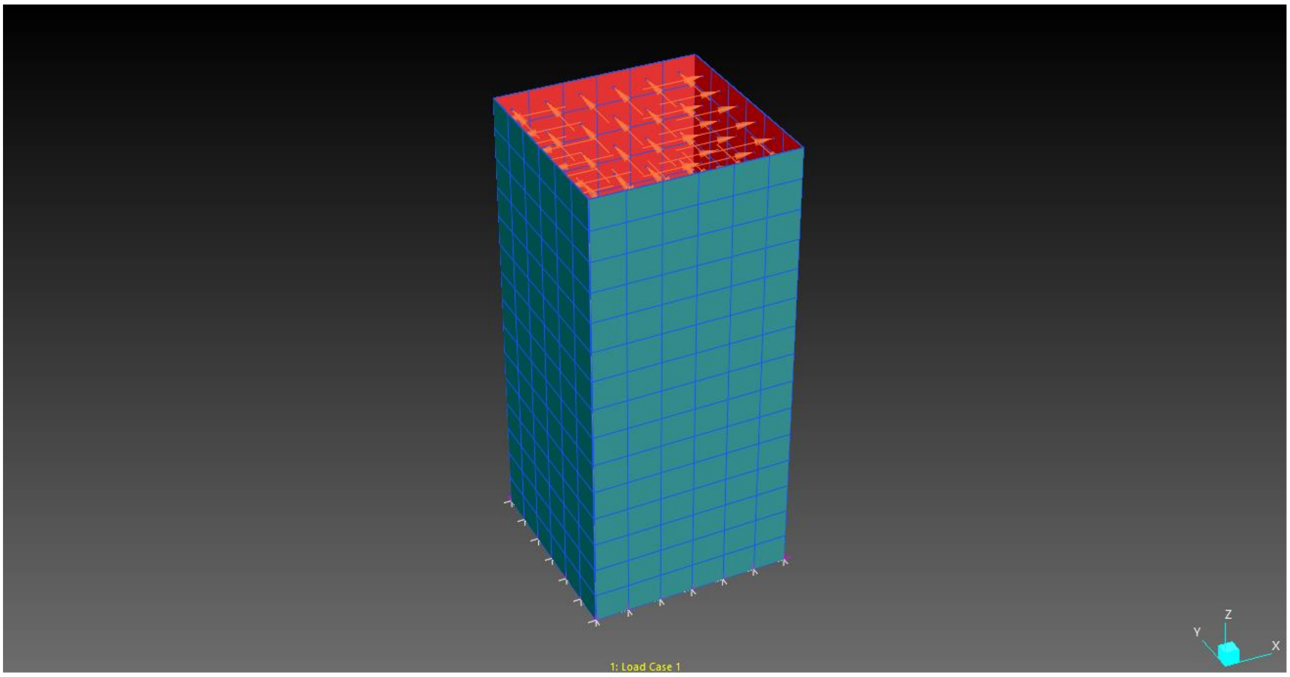
As in R24, a Factor vs Position table can be assigned to a load case to scale the load at a node (irrespective of the source of the load), based on the current position of the node. As the position of the node changes during the analysis, the applied load is scaled according to the table. For example, a buoyancy force on a ship hull can be modelled by associating a Factor vs Position table with a depth direction – as the ship moves downward, the hydrostatic pressure increases.

Another example might be the filling of a tank with water. This process could be modelled with just a Factor vs Position table by physically translating the tank relative to the defined table using enforced displacements. A more convenient approach is to define the relative movement between the structure and the Factor vs Position table via a separate table, thereby avoiding the need to physically move the structure in the analysis.

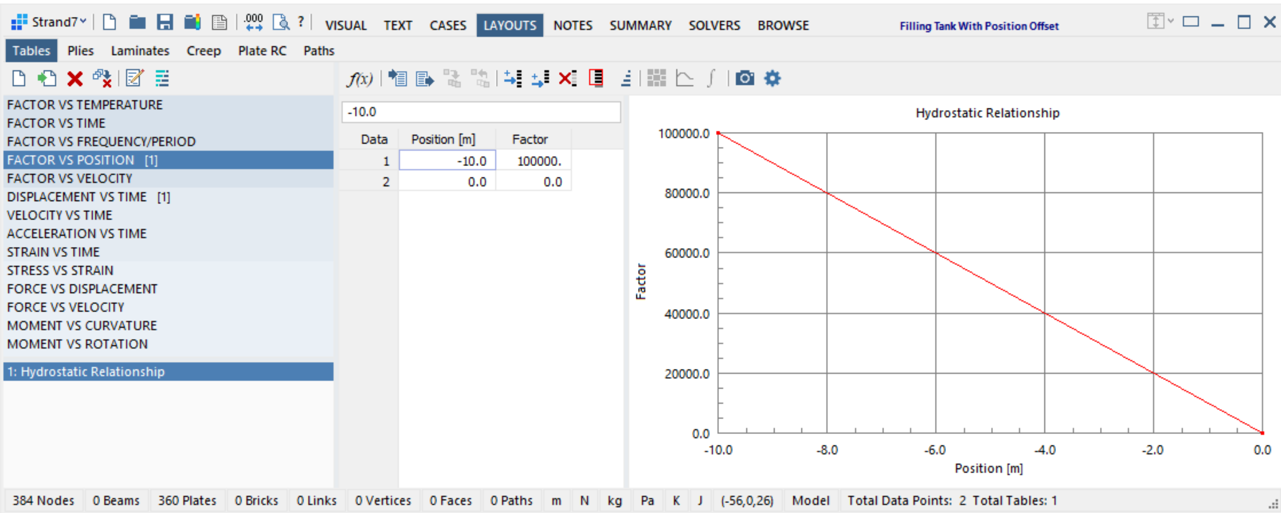
A new table option for the quasi-static and nonlinear transient dynamic solvers provides this functionality in R3. The table type is one of the new tables in R3, the Displacement vs Time table, which is selected as the Position Offset vs Time table in the Load tab of the solvers.

Example: Filling a tank with water

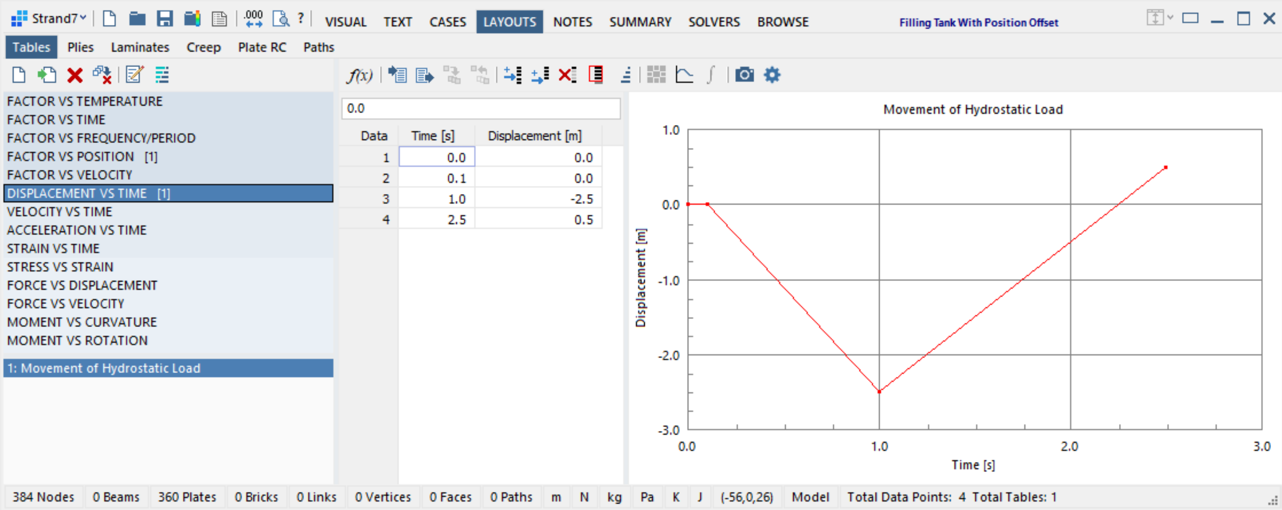
The mesh below represents the tank. There is a load case with a uniform (unit) internal pressure applied.



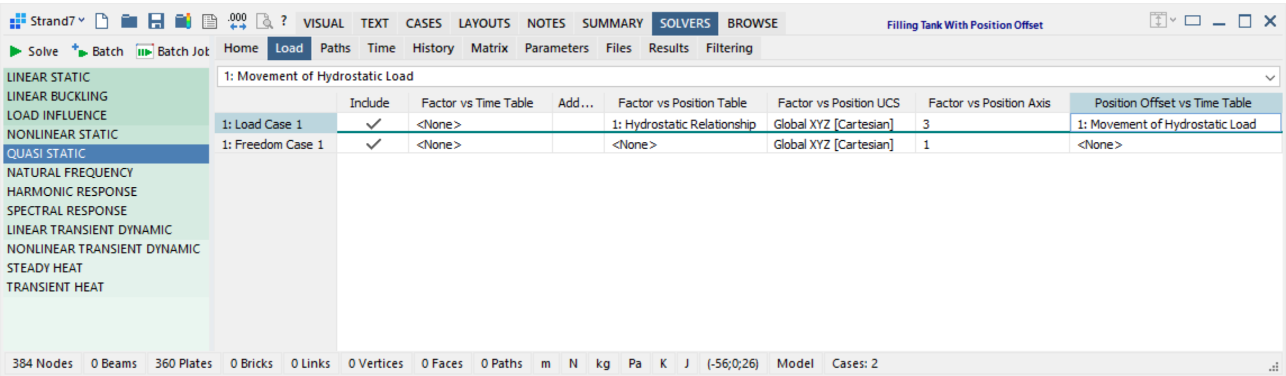
A Factor vs Position table is defined to represent water pressure with depth.



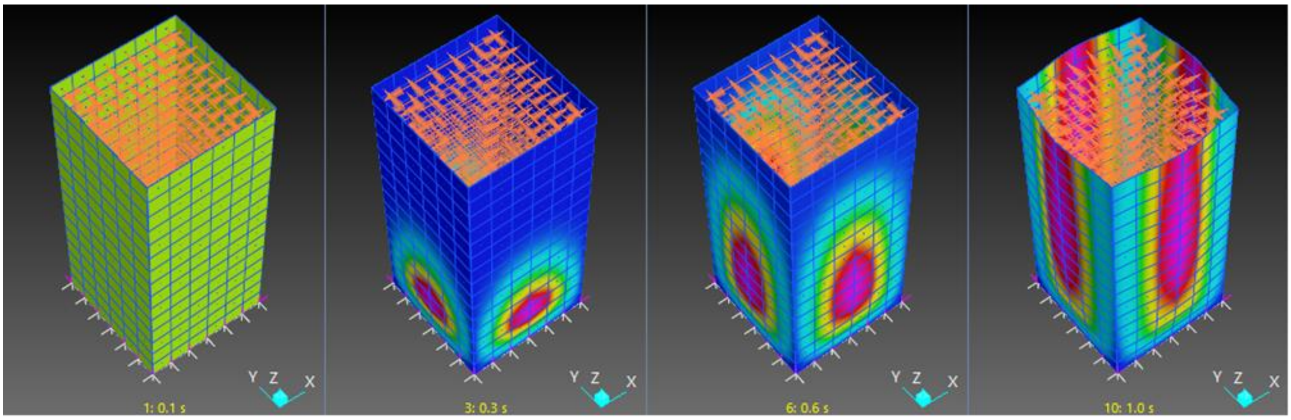
A Displacement vs Time table represents the relative movement of the tank with respect to the Factor vs Position table. This simulates filling over one second and then emptying.

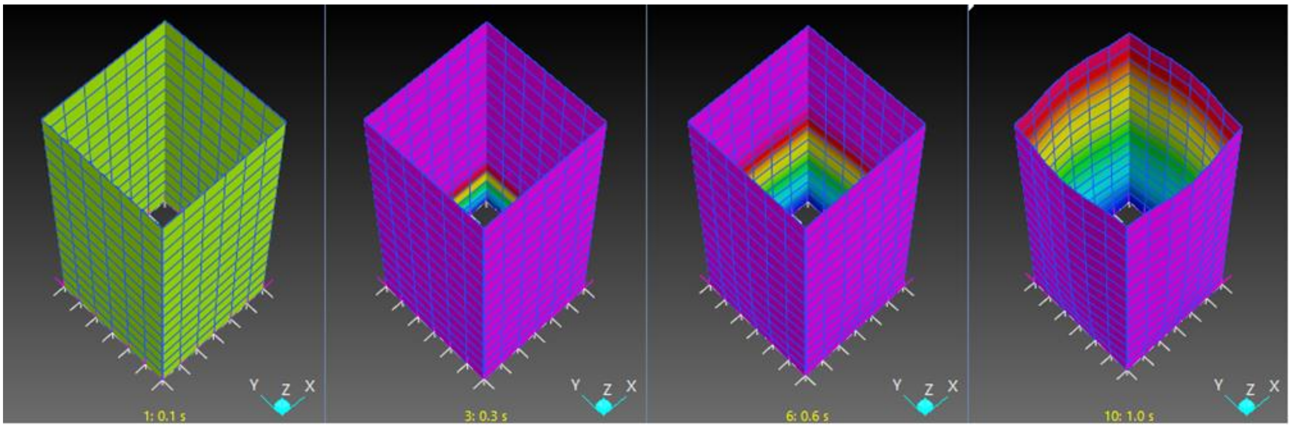


The **Load** sub-tab of the solver is shown below with the tables selected.



Displacement contours at various steps of the filling process are shown below. A plot of the local zz stress shows the corresponding normal pressure on the inside surface as the tank is filled.





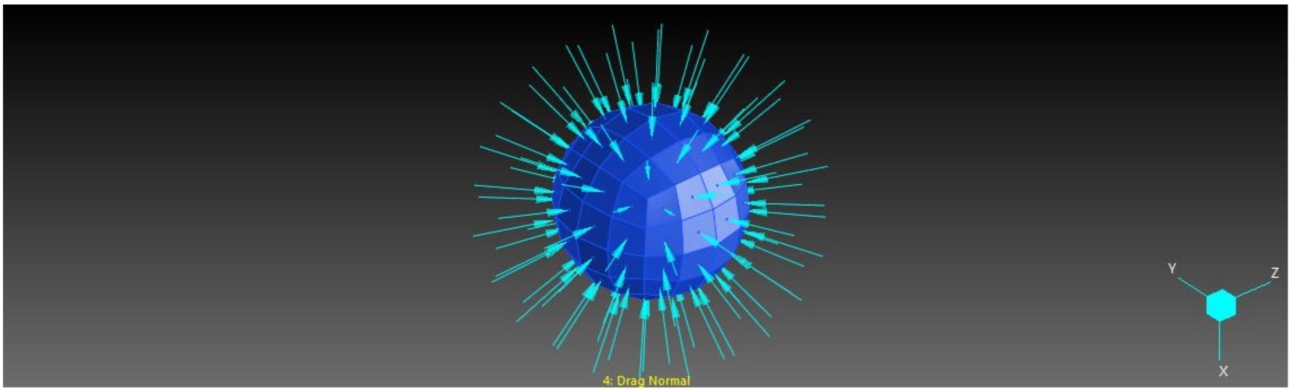
Factor vs Velocity Tables – Nonlinear Transient Dynamic Analysis

A new table option for the nonlinear transient dynamic solver in R3 provides the functionality to scale the load at a node based on the current velocity of the node. Unlike Factor vs Position tables, which apply scaling factors based on the nominated UCS and axis, Factor vs Velocity tables scale the component of the load parallel to the direction of the total velocity of the node, or parallel to the velocity components of the node.

The functionality applies to all loads on elements, except for thermal loads.

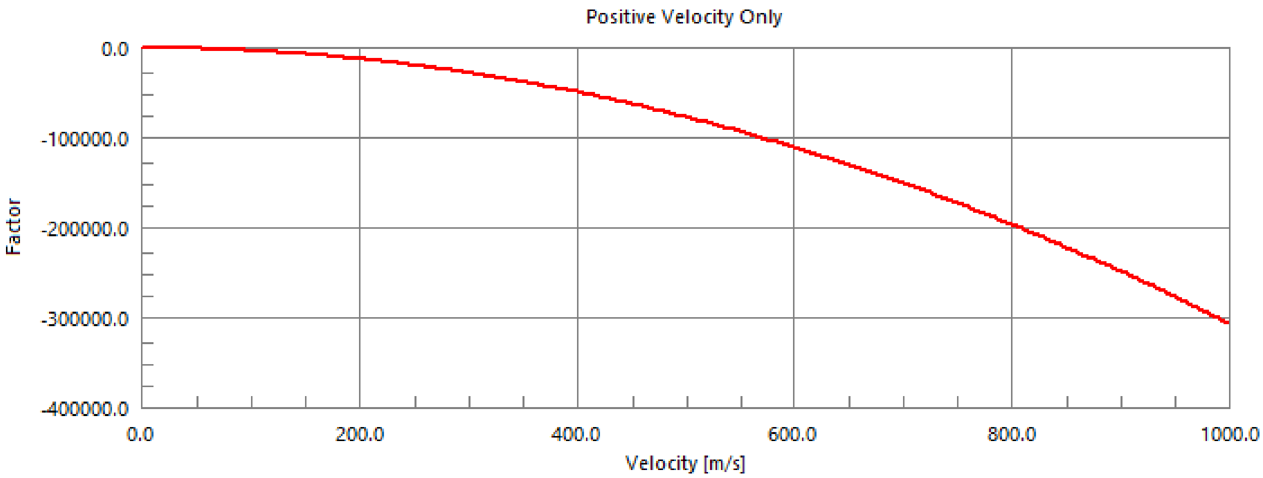
A typical use of these tables is for the application of aerodynamic or hydrodynamic forces on structures moving within a stationary fluid.

Example: Spherical projectile in 3D modelled with brick elements

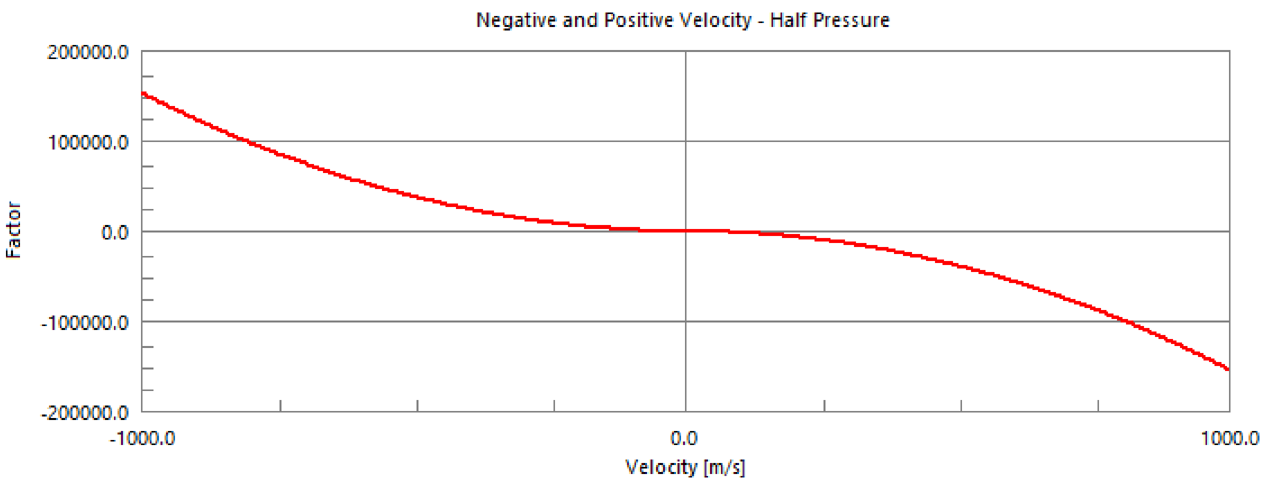


A normal pressure attribute of 1 Pa is applied over the entire outer surface in a consistent direction. The drag force equation, $F_d = \frac{1}{2}\rho_f V^2 C_d A$, is rearranged for use with the pressure load on the brick elements and is entered as $P = \frac{F_d}{A} = \frac{1}{2}\rho_f V^2 C_d$ for the Factor vs Velocity table. If this table is used as is, then the fact that there are pressure attributes on one side of the sphere that point in the direction of travel, as well as pressure attributes on the other side of the sphere that point opposite to the direction of travel, needs to be considered.

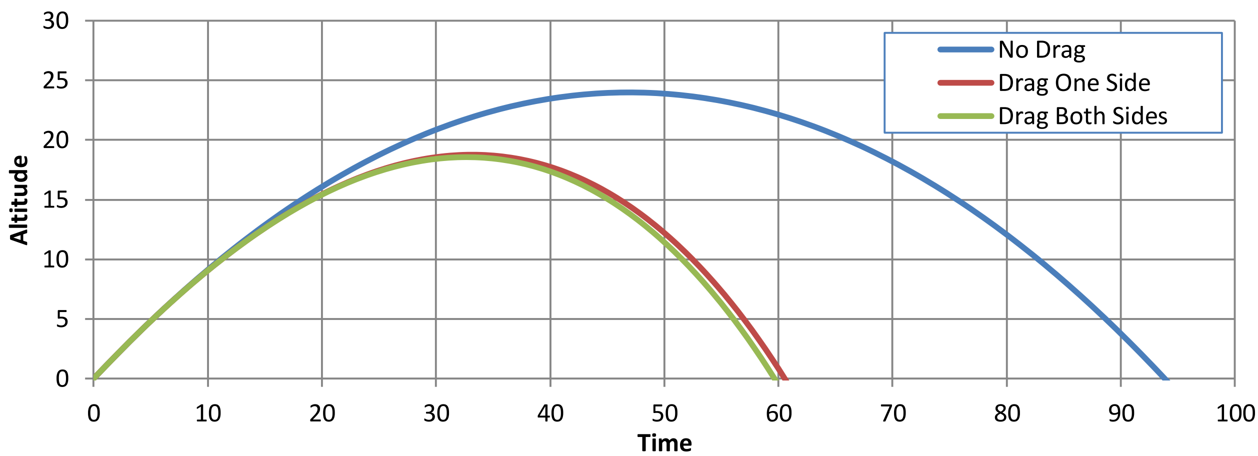
One option is to define just one half of the table so that the pressure attributes in the opposite direction of travel produce no force (i.e., negative relative velocity factors are set to zero) – this is shown below.



Alternatively, the table for both positive and negative relative velocity can be defined, and this will apply the drag pressure on both sides of the sphere in the travel direction: compressive pressure on the leading surface and suction pressure on the trailing surface. For this two-side approach, the drag equation magnitude needs to be factored by half because now the pressure is acting simultaneously on both sides of the sphere – this is shown below.



The following graph is the Strand7 result for the above mesh, simulating the projectile motion in air. The results are close to published values for this problem.

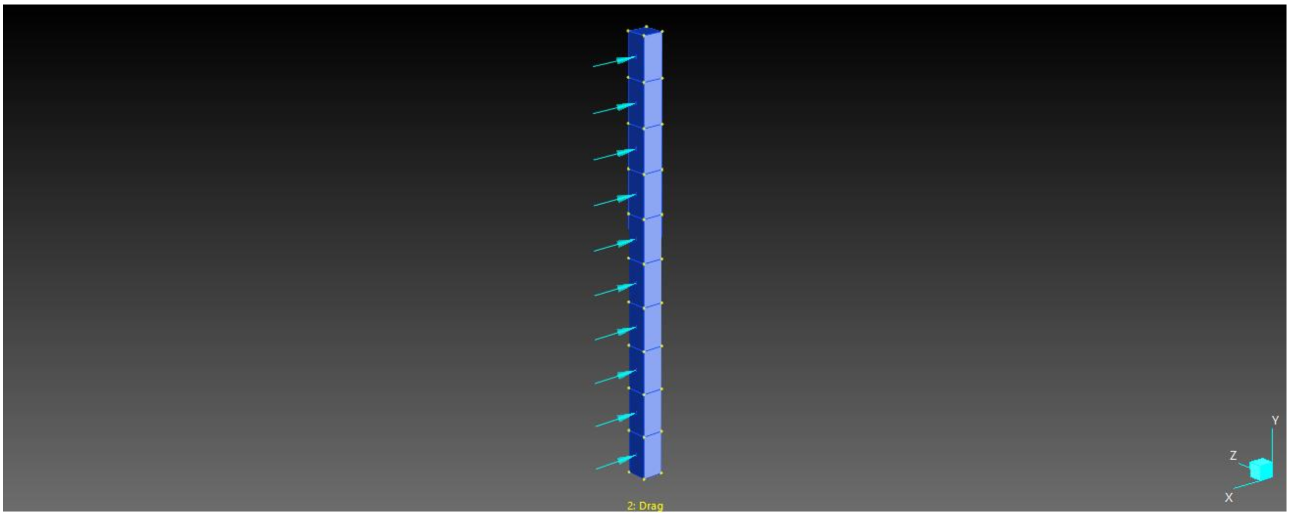


Relative Velocity Tables – Nonlinear Transient Dynamic Analysis

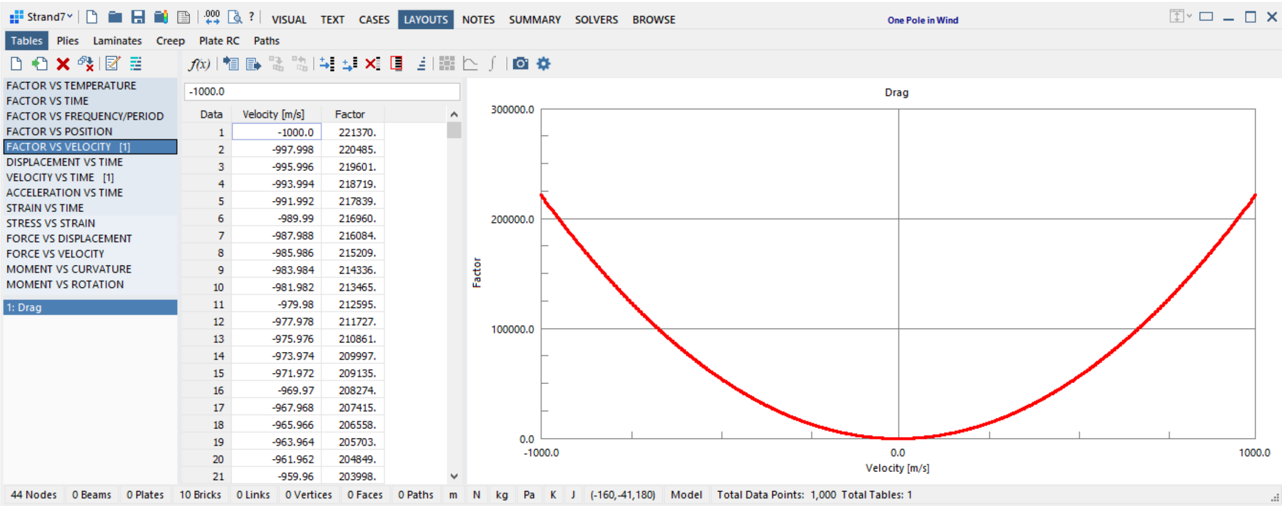
The previous example considers the case of a stationary fluid, in which the relative velocity is just the velocity of the moving node. For the case where a non-stationary fluid is required, up to three additional Factor vs Velocity tables can be associated with the load case to describe the velocity of the fluid as a function of time (Relative Velocity Tables in X, Y and Z). The relative velocity is added to the node velocity before looking up the Factor vs Velocity table.

Example: Pole excited by sinusoidal wind load

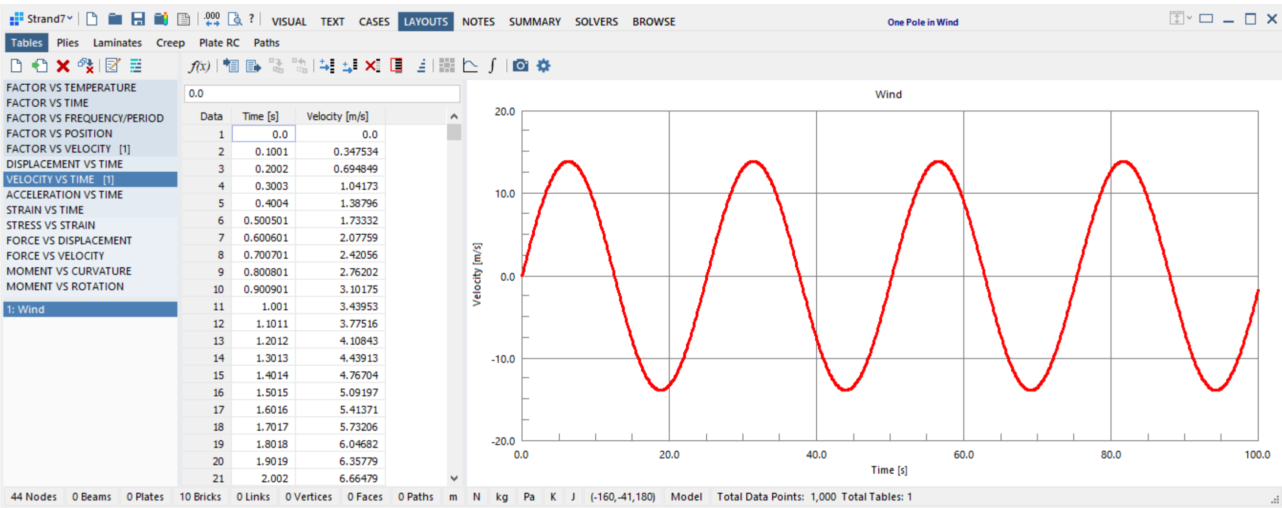
A pressure load case is defined to represent the aerodynamic drag in the X direction.



A Factor vs Velocity table controls the aerodynamic drag force.

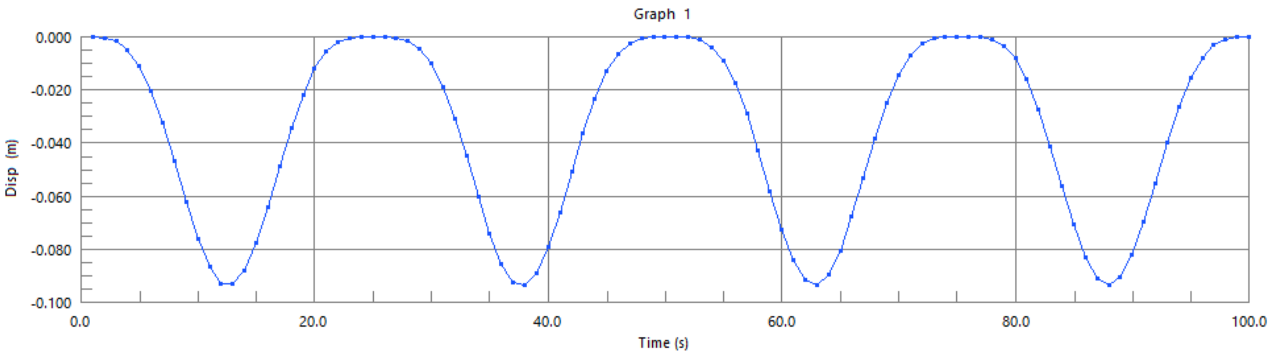


A Velocity vs Time table defines a sinusoidal wind velocity.



The tables are referenced in the **Load** sub-tab of the solver.

A plot of the displacement vs time at the top of the pole reveals a periodic, but non-sinusoidal response.

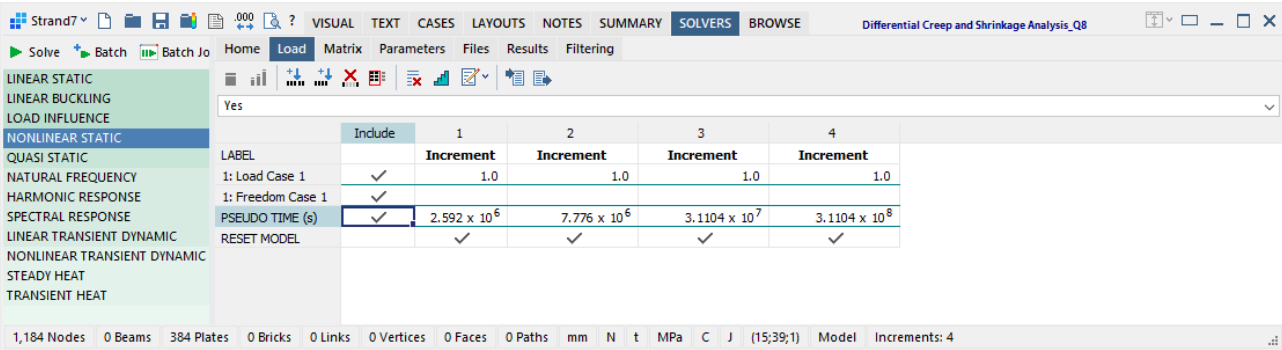


Nonlinear Static Analysis – Pseudo Time and Reset

Each load increment of a nonlinear static analysis in R3 can include a **pseudo time** value and/or a **reset** flag.

The pseudo time option enables elements to consider Factor vs Time tables in the nonlinear static analysis, similarly to quasi-static and nonlinear transient dynamic analysis. A typical use of this feature is to model creep behaviour in a simplified manner.

The reset flag causes the increment to restart as a new analysis clearing the result history up to that point. This feature enables the nonlinear static solver to be used to solve multiple combination cases independently, even if material and/or geometric nonlinearity is considered.



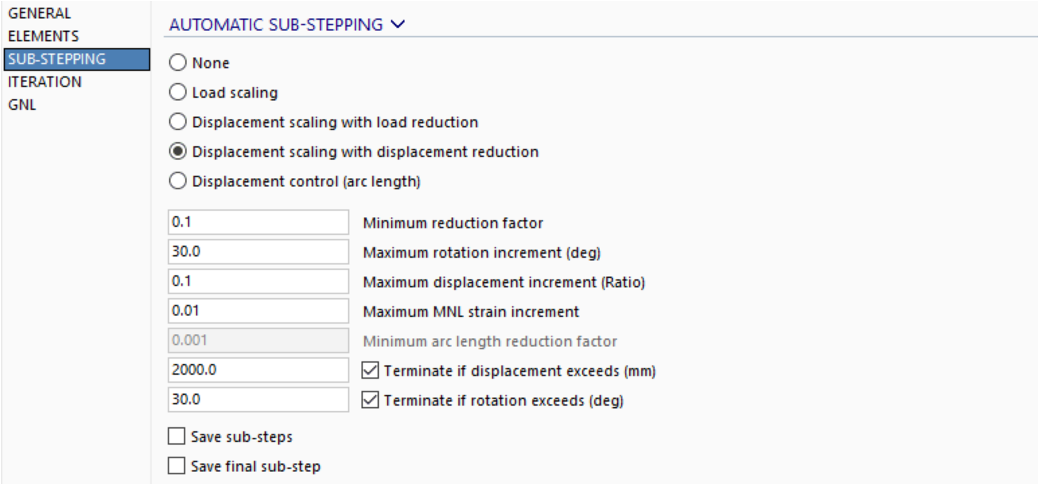
Automatic Sub-stepping Options

A new sub-stepping option in R3 is the **Displacement Scaling with displacement reduction** option. This option is similar to the **Displacement scaling with load reduction** in that it controls the iterative displacements to ensure that they are no larger than the **Maximum displacement increment (Ratio)** and **Maximum rotation increment (deg)** settings.

If convergence cannot be achieved within the set number of iterations, or according to the **Allow solver to add iterations** parameter, the solution does not sub-step. Instead, the **Maximum displacement increment (Ratio)** and **Maximum rotation increment (deg)** settings are reduced and the load step starts again. Reductions for non-converging steps can continue until the **Minimum reduction factor** is reached.

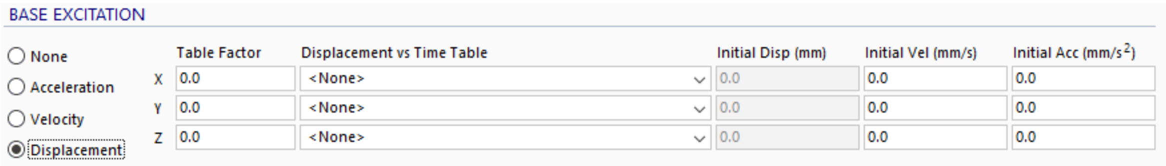
This new method is useful in situations where convergence of the majority of load steps does not require an excessively small **Maximum displacement increment (Ratio)** or **Maximum rotation increment (deg).** For the load steps that require it, the solver automatically reduces the values.

The new options **Terminate if displacement exceeds** and **Terminate if rotation exceeds (deg)** enable the solver to automatically stop once the specified node displacement or rotation limits are exceeded at any node. These options apply to all of the sub-stepping options, including **None**.

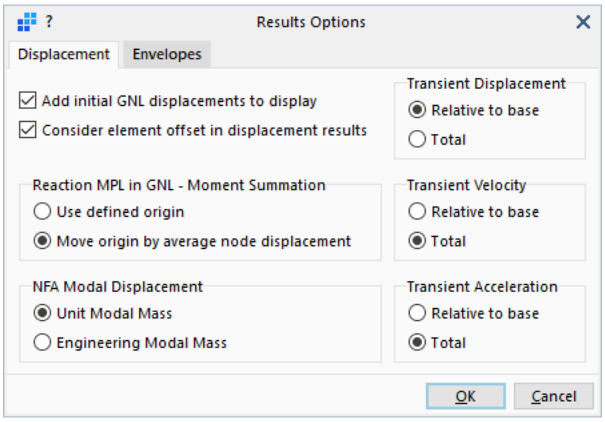


Transient Dynamic Analysis – Base Displacement, Base Velocity and Base Acceleration

In addition to the previously available base acceleration option, base excitation in R3 also supports base displacement and base velocity time history.

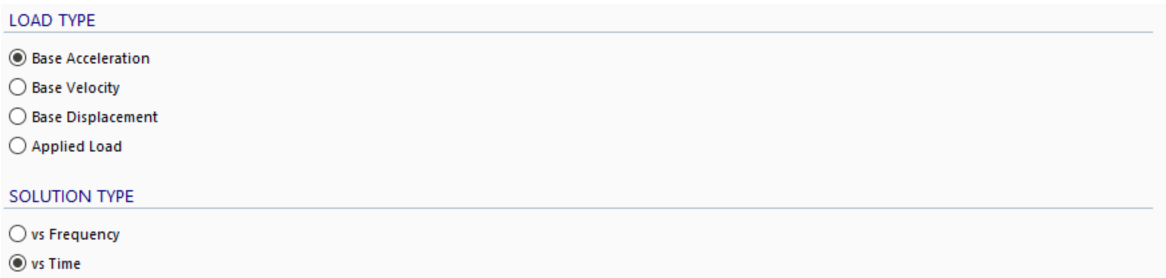


The displacement, velocity and acceleration results for transient dynamic analyses that contain base excitation can be viewed as either total results or relative to the base, by selecting the required option in **Results Options**. This can be switched during post-processing and does not require re-running the analysis.



Harmonic Response Analysis – Base Excitation

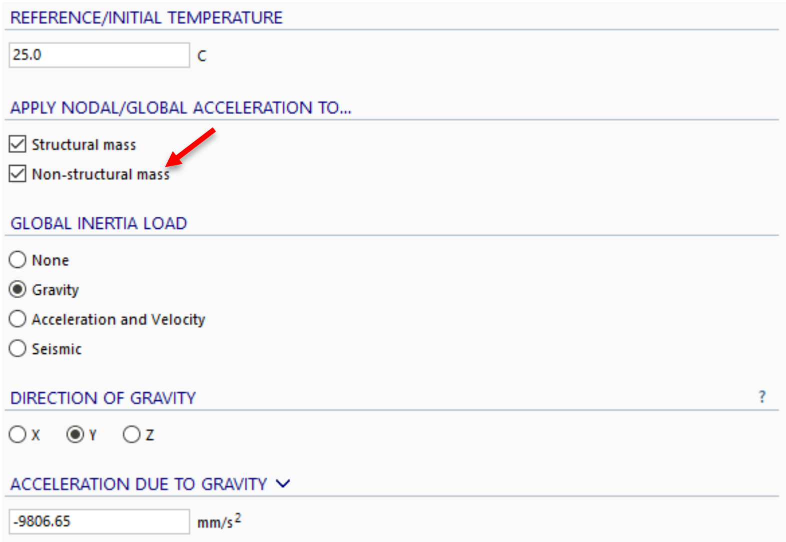
The vs Time mode of the harmonic response solver in R3 supports base excitation as an input option. Any one of base displacement, velocity or acceleration may be applied.



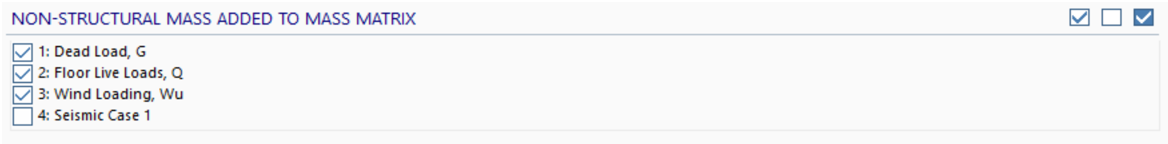
Non-structural Mass in Dynamic Analysis

Mass can be used for two purposes in Strand7: to directly generate forces when acted upon by an acceleration, and for inclusion in the global mass matrix for natural frequency and transient dynamic analysis. In R3, the use of non-structural mass load cases for either purpose is selected independently.

The inclusion of non-structural mass to produce force is set in the load case definition.

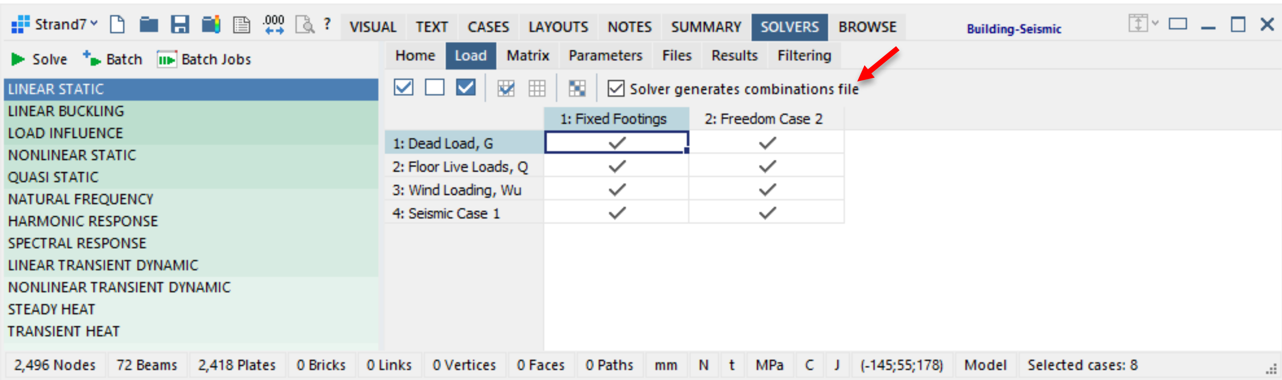


The inclusion of non-structural mass in the global mass matrix is set on the **SOLVERS/Home** tab of the dynamic solvers.



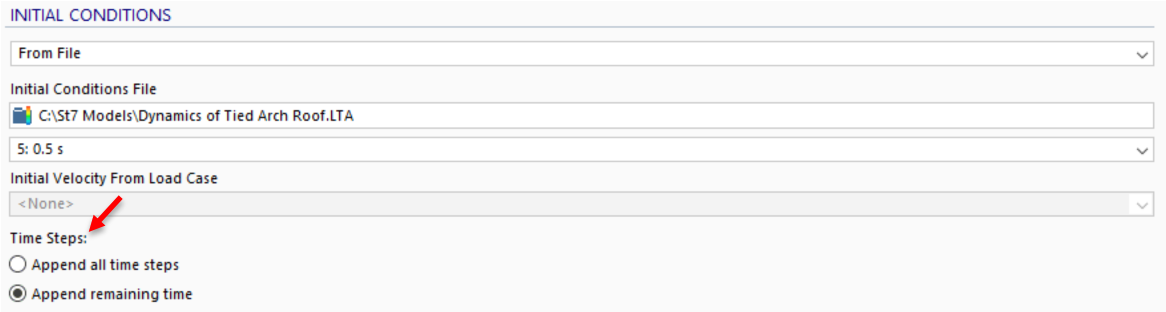
Solver Generates Combinations

To reduce post-processing time after opening a linear static result file containing load case combinations, the task of generating the combinations file can be delegated to the solver process.



Appending Time-based Results

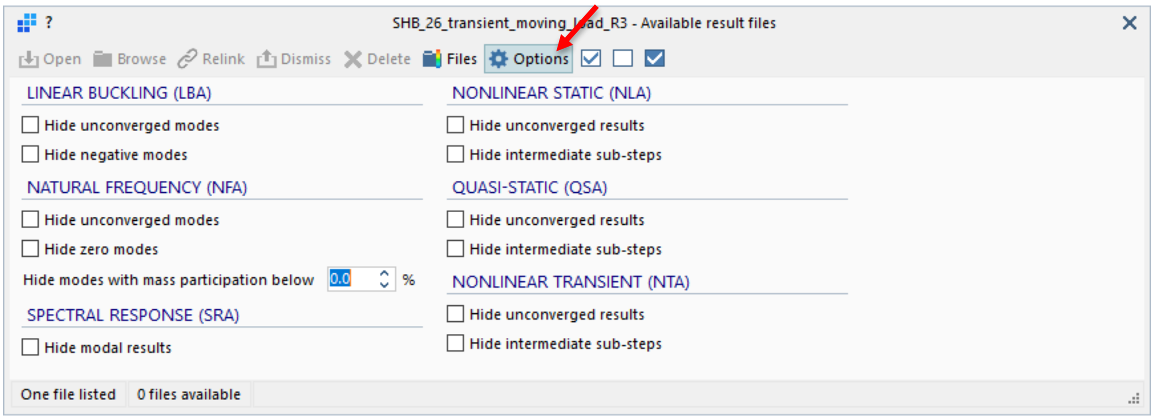
A useful new option is available when appending time-based results such as linear transient dynamic, quasi-static, nonlinear transient dynamic and transient heat results. This is the **Time Steps** option allowing for all of the time steps defined in the current time table to be appended to the results (as in R24), or appending only the time steps beyond the last time instance stored in the initial conditions file.



RESULTS

Opening Result Files

R3 offers some flexibility in the way result files are opened via the **Options** tab of the result file open dialog.



LINEAR BUCKLING (LBA) / NATURAL FREQUENCY (NFA)

- Hide unconverged modes
  If set, unconverged modes will not be available for post-processing.
- Hide negative/zero modes
  If set, negative or zero modes will not be available for post-processing.
- Hide modes with mass participation below %
  Natural frequency modes with mass participation lower than the specified value will not be available for post-processing.

SPECTRAL RESPONSE (SRA)

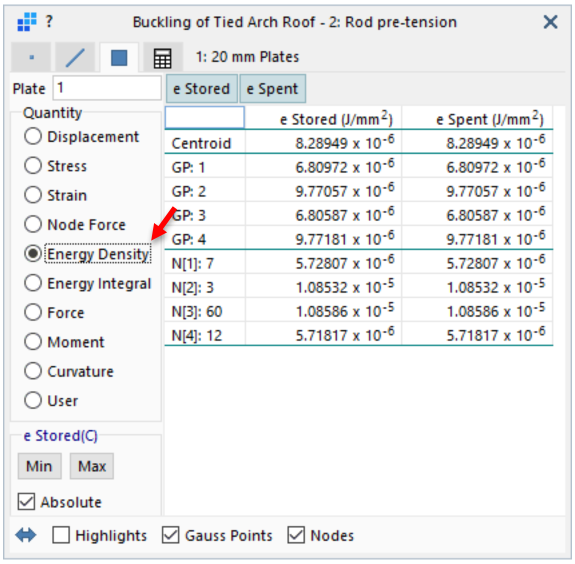
- Hide modal results
  If set, modal results will not be available for post-processing; only the SRSS and CQC result cases will be available (which are usually the important ones).

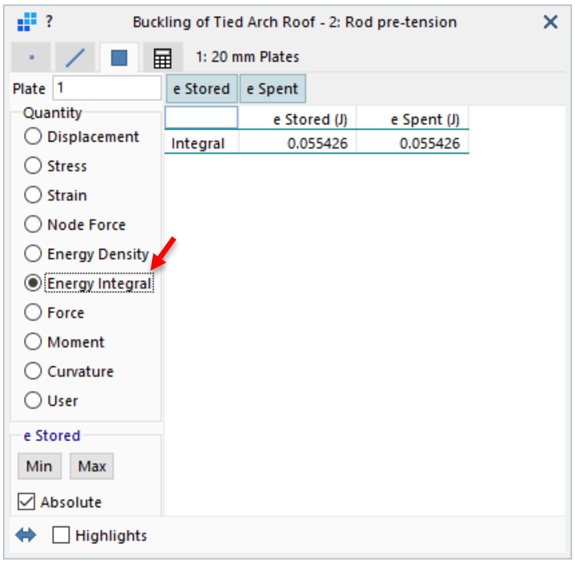
NONLINEAR STATIC (NLA) / QUASI-STATIC (QSA) / NONLINEAR TRANSIENT (NTA)

- Hide unconverged results
  If set, unconverged results will not available for post-processing.
- Hide intermediate sub-steps
  If set, intermediate sub-steps will not be available for post-processing.

Element Energy Integral

R3 offers one additional result energy quantity type, namely the Energy Integral, in the **Peek** and **Graphs** tools and in **LISTINGS**. This reports the total spent or stored energy results (not energy density) of each beam, plate or brick element, as one pair of values for each result case.





Strain Measures

R3 introduces a new strain measure offering both deformational and total strain results.

Strain is a dimensionless quantity that describes the deformation of an object with respect to a reference size. The deformation in the finite element analysis comes from the relative movement of the nodes. The reference size depends on whether the Total Strain or the Deformational Strain is requested, and is affected only by the presence of pre-load attributes (pre-stress, pre-strain, pre-tension, pre-curvature, thermal expansion, temperature gradients). In the absence of these attributes, the two strain measures are the same.

Deformational Strain

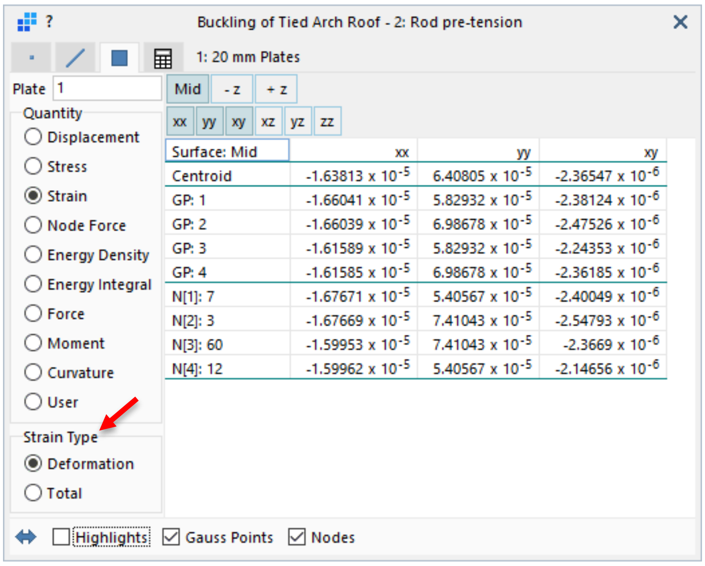
This strain is measured with respect to the initial position of the nodes as defined in the mesh. A beam element fully fixed at both ends with an applied axial pre-strain attribute will produce a zero deformational strain (the nodes do not move), even though there will be a resultant axial force. Conversely, the same axial pre-strain attribute on a cantilever beam will produce a non-zero deformational strain (equal to the pre-strain) but no resultant axial force.

Total Strain

This strain is measured with respect to the position of the nodes after all pre-load attributes have been taken into account (i.e., after an unrestrained element is allowed to fully expand/contract due to the applied pre-loads). A beam element fully fixed at both ends with an applied axial pre-strain attribute will produce a total strain equal to the pre-strain, in addition to a resultant axial force. Conversely, the same axial pre-strain attribute on a cantilever beam will produce a zero total strain and no resulting axial force.

From these two descriptions it is clear that the deformational strain reports the strain that corresponds to how much the nodes in the mesh have moved, whereas the total strain reports the strain that corresponds to the element stress for that strain on the stress-strain table of the material. The total strain is sometimes referred to as the mechanical strain.

These strain measures are available in Peek, LISTINGS and for contour plotting.

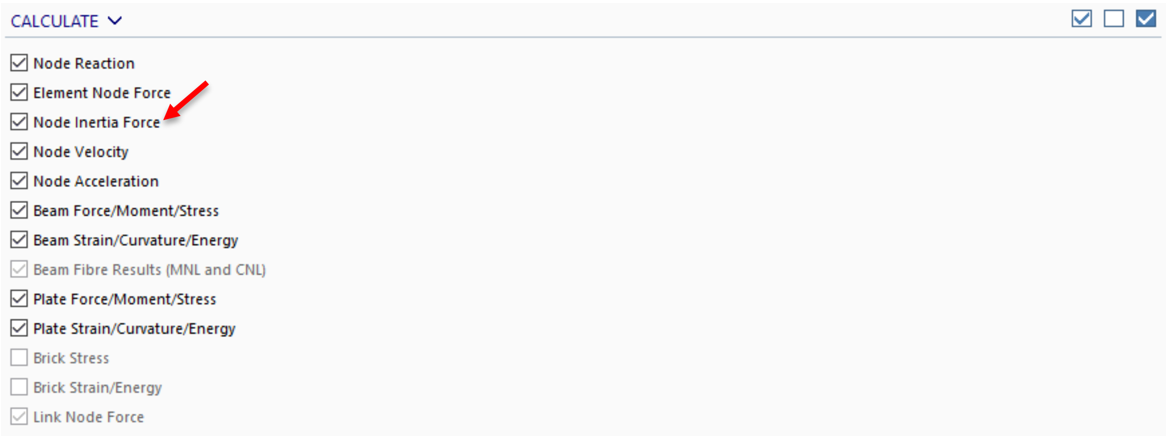


Links as Elements

In R3, links are treated similarly to beam, plate and brick elements. Link results can be inspected with **Peek**, they can be contoured and vectored, and they can be listed in **LISTINGS**.

Spectral Response Node Inertia Force

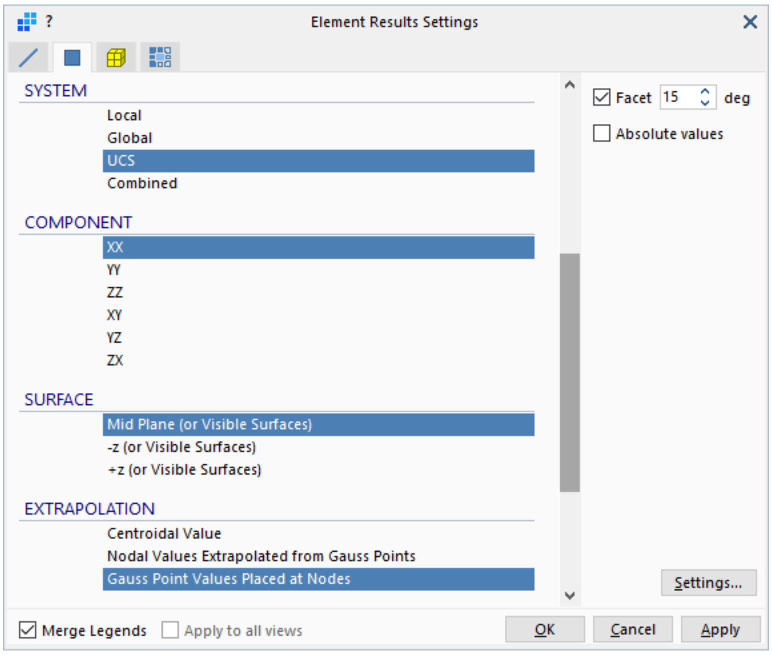
The spectral response solver calculates two types of nodal forces – the elastic forces and the inertia forces. Elastic forces equilibrate to zero at unrestrained nodes and will be non-zero at restrained nodes. The inertia forces correspond to the external forces imposed at a node due to the mass excitation. In R3, both of these result quantities can be stored together in the spectral response result file for post-processing.



POST-PROCESSING

Extrapolation

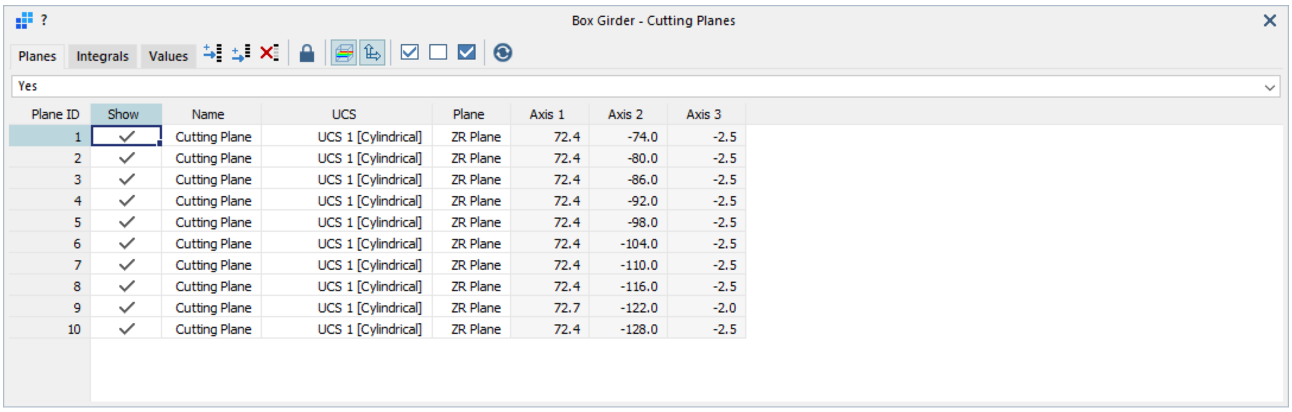
R3 introduces a new extrapolation option for the extraction of node results from Gauss point results on plate and brick elements. This is the **Gauss Point Values Placed at Nodes** option. With this option, the node result is set equal to the value of the nearest Gauss point on the element. For material nonlinear analysis, this is a better option than the extrapolation option, which cannot guarantee that the extrapolated nodal stresses will remain within the yield surface. It is also better than the centroid option because it does not limit the element contour to a constant average value (so peak Gauss point results are not lost).



Brick Cutting Planes

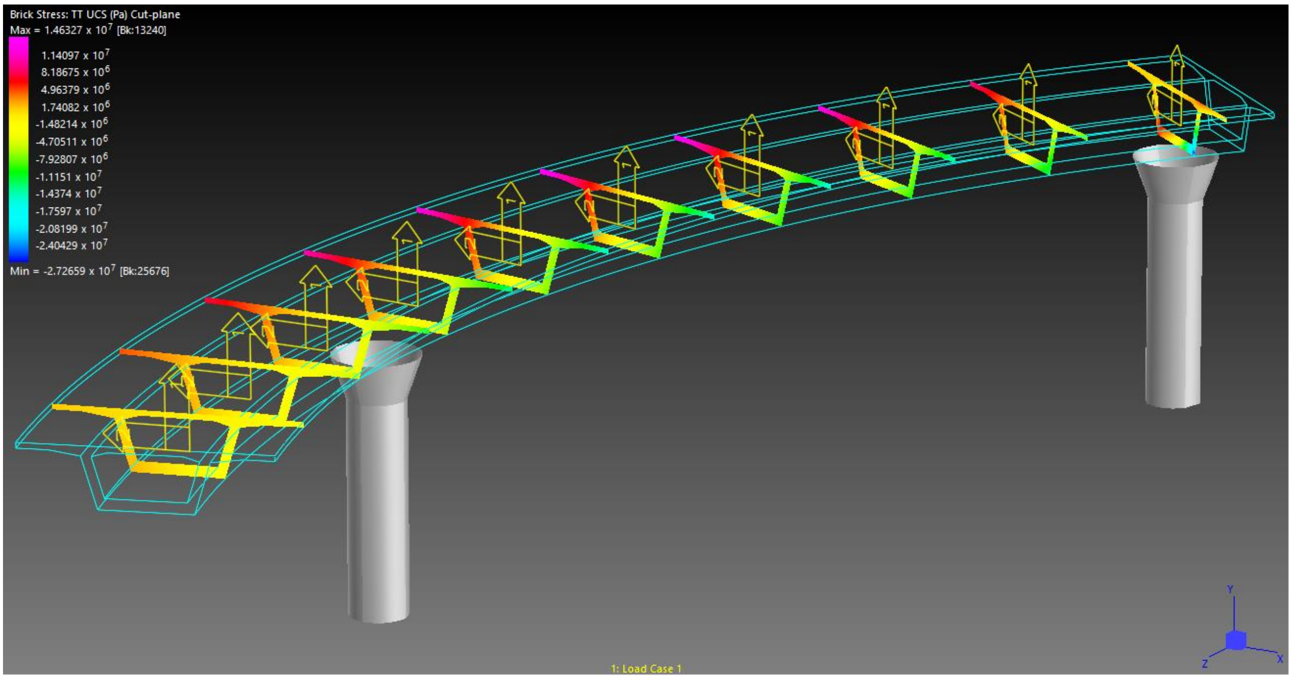
Brick cutting planes containing results are considerably enhanced in R3 compared with R24. Some of the main features in R3 include the following:

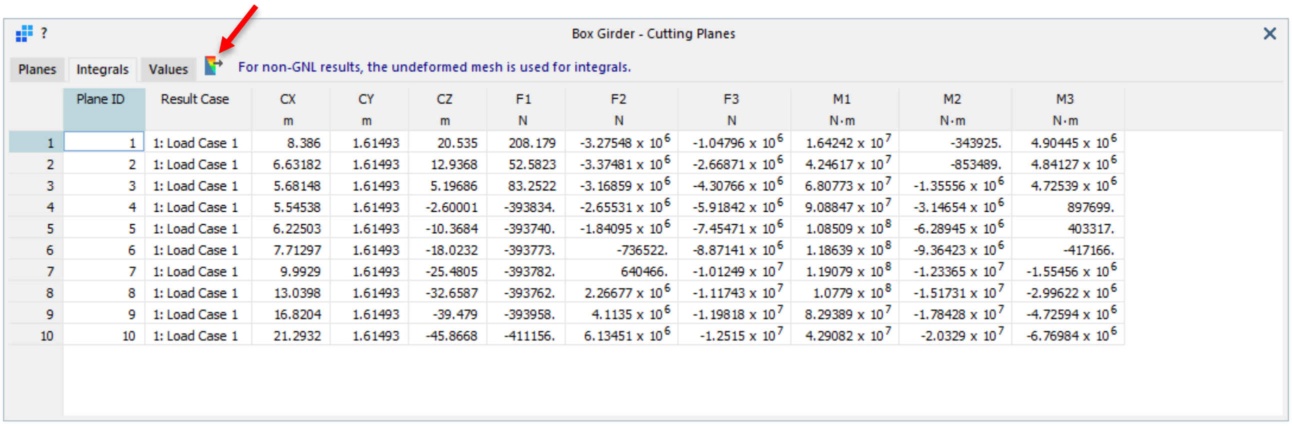
- Multiple cutting planes can be created and stored with the model.
- Cutting planes can be integrated to produce three components of force resultants and three components of moment resultants through the centroid of the plane. Resultants are independently calculated using the appropriate stresses on the plane and are not connected with the current result contour, if any, shown on the model.
- Cutting plane resultants are produced not only for equilibrated results (e.g., linear static analysis) but also for spectral response SRSS and CQC results, and combinations of these with linear static results.
- Cutting plane resultants are also available for envelope result cases.
- Multiple result cases can be calculated together and presented in a single grid.
- The individual stress components at each point on the cutting plane are available in a grid.



Plane ID	Show	Name	UCS	Plane	Axis 1	Axis 2	Axis 3
1	✓	Cutting Plane	UCS 1 [Cylindrical]	ZR Plane	72.4	-74.0	-2.5
2	✓	Cutting Plane	UCS 1 [Cylindrical]	ZR Plane	72.4	-80.0	-2.5
3	✓	Cutting Plane	UCS 1 [Cylindrical]	ZR Plane	72.4	-86.0	-2.5
4	✓	Cutting Plane	UCS 1 [Cylindrical]	ZR Plane	72.4	-92.0	-2.5
5	✓	Cutting Plane	UCS 1 [Cylindrical]	ZR Plane	72.4	-98.0	-2.5
6	✓	Cutting Plane	UCS 1 [Cylindrical]	ZR Plane	72.4	-104.0	-2.5
7	✓	Cutting Plane	UCS 1 [Cylindrical]	ZR Plane	72.4	-110.0	-2.5
8	✓	Cutting Plane	UCS 1 [Cylindrical]	ZR Plane	72.4	-116.0	-2.5
9	✓	Cutting Plane	UCS 1 [Cylindrical]	ZR Plane	72.7	-122.0	-2.0
10	✓	Cutting Plane	UCS 1 [Cylindrical]	ZR Plane	72.4	-128.0	-2.5

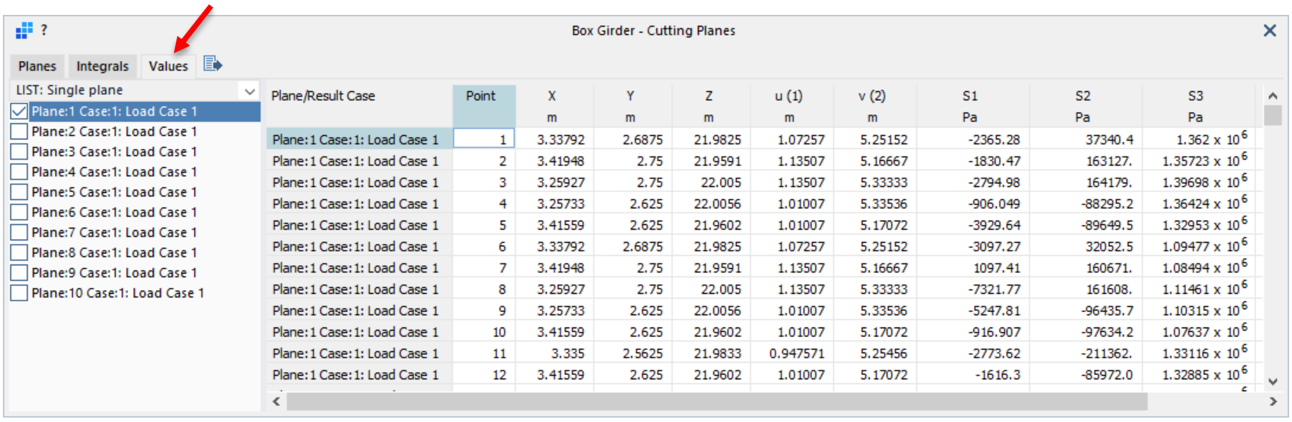
Cutting plane definitions.





	Plane ID	Result Case	CX m	CY m	CZ m	F1 N	F2 N	F3 N	M1 N·m	M2 N·m	M3 N·m
1	1	1: Load Case 1	8.386	1.61493	20.535	208.179	-3.27548×10^6	-1.04796×10^6	1.64242×10^7	-343925.	4.90445×10^6
2	2	1: Load Case 1	6.63182	1.61493	12.9368	52.5823	-3.37481×10^6	-2.66871×10^6	4.24617×10^7	-853489.	4.84127×10^6
3	3	1: Load Case 1	5.68148	1.61493	5.19686	83.2522	-3.16859×10^6	-4.30766×10^6	6.80773×10^7	-1.35556×10^6	4.72539×10^6
4	4	1: Load Case 1	5.54538	1.61493	-2.60001	-393834.	-2.65531×10^6	-5.91842×10^6	9.08847×10^7	-3.14654×10^6	897699.
5	5	1: Load Case 1	6.22503	1.61493	-10.3684	-393740.	-1.84095×10^6	-7.45471×10^6	1.08509×10^8	-6.28945×10^6	403317.
6	6	1: Load Case 1	7.71297	1.61493	-18.0232	-393773.	-736522.	-8.87141×10^6	1.18639×10^8	-9.36423×10^6	-417166.
7	7	1: Load Case 1	9.9929	1.61493	-25.4805	-393782.	640466.	-1.01249×10^7	1.19079×10^8	-1.23365×10^7	-1.55456×10^6
8	8	1: Load Case 1	13.0398	1.61493	-32.6587	-393762.	2.26677×10^6	-1.11743×10^7	1.0779×10^8	-1.51731×10^7	-2.99622×10^6
9	9	1: Load Case 1	16.8204	1.61493	-39.479	-393958.	4.1135×10^6	-1.19818×10^7	8.29389×10^7	-1.78428×10^7	-4.72594×10^6
10	10	1: Load Case 1	21.2932	1.61493	-45.8668	-411156.	6.13451×10^6	-1.2515×10^7	4.29082×10^7	-2.0329×10^7	-6.76984×10^6

Cutting plane resultants for multiple result cases.

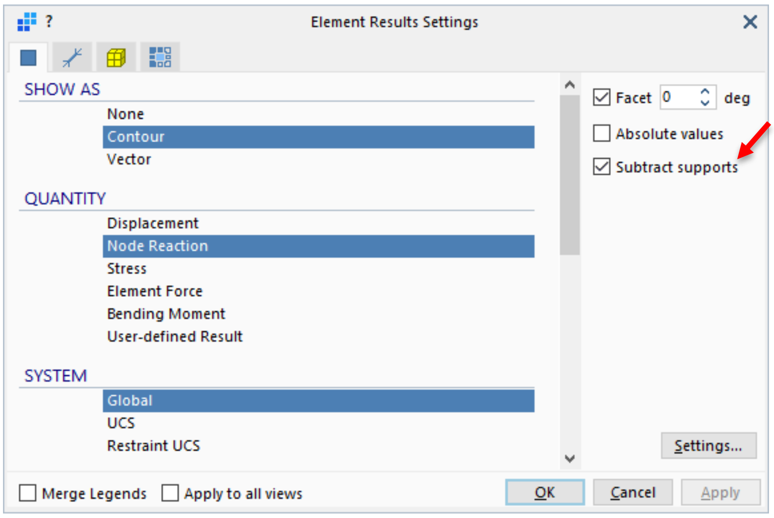


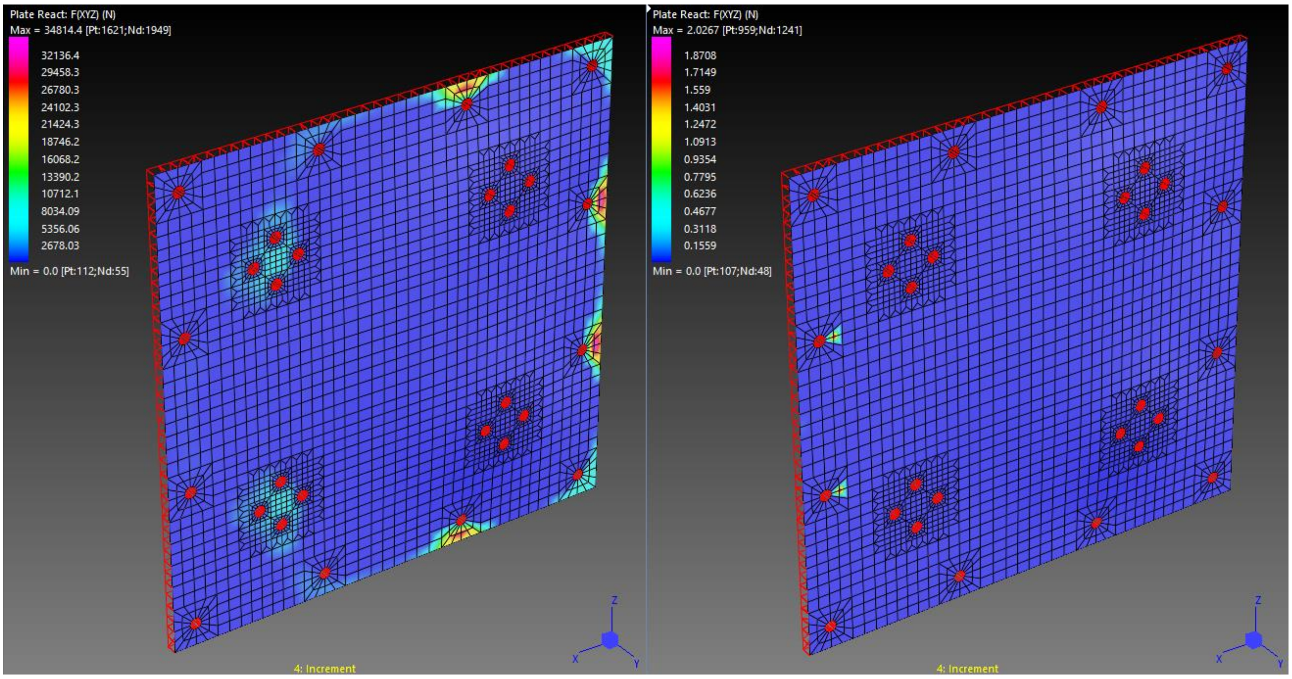
Plane/Result Case	Point	X m	Y m	Z m	u (1) m	v (2) m	S1 Pa	S2 Pa	S3 Pa
Plane:1 Case:1: Load Case 1	1	3.33792	2.6875	21.9825	1.07257	5.25152	-2365.28	37340.4	1.362×10^6
Plane:1 Case:1: Load Case 1	2	3.41948	2.75	21.9591	1.13507	5.16667	-1830.47	163127.	1.35723×10^6
Plane:1 Case:1: Load Case 1	3	3.25927	2.75	22.005	1.13507	5.33333	-2794.98	164179.	1.39698×10^6
Plane:1 Case:1: Load Case 1	4	3.25733	2.625	22.0056	1.01007	5.33536	-906.049	-88295.2	1.36424×10^6
Plane:1 Case:1: Load Case 1	5	3.41559	2.625	21.9602	1.01007	5.17072	-3929.64	-89649.5	1.32953×10^6
Plane:1 Case:1: Load Case 1	6	3.33792	2.6875	21.9825	1.07257	5.25152	-3097.27	32052.5	1.09477×10^6
Plane:1 Case:1: Load Case 1	7	3.41948	2.75	21.9591	1.13507	5.16667	1097.41	160671.	1.08494×10^6
Plane:1 Case:1: Load Case 1	8	3.25927	2.75	22.005	1.13507	5.33333	-7321.77	161608.	1.11461×10^6
Plane:1 Case:1: Load Case 1	9	3.25733	2.625	22.0056	1.01007	5.33536	-5247.81	-96435.7	1.10315×10^6
Plane:1 Case:1: Load Case 1	10	3.41559	2.625	21.9602	1.01007	5.17072	-916.907	-97634.2	1.07637×10^6
Plane:1 Case:1: Load Case 1	11	3.335	2.5625	21.9833	0.947571	5.25456	-2773.62	-211362.	1.33116×10^6
Plane:1 Case:1: Load Case 1	12	3.41559	2.625	21.9602	1.01007	5.17072	-1616.3	-85972.0	1.32885×10^6

Coefficients of stress tensor at every intersection point on the cutting plane.

Subtract Support Reactions

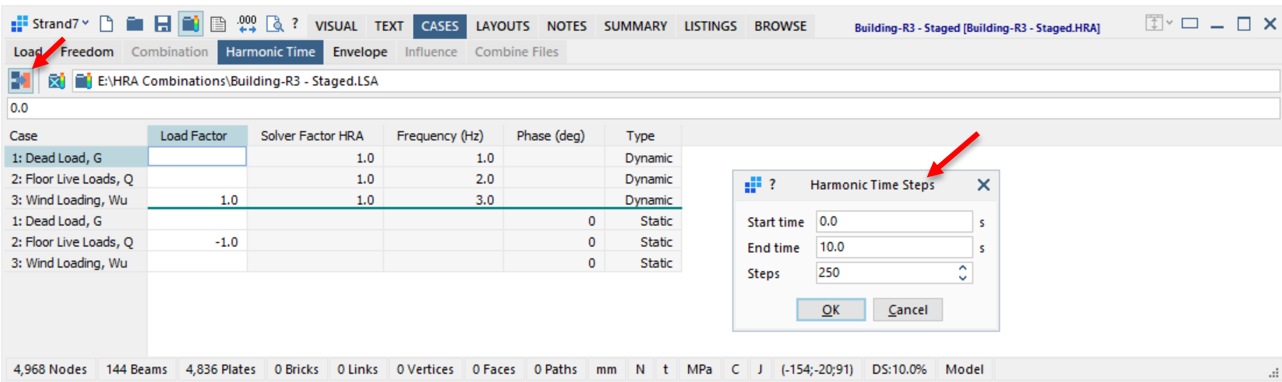
Nodal reactions are usually non-zero at restrained nodes and at nodes supported by elastic support attributes. Reactions at other nodes are theoretically zero, but due to round-off they are usually also non-zero (although they are expected to be very small). The magnitude of reactions (or unbalanced forces) at unrestrained nodes is of particular interest in nonlinear analysis as it provides a good indicator as to the level of convergence obtained. At nodes supported by face support attributes, R3 offers the option to subtract the support reaction, thereby facilitating the inspection of just the unbalanced forces.





Linear Static + Harmonic Response Combinations

R3 offers the functionality to generate combinations of linear static and harmonic response analyses.



Natural Frequency Analysis Mode Shape Normalisation

The natural frequency solver calculates the frequencies and corresponding mode shapes of a structure. By default, the mode shapes are normalised such that they produce a unit modal mass; that is,

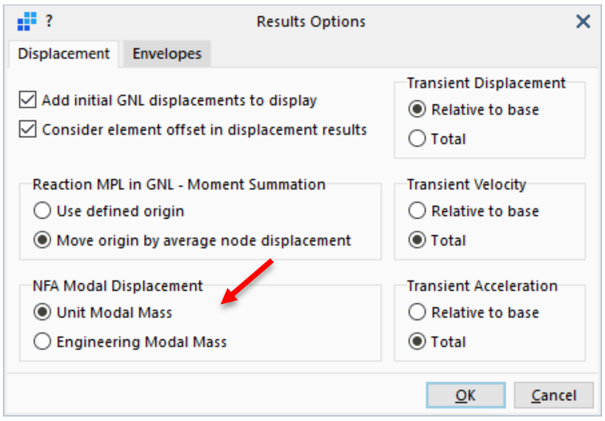
$$\varphi_i^T M \varphi_i = 1$$

where φ_i is the mode shape vector for mode i and M is the global mass matrix.

For the calculation of the engineering modal mass, a different normalisation is used: here the mode shape vector is scaled such that the largest displacement in the global X, Y, or Z direction is equal to 1.0 units of length. The natural frequency solver lists the nodes and degrees of freedom corresponding to this normalisation of the mode shape vector.

```
NODE/DOF EIGENVECTOR NORMALISATION FOR ENGINEERING MODAL MASS AND STIFFNESS
 Mode  Frequency (Hz)      Node   DoF
    1   1.466385E+00        619    DZ
    2   2.061489E+00       1664    DX
    3   2.789282E+00        564    DZ
    4   4.510150E+00        619    DZ
    5   7.475686E+00       1597    DZ
    6   7.844652E+00        564    DX
```

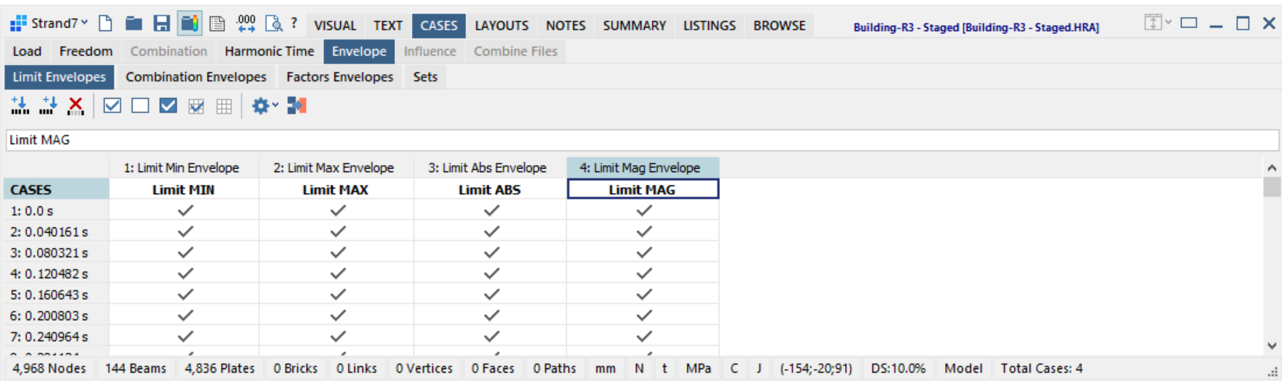
A new option is available in R3 to use this mode shape vector (based on the engineering modal mass) for post-processing of natural frequency results (graphical display, **Peek** and **LISTINGS**).



Envelopes

Limit Mag Type

A new Limit Envelope type is available in R3, namely the **Limit Mag** type. This is a signed version of the **Limit Abs** type – the envelope value with the maximum magnitude is retained together with its sign.



Harmonic Response and Heat Transfer Envelopes

In addition to the result types in which enveloping was previously available (such as linear static, nonlinear static, and so on) enveloping in R3 is also available for steady state heat transfer, transient heat transfer and harmonic response analysis. In the case of harmonic response analysis, envelopes are available for vs Frequency and vs Time solution modes, as well as for combinations of linear static and harmonic vs time results.

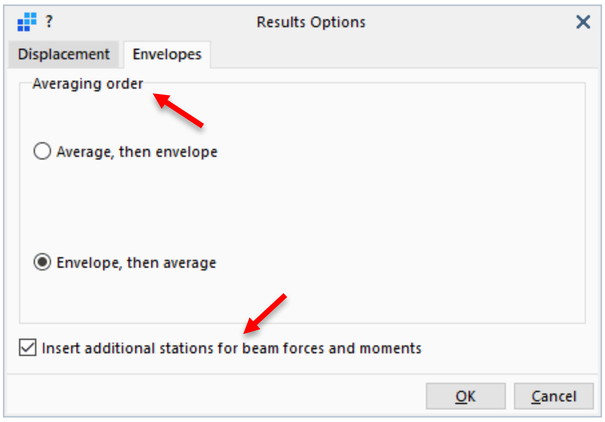
Envelope Options

Two new options are available for envelopes.

The first is to do with the order of operations and applies to plate and brick element results:

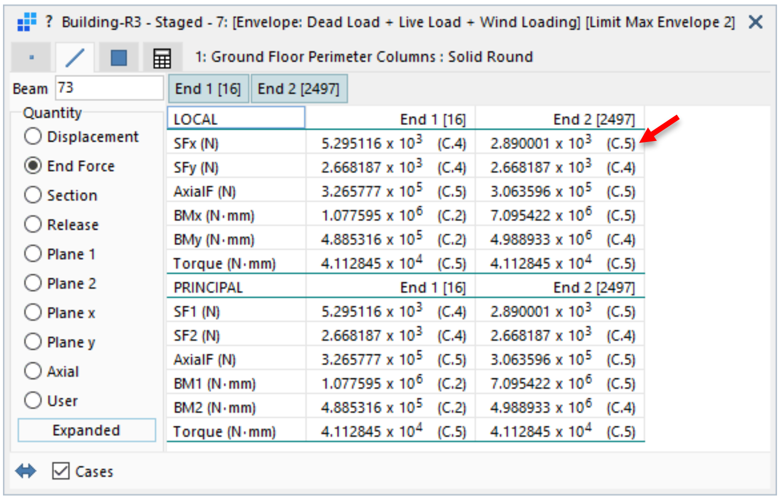
- envelope the element and then average the enveloped result; or
- average the element result and then envelope it.

The second is to do with beam force/moment envelopes. The default option is that the enveloping function will add additional stations along each beam to ensure that limits are captured (e.g., location of maximum moment). This means that the number of result stations produced for each element can be different. To enable the generation of results at a fixed number of equally spaced points along each element, the insertion of limit points can be turned off.



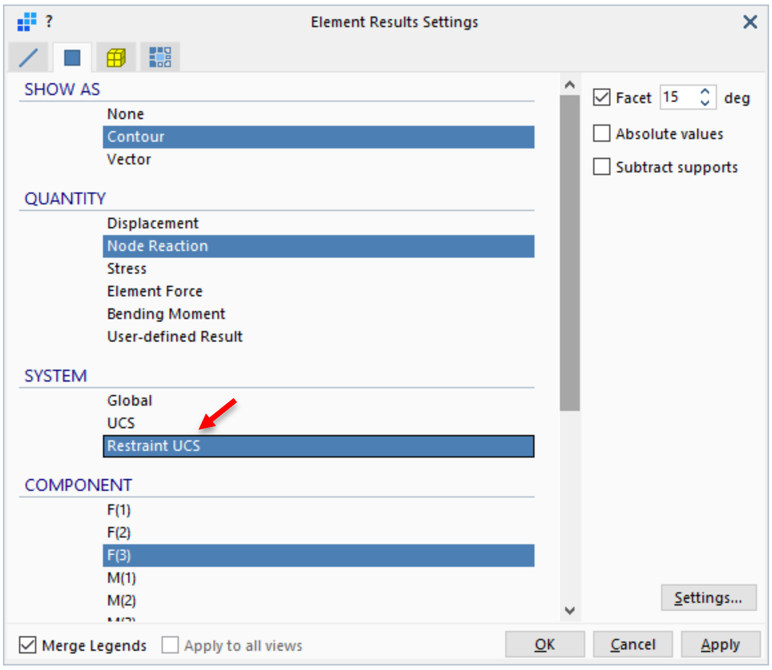
Case Identification

For the limit envelopes, R3 reports the result case that gave rise to a particular envelope result. This is available in **Peek**, **LISTINGS** and in result legends in the model window.



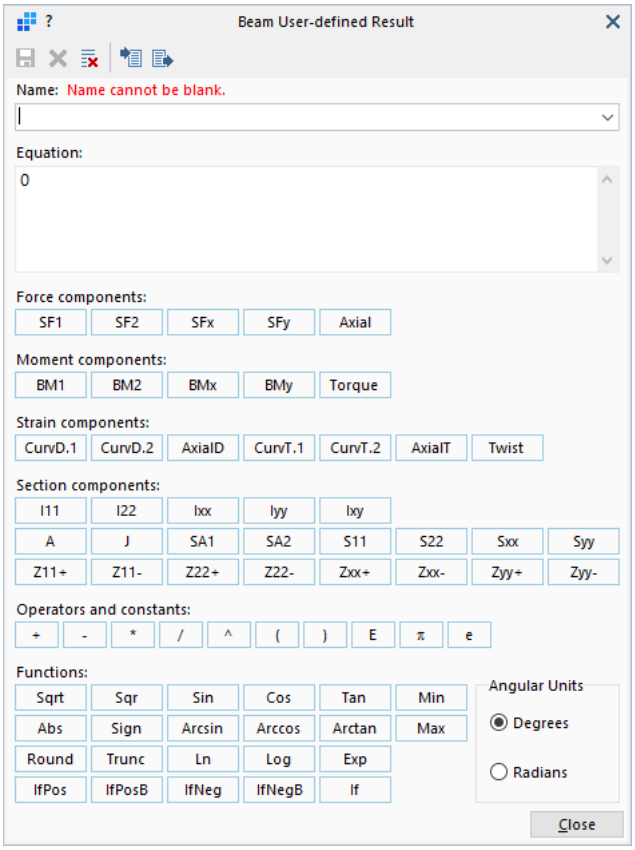
UCS Node Reactions

In addition to node reactions in the global Cartesian system and in any UCS, R3 also offers the option to plot reaction contours in the local system of the applied nodal restraint attribute.



User-defined results equations

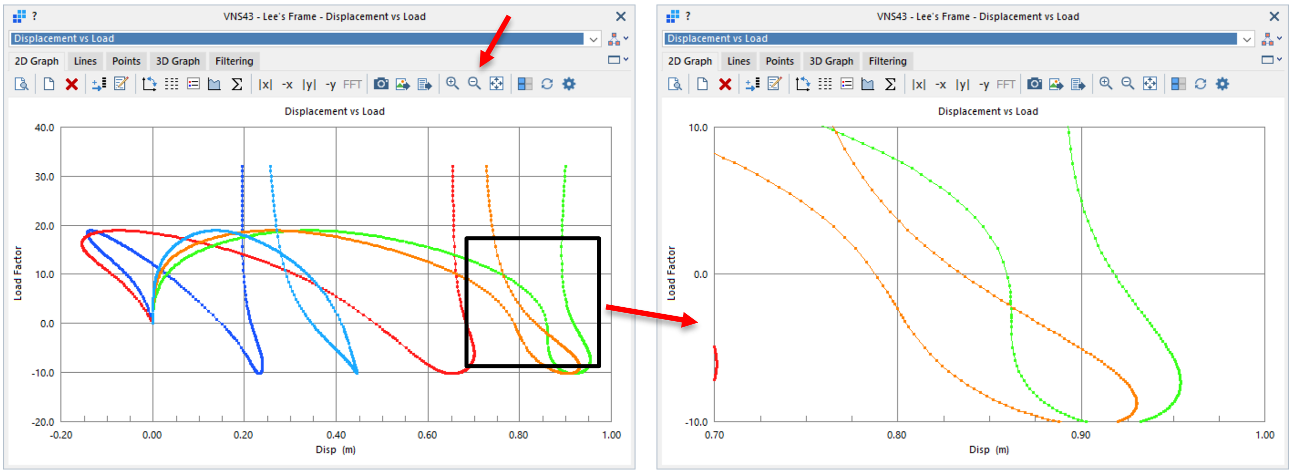
The user-defined result option, previously available for plate and brick elements is also available for beam elements in R3. In addition, multiple equations can be defined and stored in R3 for each element type. This enables easy switching between results without having to re-enter the equation every time. Equations can also be stored to file for later retrieval.



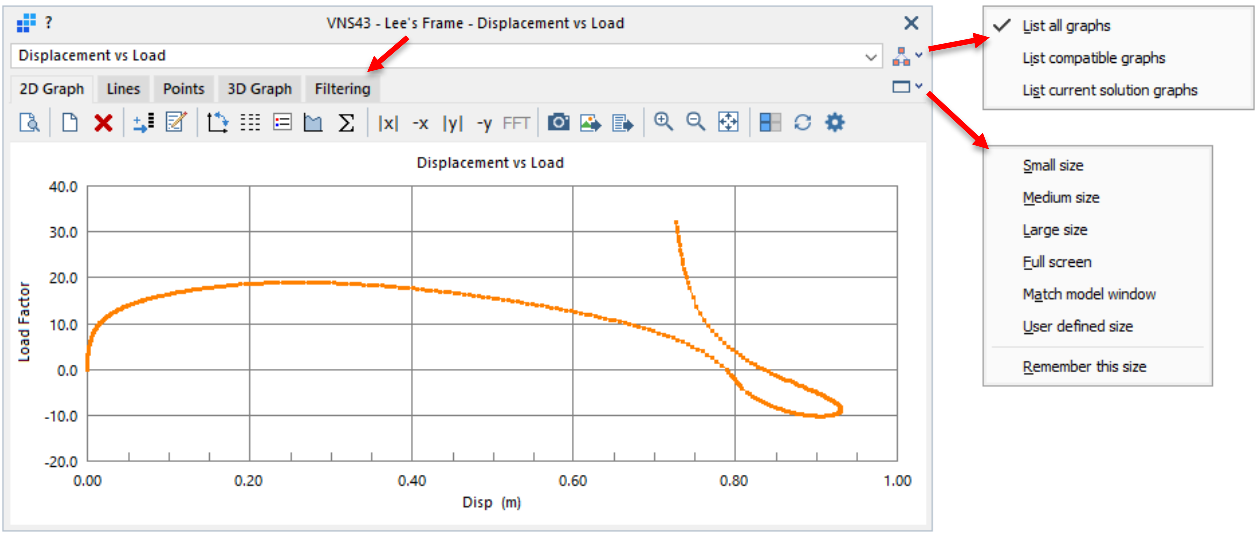
Graphs – standard size windows, types, zoom, double-click point

New features available in the R3 version of the Graphs tool include the following.

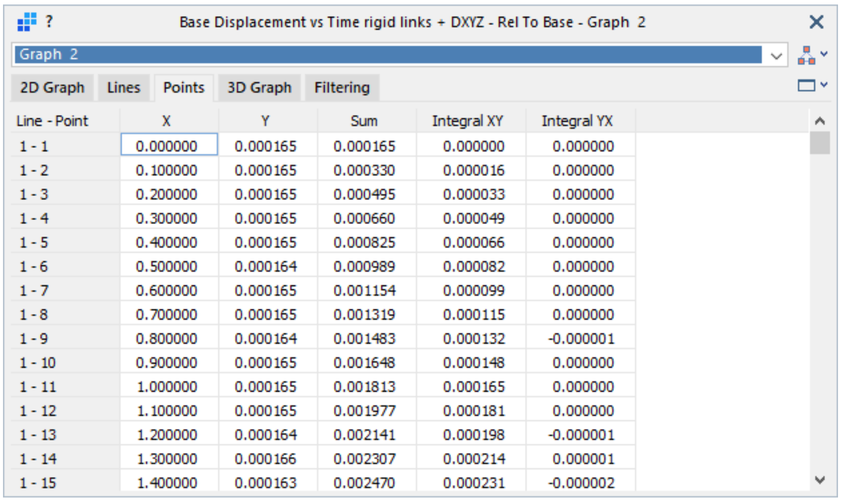
- Graphical zoom function.



- Ability to list all defined graphs, only compatible graphs, or graphs specifically generated for the currently open result file type. For example, when heat transfer results are open, do not list graphs pertaining the linear static results.
- Popup menu to set the window size based on one of the pre-set sizes or a user-defined size (this is useful for producing reports where consistent graph image sizes are required).



- Double-click function on a point on the graph to focus the model window to the same result case or entity pertaining to the double-clicked point.
- Filtering options allow more control on the averaging of adjacent element results when the graph passes through common points.
- The **Points** tab provides three new columns of data calculated based on the extracted values: **Sum**, **Integral XY** and **Integral YX**. These columns are meaningful in the context of the extracted values (for example, the integral under of a force-displacement graph gives the accumulated spent energy at each point).



Base Displacement vs Time rigid links + DXYZ - Rel To Base - Graph 2

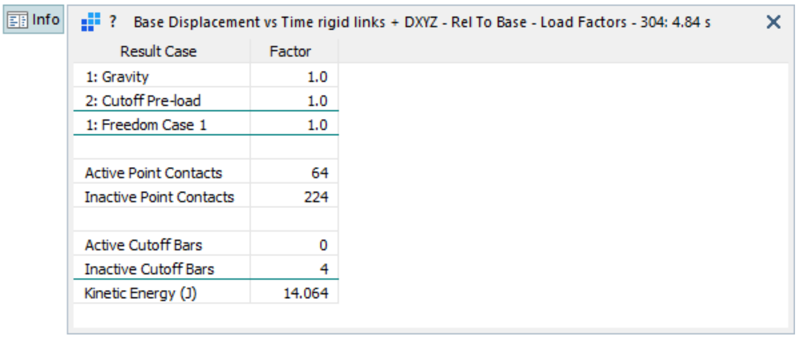
Graph 2

2D Graph | Lines | Points | 3D Graph | Filtering

Line - Point	X	Y	Sum	Integral XY	Integral YX
1 - 1	0.000000	0.000165	0.000165	0.000000	0.000000
1 - 2	0.100000	0.000165	0.000330	0.000016	0.000000
1 - 3	0.200000	0.000165	0.000495	0.000033	0.000000
1 - 4	0.300000	0.000165	0.000660	0.000049	0.000000
1 - 5	0.400000	0.000165	0.000825	0.000066	0.000000
1 - 6	0.500000	0.000164	0.000989	0.000082	0.000000
1 - 7	0.600000	0.000165	0.001154	0.000099	0.000000
1 - 8	0.700000	0.000165	0.001319	0.000115	0.000000
1 - 9	0.800000	0.000164	0.001483	0.000132	-0.000001
1 - 10	0.900000	0.000165	0.001648	0.000148	0.000000
1 - 11	1.000000	0.000165	0.001813	0.000165	0.000000
1 - 12	1.100000	0.000165	0.001977	0.000181	0.000000
1 - 13	1.200000	0.000164	0.002141	0.000198	-0.000001
1 - 14	1.300000	0.000166	0.002307	0.000214	0.000001
1 - 15	1.400000	0.000163	0.002470	0.000231	-0.000002

Result Case Information

The **Info** function found on the **VISUAL/Results** tab provides useful information about the current result case. This is available for all solver types. The information provided is in the context of both the result type and the result case.



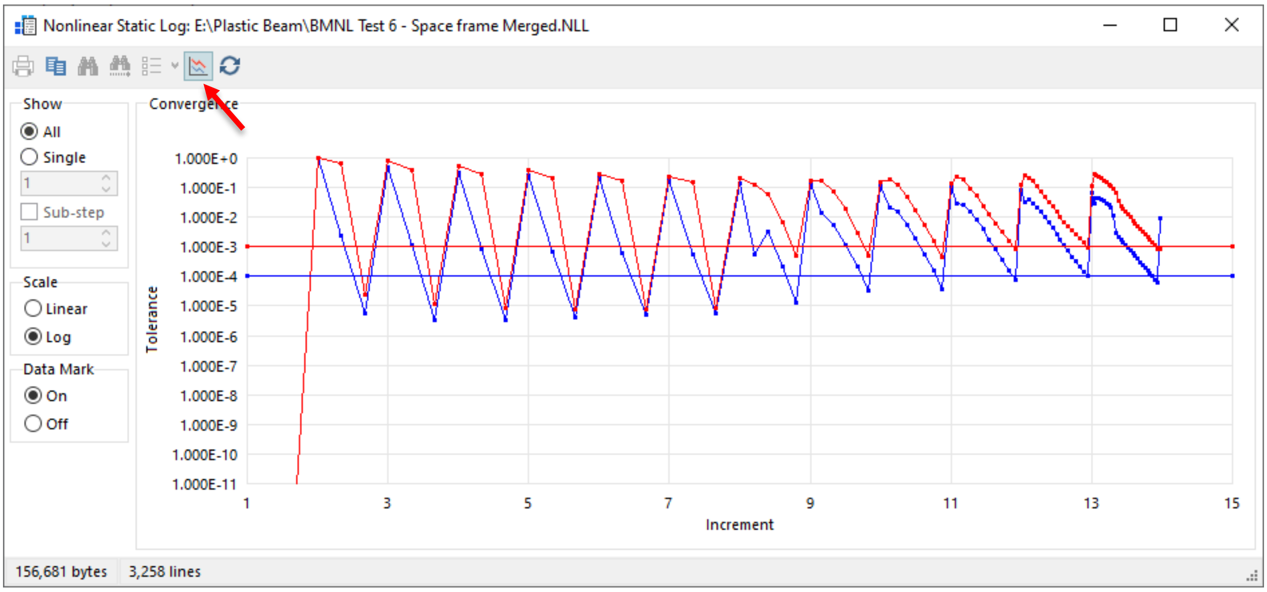
Info

Base Displacement vs Time rigid links + DXYZ - Rel To Base - Load Factors - 304: 4.84 s

Result Case	Factor
1: Gravity	1.0
2: Cutoff Pre-load	1.0
1: Freedom Case 1	1.0
Active Point Contacts	64
Inactive Point Contacts	224
Active Cutoff Bars	0
Inactive Cutoff Bars	4
Kinetic Energy (J)	14.064

Log File Viewer

The log file viewer in R3 offers the option to reconstruct the convergence graph for the nonlinear solvers (nonlinear static, quasi-static, nonlinear transient dynamic analyses and nonlinear heat).



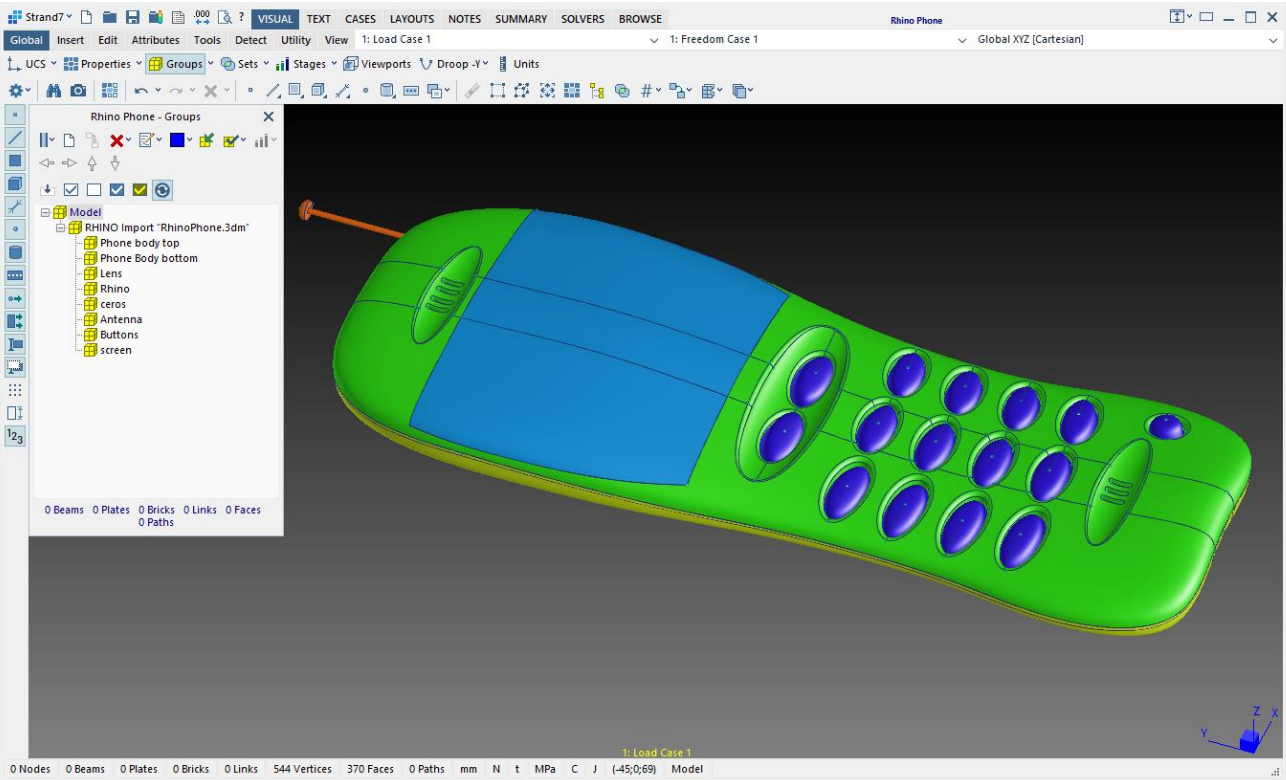
IMPORT/EXPORT

DXF Import

AutoCAD's DXF file import is enhanced in R3 with ACIS-SAT files embedded in the DXF also imported.

Rhino 3dm File Import

Native Rhinoceros® 3dm files may be imported directly into R3. This is usually more efficient than going through a neutral format such as IGES or STEP as more of the data can be imported.



API

There are extensive changes to the R3 API compared with the R24 API. However, the task of converting an R24 API program to run with the R3 API is not a major one. The **Strand7 R24 to R3 API Porting Guide** provides information that will help simplify the task.

Functions

The R3 API offers some 2100 functions compared with around 1200 functions in the R24 API.

Headers

In addition to headers for the previously available programming languages and development environments, headers are also provided for Grasshopper®.

DLL Solver

The R3 API introduces a DLL version of the solver (in addition to the EXE version). The DLL version is advantageous for API projects that need to run the solver hundreds or thousands of times on relatively small models (e.g., for optimisation). Once loaded, the DLL remains memory resident, so the time to launch the next solve is usually measurably less compared with loading the EXE version of the solver each time.

Tools Functions

All of the element and geometry modelling tools available in the R3 GUI are also available via the R3 API. For example, the function `St7CopyByIncrement()` can be used to copy selected entities via the API.

Markers

A set of new functions provide basic entity annotation functionality in the model window (`St7SetMarker()`, `St7ShowMarker()` and `St7HideMarker()` amongst others).

Results Contour File

A set of new functions, including `St7GeneratePlateContourFile()`, `St7LoadPlateContourFile()` and `St7GetPlateContourFileResult()`, provide a more efficient way to extract plate and brick element results when nodal averaged results are required over all, or most, of the elements in the model. Using contour files, the results are pre-calculated and stored for faster retrieval.