

Strand7 R3.1 Feature Summary



CONTENTS

Introduction	1
System Requirements	1
Architecture	1
64-bit	1
Multi-Core Support	1
DirectX	1
Files	3
Model ST7 Files	3
Read-only Files	3
File Sharing	4
ST7V – View-only File	4
Clone Function	4
Result Files	4
User Interface – General	5
Startup Screen	5
Model Window	5
Icon Size and High DPI	7
Dynamic Rotation	7
3D Mouse	8
Touch	9
Scroll Wheel Button Displacement Scaling	9
Context Specific Mouse Cursor	10
Quick Views and Quick Displacement Scale	11
Pin Entity Inspector	11
Help Icon	11
User Interface – Productivity Features	12
Model Bounding Box	12
Retrieve Icon	12
Window Background	13
Edit Box Equations	13
Grids – Evaluate Selected Cells	14
Load and Freedom Cases – Case View vs Grid View	15
Group Tree Functions	16
Dialog Box Menus	
Drag-reorder Lists	
Right-click Functions	



Backup Location	
Help File	21
Colour Map	21
Selection Tools	22
Entity Select Toggles Multi-select	22
Entity Select	
Select by Entity Set	
Select by Entity ID or Number	
Select Plate Edges	23
Select Link Types / Select Face Types	23
Select by 3D Region	23
Right-click Entity Select Toggles	24
Rendering	24
Display Options/Defaults	25
Fill vs Outline	25
Geometry Faces as Solid	
Plate – Average Normals	26
Groups as Parent Colour	26
Detect	27
Modelling	
Undo/Redo	
Entity Sets	
Load Cases	
Equivalent Seismic as a General Load Case	
Create Load Cases from Combinations	
Influence Case Names	
Tables	
New Table Types	
Table Units	
Convert Time History to Response Spectrum	
Elements	
Multi Point Links (MPL)	
Attachment Link	
Reaction MPL	
Links – Nonlinear Control Freedom Case	
Automatic Sector-Symmetry Links	
Beam Section Properties and Shear Stress	
Cavity Attribute	
Initial Cable Droop	42



Weightless Cable	42
Load Path Normal	43
Load Path Sets	43
Load Path Time Tables	43
Plane Strain Thickness	44
Property-based Rayleigh Damping	45
Tri3 Membrane-Bending Coupling	45
Point Contact Elements	46
Heat Transfer for Line Elements	47
Attributes – Modelling Features	47
Store Formulas	47
Attribute Bulk-edit Functions	48
Attributes – New or Enhanced	48
Beam Local Distributed Force	48
Beam Pre-Curvature	49
Global Distributed Load on Cables	49
Beam Side Attachment	49
Plate Surface-dependent Attributes	50
Support Attributes – Lateral Stiffness, Gap, Bearing Capacity, Direction	50
Plate Pre-Curvature	51
Plate Stiffness/Mass Factors	51
Plate Edge Global Pressure	51
Geometry Edge Attribute – Edge Beam	51
Geometry Edge Attribute – Edge Cluster	52
Geometry Edge Attribute – Divisions	52
Staging	53
Stage-dependent Soil Fluid Level	53
Stage Type – Reset	53
Property Switch with Birth Stage Inheritance	53
Features	54
Beam Geometry Library (BGL)	54
Plate Reinforced Concrete Module	55
Other Pre-Processing	57
SUMMARY/Property – Bulk-edit Mode	57
Assign to Property	57
Tools and Related	57
Variables	57
by Rotation	58
Cut Elements	



Extrude Plate Edges	59
Extrude Plate Edge Attributes	59
Mid-plane Thin Solids	59
Repair Tri3 Mesh	60
Direct Solid Mesh	60
Geometry Copy or Move by Thickness	61
Subdivide Beams by Length	61
Convert Beam Offsets to Rigid Links	62
Insert Node on Entity	62
Geometry Subdivide Edges	63
Geometry Detach Faces	63
Align Face Normals	64
Surface Mesher	64
Tools – Productivity Features	65
Increment Multiplier	65
Subdivide Source / Tri / Quad	65
Clean Geometry – Clean Groups Independently	66
Solvers and Related	66
Cable Inertia Case	66
Soil In-situ Options	67
Eigenvector Convergence for Natural Frequency and Linear Buckling Analysis	67
Mode Participation in Natural Frequency Analysis	68
Spectral Response Analysis Multi-case and Append Options	68
Linear Buckling Analysis – Constant + Variable Case	69
Linear Buckling and Natural Frequency Follower Load [KG]	70
Position Offset vs Time Tables – Quasi-static and Nonlinear Transient Dynamic Analysis	72
Factor vs Velocity Tables – Nonlinear Transient Dynamic Analysis	74
Relative Velocity Tables – Nonlinear Transient Dynamic Analysis	75
Nonlinear Static Analysis – Pseudo Time and Reset	77
Automatic Sub-stepping Options	77
Transient Dynamic Analysis – Base Displacement, Base Velocity and Base Acceleration	78
Harmonic Response Analysis – Base Excitation	78
Non-structural Mass in Dynamic Analysis	79
Solver Generates Combinations	79
Appending Time-based Results	79
Results	80
Opening Result Files	80
Element Energy Integral	80
Strain Measures	81



	-
Links as Elements	
Spectral Response Node Inertia Force	82
Post-processing	82
Extrapolation	82
Brick Cutting Planes	83
Subtract Support Reactions	
Linear Static + Harmonic Response Combinations	85
Natural Frequency Analysis Mode Shape Normalisation	86
Envelopes	86
UCS Node Reactions	
User-defined results equations	88
Graphs – standard size windows, types, zoom, double-click point	89
Result Case Information	90
Log File Viewer	90
Import/Export	91
DXF Import	
Rhino 3dm File Import	91
API	91
Functions	
Headers	91
DLL Solver	
Tools Functions	92
Markers	92
Results Contour File	92



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

Strand7 R3.1 Feature Summary



(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).



0 Nodes 0 Beams 0 Plates 0 Bricks 0 Links 10,448 Vertices 7,206 Faces 0 Paths mm N t MPa C J (-167;41;0) Model

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



Strand7 R3.1 Feature Summary

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.

Strand

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.

2		Export Model - Strand7 Text File	×
Export F	file		
Name:	F:\Building\Buil	ding.txt	
Type:	Strand7 Text File		~
Options	5		
Version	I	Freedom Case Attributes (Excluding Node Restraints)	
Curren	it Release 3.1.x 🗸 🗸	1: Fixed Footings	~
Curren	t Release 3.1.x		
Previou	is Release 2.4.x		
Previou	is Release 2.3.x	<u>Export</u> <u>Cance</u>	el
Previou	is Release 2.2.x		
Previou	is Release 2.1.x		
Previou	is Release 1.06		

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.

Open Strand7 file					×
\leftarrow \rightarrow \checkmark \uparrow \bullet GC5 \rightarrow Local Disk (C:) \rightarrow St7 I	Models	~	Ū ۱	Search St7 Models	
Organise 👻 New folder				III	?
Name Name	Date modified	Туре	Size		
Building.st7	7/11/2013 13:29	ST7 File	512 KB		
			_		
File <u>n</u> ame: Building.st7			~	Strand7 binary file (*.st7)	\sim
		Open read-onl	у [<u>O</u> pen Cance	el

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.

Strand7 R3.1 Feature Summary



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.

?	Save View-only Copy			
G:\ST7 Files\Building_F	R3 - Copy - 1.st7v			
Quantity visibility and op	tions			
Coordinates	SUMMARY Tab:	LAYOUTS Tab:		
TEXT Tab	Attribute	✓ Tables		
CASES Tab	Property	Plies		
Properties Dialog	🗹 Model	✓ Laminates		
LISTINGS Tab		Plate RC		
Allow saving		Creep		
		Paths		
		Cavities		
		<u>S</u> a	ve <u>C</u> lose	

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.

Strand7 R3.1 Feature Summary



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.

Strand7 R3.1 Feature Summary



👬 Strand7 🗠 🕋 🔚 🛍 🖺 👾 🕼 ? VISUAL TEXT CASES LAYOUTS NOTES SUMMARY SOLVERS BROWSE 🛛 Meint	⊡ - □ ×
Global Insert Edit Attributes Tools Detect Unity View 1: Coad Case 1 🗢 4: Freedom Case 1 - Global XYZ [Cartesian]	~
🙏 UCS × 🇱 Properties × 🥵 Groups × 🔞 Sets × 👔 Stages × 😰 Viewports 😯 Droop . V 📲 Units	
☆/ 黄 @ 韻 ◇ ^ ◇ ^ × / 『 / □ 司 / □ □ □ □ / □ 章 ② 誹 諸 8 @ # * 12 8 8 10 *	
1	
49 •	
123	
	Y
ILIORI/Case 1 O Nodes: O Rams: O Distas: O Rinks: O Vartices: O Faces: O Raths: mm. N. + MPa. C. J. (0:00). Model	Z X

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

Strand7 R3.1 Feature Summary



(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.

?	St	trand7 Preferences - these ap	oply to all models	×
FOLDER LOCATIONS	GRAPHICAL USER INTERFACE	~		^
BACKUPS AND SAVING	Icon and font size (pixels) - chang	ge takes effect after restartin	g Strand7 [Current=Small (16)]	
DEFAULT UNITS	() Auto	O Medium (24)	Very large (40)	
INSPECT AND BROWSE 2D MOUSE IN VISUAL TAB	Small (16)	🔾 Large (32)	○ Huge (48)	
3D MOUSE IN VISUAL TAB	Title case main tabs			
GRAPHICAL USER INTERFACE	Always show mini-menus			
VISUAL TAB	Always show model name at t	op		
OPEN/CLOSE	Double-click to expand/compa	act menus		
OTHER	Open VISUAL/sub-tab when u	ising keyboard shortcuts		
	MODEL NAME FONT: Small	~		
	TOOLTIP DELAY: Default	~		
				<u>O</u> K

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.

Strand7 R3.1 Feature Summary



?	Strand7 Preferences - these apply to all models	×
FOLDER LOCATIONS	2D MOUSE IN VISUAL TAB 🗸	^
BACKLIPS AND SAVING	Left-click:	
DEFAULT UNITS	Rotates (activate Select by 2D Region to left-click select)	
INSPECT AND BROWSE	Selects (press Ctrl to left-click rotate)	
3D MOUSE IN VISUAL TAB	Right-click:	
WARNINGS	© Zooms	
GRAPHICAL USER INTERFACE	O Pans	
VISUAL TAB		
OPEN/CLOSE	Shows popula menu (press Ctrl to right click zoom)	
OTHER	Shows popup menu (press curi to right-circk zoom)	
	Right-click+Shift always shows popup menu.	
	Middle-click:	
	○ Scales deformation	
	Rotates (press Ctrl to scale deformation)	
	O Pans (press Ctrl to scale deformation)	
	Left+right-click:	
	Pans	
	○ Zooms	
	SCROLL WHEEL ZOOM DIRECTION: Forward 🗸	
		<u>О</u> К

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/.



Strand7 R3.1 Feature Summary

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**.

?	Strand7 Pref	ferences - these apply to all models	×
FOLDER LOCATIONS	3D MOUSE IN VISUAL TAB		^
BACKUPS AND SAVING	Driver: C:\WINDOWS\SYSTEM32	\siappdll.dll	
DEFAULT UNITS	Driver Version: 3DxWare Library		
2D MOUSE IN VISUAL TAB	Driver Date: December 6, 2017		
3D MOUSE IN VISUAL TAB	Device Name: SpaceNavigator		
	Look for device		
VISUAL TAB	Rotate one axis at a time		
DIALOGS	SENSITIVITY Medium		
OPEN/CLOSE			~
			<u>О</u> К

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
\searrow	Default Windows cursor – no specific function.
Ċ	Dynamic rotation with 2D mouse.
中令 一令	Dynamic zoom with 2D mouse.
œ€	Dynamic pan with 2D mouse.
۲	Double-click point for setting rotation origin.
ૡૺૢૺૺૺૼૼૺૼૺ	Dynamic displacement scaling with 2D mouse via scroll wheel button.
Ð	Explicit VISUAL/View/Pan function with 2D mouse.
₿€	Explicit VISUAL/View/Zoom in function (zoom window) with 2D mouse.
₿ Q	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).
····ē·····	Select by 2D region.
	Select by 3D region.
Q	Zoom in / zoom out in print preview mode.
	Equation editor.
Ś	Retrieve by hot pointer to dialogs such as the Attribute dialogs.
Ę	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.

Strand7 R3.1 Feature Summary

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.

4 Beam 1900 [ID 0] 🛛 🗙
> Type [Beam2]
> Property 19 [Hangers]
Section Type [Hollow Rectangle]
> Group [Model\Deck\Deck 6]
> Nodes [2] 2053; 433
> Principal Angle=180.0 deg
> Release - R
End1 Dir1=Released Dir2=Fixed Dir3=Fixed
> Inspector Data
End 1
Length=131.764

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.

Strand7 R3.1 Feature Summary



• • •	± ?	Pla	te Attributes	×
Pre-cu	urvature	e. ft		
⊻ ×	0.0			<i>f(x)</i>
🗹 у	0.0			f(x)
		y .	×	
🗌 Kee	p selec	tion		
<u>S</u> ca	le	A <u>d</u> d	D <u>e</u> lete	<u>A</u> pply

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.

Strand7 R3.1 Feature Summary



• • e	F 5	No	de Attribu	tes 🗙
Force:	Ibr			
∠ X	0.0			f(x)
🗸 ү	0.0			f(x)
∠ z	0.0			f(x)
Global	XYZ [C	[artesian]		~
🗌 Keej	o sele	ction		
<u>S</u> cal	e	A <u>d</u> d	D <u>e</u> lete	Apply

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.

?				View S	ettings			×		
Background	Axes	Rotation	Rotation Drawing Pre Numbers Post Numbers Free Edge							
Pre-processing Colour Gradient Add image Loose Colour Col										
Image										
Location	Centre	\sim	Size 0	0 % of	screen	width (0 for pixel	resolution)			
☐ Title label ☐ Crosshair	🗹 Loa	ad case labe	I 🗌 Freed	om case	label	Font				
Defaults						<u>O</u> K	<u>C</u> ancel	Apply		

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.

	?							Equation Editor	×
	cos(x)								^
									~
	Operator	s and cons	stants:					History:	+ ×
	+	- *	1	۸	()	E	sin(x)	
🐂 👻 🛃 ? Node Attributes 🗙	π	e L	TL	Α	TA	V 1	TV		
Force: N	Function	s (angles i	n degree	es):					
✓ X cos(X) ƒ(x)	Sqrt	Sqr	Sin	Co	s T	an	Min		
✓ Y 0.0 f(x)	Abs	Sign	Arcsin	Arco	os Ar	ctan	Max		
✓ Z 0.0 f(x)	Round	Trunc	Ln	Lo	g E	хр	Rand		
	IfPos	IfPosB	lfNeg	lfNe	gВ	lf			
Global XYZ [Cartesian]	Factor vs	Position ta	ables:						
Keep selection						\sim			
Scale Add Delete Apply									OK <u>C</u> ancel

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

Strand7 R3.1 Feature Summary



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.

?						Equation Editor		×
								^
								\vee
Operator	s and con	stants:				History:	+	×
+ π Function:	e C	R R n degrees	^ ():)	Ε	sin(x)		
Sqrt	Sqr	Sin	Cos	Tan	Min			
Abs	Sign	Arcsin	Arccos	Arctan	Max			
Round	Trunc	Ln	Log	Exp	Rand			
IfPos	IfPosB	lfNeg	lfNegB	lf				
						<u>O</u> K	<u>C</u> ano	el

Strand7 R3.1 Feature Summary



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.

	IAL TEXT CASES LAY	OUTS NOTES SUMMAR	Y SOLVERS BROWSE		Mesh1	I. □ – □ ×
Load Freedom Combination Harmonic Time E	nvelope Influence Con	bine Files				
🛄 💷 🗅 🗙 🗹 🗟 🖏 🗟 👘 🕒	2					
1: Load Case 1	REFERENCE/INITIAL T	EMPERATURE				
2: Load Case 2						
3: Load Case 3	25.0	С				
4: Load Case 4						
S: Load Case 5	APPLY NODAL/GLOBA	L ACCELERATION TO				
7: Load Case 7						
8: Load Case 8						
9: Load Case 9	Non-structural mass					
10: Load Case 10						
11: Load Case 11	GLOBAL INERTIA LOA	D				
12: Load Case 12	O None					
13: Load Case 13						
14: Load Case 14	Gravity					
15: Load Case 15	Acceleration and Vel	ocity				
16: Load Case 16	Seismic					
17: Load Case 17						
18: Load Case 18	LINEAR ACCELERATIC	NV		?		
19: Load Case 19						
20: Load Case 20	AX (mm/s²)	AY (mm/s²)	AZ (mm/s²)			
21: Load Case 21	0.0	0.0	0.0			
22: Load Case 22						
23: Load Case 23	ANGULAR VELOCITY					
24: Load Case 24	VV (deg/s)	W (deg (s)	\/7 (deg(s)			
25: Load Case 25	vx (deg/s)	vr (deg/s)	vz (deg/s)			
20: Load Case 26	0.0	0.0	0.0			
27: Load Case 27						
20. Load Case 20	ANGULAR ACCELERA	TION				
30: Load Case 30	WX (deg/s ²)	WY (deg/s ²)	WZ (deg/s ²)			
31: Load Case 31	0.0	0.0	0.0			
32: Load Case 32	0.0	0.0	0.0			
33: Load Case 33						
34: Load Case 34	UNIGIN FUR ANGULA	R QUANTITIES				
35: Load Case 35	X (mm)	Y (mm)	Z (mm)			
36: Load Case 36	0.0	0.0	0.0			
37: Load Case 37 🗸 🗸						
0 Nodes 0 Beams 0 Plates 0 Bricks 0 Links 0	Vertices 0 Faces 0 Pat	hs mm N t MPa	C J (0;0;0) Model	Total Cases: 38		

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.

	D 🗙 🛛	1 3 14 I	* .													
inert	a cases Gravi	ty cases Acceler	ation/Velocity cases	Seismic cases												
I Cas	e 7	,														
ase	Name	Temperature (C)	Structural Mass	Non-structural Mass	AX (mm/s ²)	AY (mm/s ²)	AZ (mm/s ²)	VX (deg/s)	VY (deg/s)	VZ (deg/s)	WX (deg/s ²)	WY (deg/s ²)	WZ (deg/s ²)	X (mm)	Y (mm)	Z (mm)
7	Load Case 7	25.0	\checkmark	~	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
10	Load Case 10	25.0	~	~	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
14	Load Case 14	25.0	~	~	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
18	Load Case 18	25.0	~	~	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
19	Load Case 19	25.0	\checkmark	\checkmark	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
20	Load Case 20	25.0	\checkmark	~	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
21	Load Case 21	25.0	~	\checkmark	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
22	Load Case 22	25.0	\checkmark	\checkmark	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
23	Load Case 23	25.0	\checkmark	\checkmark	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
24	Load Case 24	25.0	\checkmark	\checkmark	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
25	Load Case 25	25.0	\checkmark	\checkmark	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
26	Load Case 26	25.0	\checkmark	~	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
27	Load Case 27	25.0	\checkmark	\checkmark	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
28	Load Case 28	25.0	\checkmark	\checkmark	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
29	Load Case 29	25.0	\checkmark	~	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
30	Load Case 30	25.0	\checkmark	\checkmark	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
31	Load Case 31	25.0	\checkmark	\checkmark	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
32	Load Case 32	25.0	\checkmark	\checkmark	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
33	Load Case 33	25.0	\checkmark	\checkmark	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
34	Load Case 34	25.0	\checkmark	\checkmark	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
35	Load Case 35	25.0	\checkmark	\checkmark	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
36	Load Case 36	25.0	\checkmark	\checkmark	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
37	Load Case 37	25.0	\checkmark	\checkmark	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
38	Load Case 38	25.0	\checkmark	\checkmark	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

O Nodes O Beams O Plates O Bricks O Links O Vertices O Faces O Paths mm N t MPa C J (0;0;0) Model Total Cases: 38

Strand7 R3.1 Feature Summary



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.





Strand7 R3.1 Feature Summary



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 reparented to the nearest parent in the hierarchy so that the group tree remains connected.

	Delete Multiple Group	5	×
Model			^
Ground Floo	r		
- Columns			
Lift Core			
- 1st Floor			
- Columns			
- 🗙 Perimeter	Beams		
- Cross Bear	ns		
🛛 🗙 Floor Slab			
- Lift Core			
🛛 🗙 Beam/Colu	umn Connection		
2nd Floor			
- Columns			
🛛 🗙 Perimeter	Beams		
- Cross Bear	ms		
- Floor Slab			
- Lift Core			
Beam/Colu	umn Connection		
- 3rd Floor			~
		Delete	Close

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.



Strand7 R3.1 Feature Summary



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.

		' * ?	Face Attributes	- ×
Move by	Rotation X	Property Type Group		f(x) f(x)
by Increment	×	<u>I</u> D Thickness		f(x)
by <u>R</u> otation by <u>P</u> rojection	point	Offset Support	<none></none>	~
by <u>T</u> hickness by <u>M</u> irror		Temperature Gr	adient Itants	
to <u>A</u> bsolute	points	Global Pressure	Mass	
to <u>U</u> CS Intersection to <u>O</u> rigin	Apply	Heat Transfer	•	~
to U <u>C</u> S Plane		Acacriment	<u>A</u> p	ply

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.

1: Self Weight	
2: Cable Tension - Launch 1	
3: Cable Tension - Launch 2	N
4: Cable Tension - Launch 3	157
5: Cable Tension - Launch 4	
6: Cable Tension - Launch 5	

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

÷		Reorder Tables								
📥 Ascen	Ascending 🔻 Descending									
1										
Order	Current ID 🔺	New ID	Current Name	New Name						
1	1	1	Vertical PSD Curve (mm/s2)2/Hz	Vertical PSD Curve (mm/s2)2/Hz						
2	2	2	Transverse PSD Curve (mm/s2)2/Hz	Transverse PSD Curve (mm/s2)2/Hz						
3	3	3	Longitudinal PSD Curve (mm/s2)2/Hz	Longitudinal PSD Curve (mm/s2)2/Hz						
					OK Cancel					

Right-click Functions

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

Checkbox lists

Select, unselect or toggle options.

Strand7 R3.1 Feature Summary





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

Max
 Min
 Undo
 Qut
 Copy
 Paste
 Qelete selected
 X Clear all
 Select all

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.

MAT	'ERIAL Y	/		
Con	crete:	Assign default concrete (AS 3600)	•	Peduction factor:
Ec	0.0	Assign default steel (AS 3600)		fy = 400 MPa 0.0
εc	0.0	Clear concrete		fy = 500 MPa
fc	0.0	Clear steel		fy = 600 MPa
α	0.0			

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.

GLOBAL STIFFNESS MATRIX UPDATE							
 Every iteration First two iterations 		Set User Defaults					
 First iteration Automatic 		Save as User Defaults]				
1.0	Max displacem						
1.0	Max residual ch	nange					
5 🗘	Max update int	erval					

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.

Strand7[®]

Strand7 R3.1 Feature Summary



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.

Strand7	× 🗅 🗰 🖬 🛋	- 🖧 ?	VISUAL TEXT C	ASES LAYOUTS NOTES	SUMMARY S	OLVERS BROWSE	Building	1_R3		II × □ – □ ×
Global In:	sert Edit Attrib	utes Tools Deter	ct Utility View	1: Dead Load, G		 1: Fixed Footing: 	s 🗾 🖊	✓ Glo	Open file location	~
- Node Y	🏏 Beam 🗠 🎦 Pla	te 🔨 🍟 Brick 🗠 🥍 I	Link 🗡 🍡 Vertex 🗠	🔚 Edge 🐃 🍋 Face 🐃 📩	Path 👻 🏪 Scale	$f(x)$ Formulas \vee			<u>C</u> opy file name	
☆ ~ M	o 📰 🗠 🗸	~ * X * 🗖	/, 0, 0, /,	• 🗒 📼 🖥 🖌	🗖 🗗 🕱 🛙	📙 🙆 #* 🖫* 🖻* 🛛	Ēř			
	Home Load	Matrix Para	ameters File	Results Filterin	a					
	Paruli Fila				-					
	C:\St7.Mod	iels\SHB 23.LSA	1							
1	Log De	-								
	C:\St7 Mod	tels\SHB 23.LSL								
	Star Open	file location								
	C:\St7 Mod	lels\SHB_23.SRF	F							

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.

?	Preferences - these apply to all models	×
FOLDER LOCATIONS HELP FILE	BACKUPS AND SAVING V	^
BACKUPS AND SAVING	Autosave every 10 🗘 minutes (set 0 to disable)	
DEFAULT UNITS INSPECT AND BROWSE	Make numbered backup file on save (recommended)	
2D MOUSE	O Store backups in model folder	
3D MOUSE WARNINGS	Store backups in specified folder	
GRAPHICAL USER INTERFACE	C:\St7 Models\All Backups\	
OTHER	Remember "Save Copy" location	~
		<u>о</u> к

Strand7 R3.1 Feature Summary



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.

?	Preferences - these apply to all models				×
FOLDER LOCATIONS HELP FILE BACKUPS AND SAVING DEFAULT UNITS INSPECT AND BROWSE 2D MOUSE 3D MOUSE WARNINGS GRAPHICAL USER INTERFACE OTHER	FOLDER LOCATIONS Scratch files (Free space = 87.68 GB) C:\St7-Scratch\ Property libraries C:\Strand7-R3\Data4\ Display options	1 Number 1 2	Colour Map Colour	×	
	C:\Users\GC\Strand7\Cfg\Settings.cfg3 Colour map C:\Users\GC\Strand7\Cfg\St7Colours.map	4 5 6 7 <u>D</u> efaults	<u>S</u> ave	↓ <u>C</u> lose	

Strand7 R3.1 Feature Summary



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 Set	×
1: CCC 2: DDD 3: FFF 4: GGG 5: AAA 6: HHH 7: EEE 8: III 9: BBB		
<u>0</u> k	<u>C</u> lose	Apply

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 by Entity Number X						
Entity Number Selection						
Range separator = ""						
List separator = ;						
(Example: 1;3;512)						
Select entity						
Select node/vertex on entity						
<u>O</u> K <u>C</u> lose <u>A</u> pply						

Strand7 R3.1 Feature Summary



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 Link Types	×
Master	-Slave	
	na	
Pinned		
Rigid		
Shrink		
2-Point	t mant	
	plated MPI	
User-d	efined MPL	
Reactio	on MPL	
Rigid N	/IPL	
Pinned	MPL	
Master	-Slave MPL	
Select I	ink	
Select 1	nodes on link	
<u>0</u>	K <u>C</u> lose <u>A</u>	Apply

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.

Global XYZ [Ca	ect by 3D Region artesian]	×	Global XYZ	Select by 3D Region Z [Cartesian]	× ~	Free Cylindr	elect by 3D Region ical	×
Previous	Viewports 👻	Clear	Previous	s Viewports Y	Clear	Previous	Viewports Y	Clear
Extend	<u>1</u> : Viewport <u>2</u> : Viewport <u>3</u> : Viewport	+ + +	Extend	□ +X □ +Y □ +Z	** **	Theta Invert Wrap	Extend -R +R -Z +Z	** **
] Select parti 2 Auto-highli <u>O</u> K	ally enclosed ight position	Apply	☐ Select p ☑ Auto-hi 	partially enclosed ighlight position K <u>C</u> lose	Apply	☐ Select par ☑ Auto-high <u>_</u> K	tially enclosed nlight position	Apply

Strand7 R3.1 Feature Summary



Right-click Entity Select Toggles

Strand7 × 🗅 💼	8 🛋 🗈 🗄	000 👌 ? 🚺	UAL TEXT	CASES LAYOUTS	NOTES	SUMMARY	SOLVERS	BROWSE	Building-Seismic	□ × □
Global Insert Edit	Attributes 1	Tools Detect	Utility Vie	w 4: Seismic Case	1	~	1: Fixed Fo	otings	 Global XYZ [Cartesian] 	~
🖕 UCS ≚ 🏭 Properties 🐃 Groups 👻 🚳 Sets 👻 👔 Stages 👻 🗊 Viewports 👽 Droop -Y 📲 Units										
🌣 🖌 🖸 🔛	$\sim \sim \sim \sim$	× · · /	/ . . ,	/ • 🔍 🖂 🖥	1¥ 🖉	🗆 🗗 🗃	8	⊚ #∙	°av ਛv ©r	

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.

?				Atti	ribute Display S	ettings			×
🗹 🗆 123 123 12 t2 🦯 ,	× Б. Т	11							
·· 🗡 📭 🗊 📼									Face Load - Normal Pressure
4227327									
PLATE/FACE ATTRIBUTE	Show	Values	C1	C2	C3	Thickness	Vector Type	^	
Age at First Loading	~								1
Attachment - Edge	\checkmark					2			
Attachment - Face	~					2			
Cavity	~					1			
Convection - Edge	\checkmark					1			
Convection - Face	\checkmark					1			
Edge Load - Global Pressure	\checkmark					1	Components		\sim * >
Edge Load - Normal Pressure	\checkmark					1			
Edge Load - Shear Stress	\checkmark					1			Ť
Edge Load - Transverse Shear Stress	\checkmark					1			C1-Attribute colour
Edge Release	\checkmark					1			
Face Load - Global Pressure	\checkmark					1	Components		
Face Load - Normal Pressure	\checkmark					1			
Face Load - Shear Stress	\checkmark					1	Components		
Heat Flux	\checkmark					1			
Heat Source	\checkmark								
Load Patch	\checkmark					1		v	
<								>	Defaults OK Cancel Apply

Strand7 R3.1 Feature Summary



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.

?	Set Mode	Defaults	×								
UIEW OPTIC	ONS		^								
	ground tab										
🗸 Axes	tab										
- 🗸 Rota	tion tab										
··· Draw	- Drawing tab										
Pre n	umbers tab										
Free	edge tab										
Selec	tion tab										
ENTITY DISP	PLAY										
- Node	e tab										
- Beam	i tab										
Plate	tab										
Brick	tab										
- Link	tab										
Verte	Vertex tab										
··· Face	Face tab										
- Path	tab										
PRE CONTO	OURS - BEAM										
PRE CONTO	OURS - PLATE										
PRE CONTO	OURS - BRICK										
	DISPLAY										
RESULTS O	PTIONS										
🗸 Shov	v/hide tab										
🗸 Post	numbers tab										
Com	binations tab		~								
	Apply User Defaults	Apply Factory Defaults	<u>C</u> lose								

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.

?	Entity Displ	ay Settings
/ 🔳 🗐	* • 🗉 🛰	
Fill		Outline
🔿 None 🖂 Ligh	ting	None Lighting
O Property	-	O Property
Group		O Group
		Global
		Thickness 1 0
	-	Style
Contour		Facet Edge
Contour Type		
<none></none>		Settings
in the second se		
Labels	Style	Extras
<none> ~</none>	Surface	Move to Offset
	Solid	Show Offset Nodes
Plate Font	Face Nodes	Draw Axes (0) or Ply (#)
	Average Normals	Shripk (%)
	Tol (deg) 25 🗘	
rauits		<u>Ok</u> <u>Cancel</u> <u>Apply</u>

Strand7 R3.1 Feature Summary



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.



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.

Strand7 R3.1 Feature Summary

Strand7°



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.

Global	Insert	Edit	Attributes	Tools	Detect	Utility	View	
🦯 Beam	End 🗡	Beam	Junction 🤺	^K Link Er	nd 🗌 Pla	ate Edge	: ڷ Plate Junction 🗍 Brick Edge 🗂 Brick Face 🕠 Face Edge 🗍 Face Junctio	n

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.

Strand7 R3.1 Feature Summary





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.

0?	? Storage Bin_R3 - Entity Sets										>	
D X 4	隆 🗹 🕀 😑 🔘											
Show	Set name	Nodes	Beams	Plates	Bricks	Links	Paths	Vertices	Faces	Coedges	Reaction MPL	Path Sets
	1: Cross Braces		56									
	2: Ring Beam			1440								
	3: Conical Hopper			5040								
Show if contained in any shown set [UNION]												
Show e	Show antitics in parts Kons calaction											

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

Show if contained in any shown set [UNION]	
Show if contained in all shown sets [INTERSECTION]	
Show if contained only in all shown sets [AND]	
Hide if contained in any hidden set - show others	
Hide if contained in all hidden sets - show others	

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 dragand-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.
Strand7 R3.1 Feature Summary



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.

📑 Strand7 🗠 📄 🔚 📑 🛄 🖏 🤇 ? VISU	JAL TEXT CASES LAYOUTS NOTES SUMMARY SOLVERS BROWSE Chimney - Remedial 3 - Tension-Only-A	II - 🗆 X
Load Freedom Combination Harmonic Time E	nvelope Influence Combine Files	
🔲 🔲 🗅 🗙 🗭 🛸 🔜 🛤	?	
1: Gravity		^
2: Seismic		
	○ None	
	Gravity	
	Acceleration and Velocity	
	Seismic	
	DIRECTION OF GRAVITY ?	
	Οχ. Ογ. ΘΖ	
	ACCELERATION DUE TO GRAVITY V	
	-9806.65 mm/s ²	
	VX 1.0	
	VY 0.0	
	BASE ELEVATION	
	h0 -1800.0 mm	
	SEISMIC PARAMETERS	
	Structural mass Elevation exponent	
	α 1.0 k 1.14288	
	Non-structural mass Base shear	
	φ 1.0 β 0.114402	
	Apply dynamic factors to non-structural mass attributes	~
62,392 Nodes 576 Beams 0 Plates 50,460 Bricks	2,378 Links 0 Vertices 0 Faces 0 Paths mm N t MPa C J (-76;-1;-14) Model Total Cases: 2	.:



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.

	Building-Seismi	c		
📑 Strand7 🖌 🛅 🔚 📑 😳	🗟 ? VISUAL TEXT CASES LAYOUTS	NOTES SUMMARY	SOLVERS BROWSE	I - 🗆 ×
Load Freedom Combination Harm	ionic Tine Envelope Influence Combine Fi	les		
👬 👬 👗 🗹 🗆 🗹 👘 🕒				
Combined : 1.25*G + 1.5*Q + 1.25*Wu				
	1	2	3	
NAME	Combined : 1.25*G + 1.5*Q + 1.25*Wu	Combined: 1.25*G	SRSS	
GENERATE	✓	\checkmark	✓ ⁴	
1: Dead Load, G [1: Fixed Footings]	1.25	1.0		
2: Floor Live Loads, Q [1: Fixed Footings]	1.5	2.0	3.0	
3: Wind Loading, Wu [1: Fixed Footings]	1.25		4.0	
4: Seismic Case 1 [1: Fixed Footings]				
5: Dead Load, G [2: Freedom Case 2]	2.0		4.0	
6: Floor Live Loads, Q [2: Freedom Case 2]		3.0		
7: Wind Loading, Wu [2: Freedom Case 2]			5.0	
8: Seismic Case 1 [2: Freedom Case 2]				
2,496 Nodes 72 Beams 2,418 Plates	0 Bricks 0 Links 0 Vertices 0 Faces 0 F	Paths mm N t	MPa C J (-145;45;	180) Model Total Combir

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.

			AL TEXT	CASES LA	YOUIS	NOTES SUMMA	RY SOLVERS BROW	/SE	Curve	d Concrete Brid	dge - A55100
Load Freedom Combination	n Harmo	nic Time En	ivelope	nfluence Co	mbine Fil	es					
Model Arroup	Gro	유 유 🛪	비호 3	E 🛛 🖂		*					
G2-3	ups	E:\Influ	ence Tests	P12\Parallel-A	Curved (Concrete Bridge - /	AS5100.LIA				
- 🜐 K Frames		Case	Entity	Number	Label	1	/ariable	Load Case Name	Freedom Case Name	Include	Generated Load Case Name
- 🖽 Shear studs	ean	1	Node	2187		Node(2187) D(X)	[Global XYZ]	Load Case 1	Freedom Case 1	~	[Node(2187) D(X)[Global XYZ]] [Load Case 1] [Freedom Case 1]
K frame	P	2	Node	2187		Node(2187) D(Y)	[Global XYZ]	Load Case 1	Freedom Case 1	~	[Node(2187) D(Y)[Global XYZ]] [Load Case 1] [Freedom Case 1]
	ope	3	Beam	117		Beam(117) End	1SF1	Load Case 1	Freedom Case 1	~	[Beam(117) End 1 SF1] [Load Case 1] [Freedom Case 1]
	4	4	Beam	117		Beam(117) End	1 SF2	Load Case 1	Freedom Case 1	~	[Beam(117) End 1 SF2] [Load Case 1] [Freedom Case 1]
- # G3	-	5	Beam	237		Beam(237) End		Lond Core 1	Freedom Case 1	\checkmark	[Beam(237) End 1 SF1] [Load Case 1] [Freedom Case 1]
- 🛱 G2	Plat	6	Beam	237		Beam(237) End	Load case na	me options 🛛 🗙	Freedom Case 1	\checkmark	[Beam(237) End 1 SF2] [Load Case 1] [Freedom Case 1]
- 🗍 G1	G1 9 7 Be	Beam	357		Beam(357) End	- I abal		Freedom Case 1	~	[Beam(357) End 1 SF1] [Load Case 1] [Freedom Case 1]	
- 🜐 concrete	ope	8	Beam	357		Beam(357) End	Variable		Freedom Case 1	~	[Beam(357) End 1 SF2] [Load Case 1] [Freedom Case 1]
- 🖽 top reo	4	9	Beam	477		Beam(477) End			Freedom Case 1	~	[Beam(477) End 1 SF1] [Load Case 1] [Freedom Case 1]
🗄 💾 bottom reo		10	Beam	477		Beam(477) End			Freedom Case 1	\checkmark	[Beam(477) End 1 SF2] [Load Case 1] [Freedom Case 1]
		11	Beam	597		Beam(597) End	Freedom Case N	ime	Freedom Case 1	~	[Beam(597) End 1 SF1] [Load Case 1] [Freedom Case 1]
		12	Beam	597		Beam(597) End	Response Type		Freedom Case 1	~	[Beam(597) End 1 SF2] [Load Case 1] [Freedom Case 1]
		13	Beam	717		Beam(717) End		OK Cancel	Freedom Case 1	~	[Beam(717) End 1 SF1] [Load Case 1] [Freedom Case 1]
		14	Beam	717		Beam(717) End			Freedom Case 1	\checkmark	[Beam(717) End 1 SF2] [Load Case 1] [Freedom Case 1]
		15	Beam	837		Beam(837) End	1 SF 1	Load Case 1	Freedom Case 1	~	[Beam(837) End 1 SF1] [Load Case 1] [Freedom Case 1]
		16	Beam	837		Beam(837) End	1 SF2	Load Case 1	Freedom Case 1	~	[Beam(837) End 1 SF2] [Load Case 1] [Freedom Case 1]
		17	Beam	957		Beam(957) End	1 SF 1	Load Case 1	Freedom Case 1	~	[Beam(957) End 1 SF1] [Load Case 1] [Freedom Case 1]
gle-variable (Min)	0	18	Beam	957		Beam(957) End	1 SF2	Load Case 1	Freedom Case 1	\checkmark	[Beam(957) End 1 SF2] [Load Case 1] [Freedom Case 1]
gle-variable (Max)	ases	19	Plate	1980		Plate(1980) FTT	[Curved Bridge Centre]	Load Case 1	Freedom Case 1	~	[Plate(1980) FTT[Curved Bridge Centre]] [Load Case 1] [Freedom Ca
		20	Plate	2340		Plate(2340) FTT	[Curved Bridge Centre]	Load Case 1	Freedom Case 1	~	[Plate(2340) FTT[Curved Bridge Centre]] [Load Case 1] [Freedom Ca
		21	Plate	2460		Plate(2460) FTT	[Curved Bridge Centre]	Load Case 1	Freedom Case 1	~	[Plate(2460) FTT[Curved Bridge Centre]] [Load Case 1] [Freedom Ca
		22	Plate	2580		Plate(2580) FTT	[Curved Bridge Centre]	Load Case 1	Freedom Case 1	~	[Plate(2580) FTT[Curved Bridge Centre]] [Load Case 1] [Freedom Ca
		72 { + + +	Single	-variable (Min) Single	-variable (Max)	Curved Bridge Centrel	Load Care 1	Freedom Case 1	./	[Diste(3787) ETT[() unual Bridge Centre]] [] and Care 1] [Freedom Ca

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.

🗅 🔁 🗙 😤 🗹 🗮

FACTOR VS POSITION

FACTOR VS VELOCITY

FACTOR VSTEMPERATURE FACTOR VS TIME FACTOR VS FREQUENCY/PERIOD

Strand7 R3.1 Feature Summary



		-																
VELOCITY VS TIME		4	0.03	-0.002661			L		- 1	'								
ACCELERATION VS TIME [17]		5	0.04	-0.003907					- 1 d	1	_							
STRAIN VS TIME		6	0.05	-0.004741					- 4 11	1 5								
TEMPERATURE VS TIME		7	0.06	-0.004892		-	-				4							
STRESS VS STRAIN		8	0.07	-0.004562		-		d i	64 (S 🗄 I	i i i	din I -		4.4.4	1.1				
FORCE VS DISPLACEMENT		9	0.08	-0.00381		0.0	الصيعان		61 (A A I	國加強		.	145.4	114	in a state	in the second	and the	
FORCE VS VELOCITY		10	0.09	-0.002756			1.00		'엄마 요네		1.31	i cheri	N 등 등 위 (7 1	(all a la second		1.410.01	
MOMENT VS CURVATURE		11	0.1	-0.001592		2		5.4	1翻 排册	• F	8"5	- 1 -	11.					
MOMENT VS ROTATION		12	0.11	-0.000131			-		- 19 88	14	- E							
9: Japanese Standard Earthquake Motion, Type211	^	13	0.12	0.001499					- 1 8	11								
10: Japanese Standard Earthquake Motion, Type21	2	14	0.13	0.002589			-		- 11									
11: Japanese Standard Earthquake Motion, Type21	З	15	0.14	0.002878					- 1°	•								
12: Japanese Standard Earthquake Motion, Type22	1	16	0.15	0.002627														
13: Japanese Standard Earthquake Motion, Type22	2	17	0.16	0.002025														
14: Japanese Standard Earthquake Motion, Type22	3	18	0.17	0.001056														
15: Japanese Standard Earthquake Motion, Type23	1	19	0.18	-0.00046	~	-1.0			1			1						
16: Japanese Standard Earthquake Motion, Type23	2						0.0			10.0			2	0.0			30	.0
17: Japanese Standard Earthquake Motion, Type23	3	Accelerati	on units:	Gravity units (g)	\sim							Time [s]						
3,960 Nodes 0 Beams 3,600 Plates 0 Bricks	0 Li	nks 0 Ver	tices 0 Fa	ces 0 Paths mm N	t MPa	С	J (0;0;0) Model										

Table Units

In addition to the model units option (e.g., mm/s²) 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.

?	Response Spectrum Table from Time History Table	×
Table name El Ce	entro	
Damping ratio		
○ Constant	0.05	
Rayleigh	₩ Factors	
OTable		\sim
Period range (s)	5.0	
Steps	100 🗘	<u>O</u> K
Total time factor	3	<u>C</u> ancel

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).

Strand7 R3.1 Feature Summary





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.

	🖽 Y ? Attach Parts 🗙
	Attach Parts Source Elements
	Visible Only
	○ Visible and Selected
	O All Elements
	Target Elements
? Insert Link X	Visible Only
Attachment ~	O All Elements
Couple	Brick Targets
Translations	Attach to free faces only
○ Rotations	Attach to all faces
⊖ Both	Attach inside bricks
Local position	
u 0.0	Maximum Merging Angle (deg) 30
v 0.0	Delete existing attachments
.0.0 O.0	Keep selection

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.

Strand7 R3.1 Feature Summary

Strand7°



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 *Left* entity set. The brick elements to the right of the link are assigned to the *Left* entity set. The brick in the mesh, *All*, is also defined.

Strand7 R3.1 Feature Summary





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	МХ	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.

?	Insert Link	×
Shrink		~
Shrink Dire	ections	
<mark>∠ D</mark> X		
V DY		
DZ 💟		
Nonlinear Co	ontrol Freedom Case	
1: Fixed Foo	otings	~

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 precalculated 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:

$$\begin{aligned} \tau_{31} &= F_1 \varphi_1 + F_2 \varphi_2 + T \varphi_3 \\ \tau_{32} &= F_2 \varphi_4 + F_1 \varphi_5 + T \varphi_6 \end{aligned}$$

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).

Strand7 R3.1 Feature Summary





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.

18 A.	Information	×
	Unit shear force 1 = 0.9956	
	Unit shear force 2 = 1.0036	
U	Unit torque = 0.9688	
	MNL quality = Good	
	<u>O</u> K	

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.



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.

📫 🕩 ?	Mesh1 - 1 Beam Property	×
Image: A marked and A marked	👷 🛼 🗹 🧮 📽 號 🇃 🗈 🛨 🕩	
1: Beam Property 1	Unknown Material	
	Beam	~
	Material Nonlinear Damping Tables Shape Properties	
	Temperature/Time Elastic Material Elastic-Plastic Mield Criterion Fibre Hardening Tresca Stress vs Strain Table von Mises 1: Typical tensile stress-strain curve for 6061-T6 aluminium alloy sheet (REF: MIL-HDBK-SE) Use Moment vs Curvature tables for bending Moment vs Curvature Table Plane 1	~
	<none></none>	\leq
	Moment vs Curvature Table Plane 2 <none></none>	✓ 🗸

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.

Strand7 R3.1 Feature Summary





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



Strand7 R3.1 Feature Summary



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.

📑 Strand7 🗠 🛅 🔚 📑 🛗 🖏 ? VIS	JAL TEXT CASES LAYOUTS NOTES SUMMARY SOLVERS BROWSE Two Cavities	T · 🗆 🗕 🗖 🗙
Tables Plies Laminates Creep Plate RC Paths	Cavities	
🗅 🔺 🛠 😵 🗹 🗮		
1: Cavity Data 1 2: Cavity Data 2	ТҮРЕ	
	Ideal Gas	
	🔿 Constant Bulk Modulus	
	OPTIONS	
	Treat multiple cavities as one	
	Stiffness Matrix in Nonlinear Analysis:	
	None (suitable for bulbous cavities)	
	Approximate (suitable for narrow cavities)	
	 Complete (suitable for cavities with few facets) 	
	MATERIAL V	
	Colour	
	Initial Pressure 0.0 MPa	
	Initial Temperature 0.0 C	
	PRE-LOAD	
	Pressure Nonlinear Control Case	
	1: Cavity Pressure 1 v	
0 Nodes 0 Beams 0 Plates 0 Bricks 0 Links 0	Vertices 0 Faces 0 Paths mm N t MPa C J (0;0;0)	

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).



Strand7 R3.1 Feature Summary

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.



Strand7[®]

Strand7 R3.1 Feature Summary

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







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



Strand7 R3.1 Feature Summary



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.

Strand7 R3.1 Feature Summary



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 Attributes	×	• •	Load Path Attributes	×
Apply load to			Apply load t	0	
Any element		~	Elements in	entity set	\sim
		~	Any elemen Beams only Plates only	t	
Keep selection	A	pply	Bricks only 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.



Strand7 R3.1 Feature Summary



📲 Strand7 🗠 🛅 🔚 📑 🛗 🖓 🐧 ?	ISUAL TE	EXT CASES	LAYOUTS NO	SUMMARY SOLVE	RS BROW	SE Curved Footbridge - Fi	nal 🏹 🗸 🗖 🗖	×
Solve statch Imb Batch Jobs	Home	Load Path	s Time Histo	ry Matrix Parameters	Files Re	sults Filtering		
LINEAR STATIC	Yes				/			\sim
LINEAR BUCKLING	Path 🔺	Include	Auto Divisions	Time Table		Path Template	Load Case Name	
LOAD INFLUENCE	1	 	~	1: Load Time History (Wa	king)	Load Path Template (Walking)	3: Moving Load (Walking)	
	2		~	2: Load Time History (Run	ining)	Load Path Template (Running)	4: Moving Load (Running)	
NATURAL FREQUENCY								
HARMONIC RESPONSE								
SPECTRAL RESPONSE								
LINEAR TRANSIENT DYNAMIC								
NONLINEAR TRANSIENT DYNAMIC								
STEADY HEAT								
TRANSIENT HEAT								
2,086 Nodes 58 Beams 2,288 Plates 0 Bricks	42 Links	0 Vertices	0 Faces 2 Pa	hs mm N t MPa	C J (-	70;7;-128) Model Selected	paths: 1	.::

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.

₽ .		I	Mesh5 - 1 Pla	te Proper	ty			×
/ 🔳 🗊 🗅 🖪 🗶 🗳	🗙 🗐 🐹 🗹		1. T	I I	•			
1: Plate Property 1	Unkno	wn Material						
	2D Plane S	itrain			 Isotro 	pic		~
	Material	Nonlinear	Damping	Tables	Thickness	Element		
	Thickness			0.0			mm	

Strand7 R3.1 Feature Summary



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.

• • ?				Me	sh - P35 - 1	Plate Pro	operty		×
/		k 🔜 🛛	1		a 🖻 🗄	►	Þ		
1: Plate Prop	perty 1	Unkn	own N	laterial					
		Plate/She	91				Isotropic		\sim
		Material	Non	linear	Damping	Tables	Thickness		
		Damping	Ratio ((NFA)		0.	0		
		Viscous D	ampin	g		0.	0	N·s/m/m ³	
		Rayleigh	Dampii	ng		F	requencies 🗸	1 <u>v</u>	
		Frequenc	y 1			10).0	Hz	
		Frequenc	y 2			30	0.0	Hz	
		Damping	Ratio	1		0.	02		
		Damping	Ratio 2	2		0.	05		
?				R	ayleigh Dam	ping			×
Mode									
Freque	encies						Rayleigh Damping	1	
🔿 Alpha,	Beta			0.0100	-			/	7
Values			atio	0.0090					
Freg. 1	10.0	Hz	jg R	0.0080					_
Freg. 2	200.0	Hz	mpir	0.0070	-				
Ratio 1	0.01		e Da	0.0070	= \				
Ratio 2	0.01		ectiv	0.0060					_
Display Pa	inge (Hz)		Eff	0.0050	F				
Freq. 1	10.0					_			
Freq. 2	200.0			0.0040 0	.0		100.0	<u> </u>	200.0
ineque .				Ŭ	-		Frequency (Hz)		
								OK	ancel
								<u>U</u> K (ancer

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.

MEMBRANE MA	ATRIX [C]		Units: N/mm
14014.9	5706.27	0.0	
	14014.9	0.0	
		4159.42	
ENDING/CURV	ATURE MATRIX [D]	0.0	Units: N·mm
4.14454 x 10 ⁶	1.68547 x 10 ⁶	0.0	
	4.14454 x 10 ⁶	0.0	
		1.26559 x 10 ⁶	
MEMBRANE/BE	NDING COUPLING MATRI	X [B]	Units: N
-11221.0	-2864.11	0.0	
-2864.11	-11221.0	0.0	

Strand7[®]

Strand7 R3.1 Feature Summary

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.

📫 🔸 ?	P42-GC Test20 - 2 Beam Properties									
/ 🔳 👩 🗅 🐚 🗡	(🗞 🛼 🗹 🚊 📽 🛸 *	I 🗈 🖂	E III							
1: Contacts	Unknown Material									
2: Bolt	Point Contact	Point Contact								
	Properties	Properties								
	Туре	Normal Gap V Dynamic Stiffnes								
	Friction Model		Plastic	~	Update Direction					
	Yield Surface		Elliptical	~						
	Takeup Action		Tension	\sim						
	Friction Coefficients	C1	0.45							
		C2	0.45							
	Maximum Tension		0.0		kN					
	Initial Axial Stiffness		64808.7		kN/mm					
	Sticking Friction Stiffness		64808.7		kN/mm					
	Strain Tolerance		0.00001		[1E-10 < Tol <1]					
	Conductivity		0.0		J/s/mm/C					
	Specific Heat		0.0		J/t/C					
	Effective Thermal Area		1.0		mm ²					
	Effective Thermal Mass		1.0		t					



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.

📫 🖶 ?	Mesh5 - 1 Beam Property						
Image: A marked and a marked	🗴 🗟 🗷 🗮 🗟 🕏	1 1 E		E I.			
1: Beam Property 1	Unknown Material						
	Connection	nnection					
	Material Nonlinear	Tables					
	Translation Stiffness		1	0.0	N/mm		
		2 3	0.0	N/mm			
				3	0.0	N/mm	
	Rotation Stiffness		1	0.0	N·mm/rad		
			2	0.0	N·mm/rad		
	_		3	0.0	N·mm/rad		
ſ	Conductivity			0.0	J/s/mm/C		
	Specific Heat			0.0	J/t/C		
	Effective Thermal Area			0.0	mm ²		
L	Effective Thermal Mass			0.0	t		

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.





Strand7 R3.1 Feature Summary

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.

Strand7 R3.1 Feature Summary





Beam Pre-Curvature

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

* •	↓] ?	Bean	n Attribu	tes	×			
Pre-cu	Pre-curvature: /mm							
1	0.000000				f(x)			
2	0.000000				<i>f</i> (x)			
	2							
Keep selection								
<u>S</u> cal	e A <u>d</u> e	ł	D <u>e</u> lete		Apply			

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.

Strand7 R3.1 Feature Summary





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.

"■~ 🕂 ?	Plate Attributes	×	□ 2	Plate Attributes	×			
-Face Support:	MPa/mm		Normal Face	Dressure MDa	<u> </u>			
K normal	0.0	f(x)			<i>(</i> ()			
K lateral	0.0	f(x)	✓ -2 Face	0.0	<i>J(X)</i>	[™] ■~ ↓ ?	Plate Attributes	×
Compressio	on-only		✓ +z Face	0.0	Л×Л	-Face Convection	Ambient: C	
Gap (mm)	0.0	<i>f(x)</i>	Store for	mulas		Value 0.0		f(x)
Limited bea	ring capacity		Formula uni	ts <none> <!--</td--><td>e> 🗸</td><td>✓ Temperature</td><td>vs Time Table</td><td></td></none>	e> 🗸	✓ Temperature	vs Time Table	
Max (MPa)	0.0	<i>f(x)</i>		+		<none></none>		~
Surface						Surface		
● -z	() +z					● -z	() +z	
Keep selection	on		Keep selec	tion		Keep selection		
<u>S</u> cale	A <u>d</u> d D <u>e</u> lete <u>A</u> p	ply	<u>S</u> cale	A <u>d</u> d D <u>e</u> lete	Apply	<u>S</u> cale A	dd D <u>e</u> lete	<u>A</u> pply

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.

'@≚ •	Brick Attributes	×	² ∠u -l⇒ 2			
Face Support:	MPa/mm			Dt	am Attribut	es 🔨
K normal	0.000000	f(x)	Support: N/m	im/mm		
K lateral	0.000000	f(x)	Ks Ks	0.0		f(x)
Compressi	on-only		Compressi	ion-only		<i>(</i>)
🖂 Gap (mm)	0.000000	<i>f(x)</i>	igap (mm)	0.0		J(×J
Limited be	aring capacity		Direction:			
🖂 Max (MPa)	0.000000	<i>f(x)</i>	• -1	○ +1	O -2	○ +2
Keep selecti	on 🗹 Free faces only		Keep select	ion		
<u>S</u> cale	A <u>d</u> d D <u>e</u> lete	Apply	<u>S</u> cale	A <u>d</u> d	D <u>e</u> lete	Apply

Strand7 R3.1 Feature Summary



Plate Pre-Curvature

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

• • •	↓ ? Pla	te Attributes	×				
Pre-cu	ırvature: /mm						
∠ x	0.000000		<i>f(x)</i>				
🗹 у	0.000000		<i>f(x)</i>				
	y	× -					
Kee	Keep selection						
<u>S</u> cal	le A <u>d</u> d	D <u>e</u> lete	Apply				

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 Attributes	×				
Plate Stiff	ness/Mass Factor					
Cxx	1.000000	f(x)				
🗹 Суу	1.000000	f(x)				
🗹 Cgg	1.000000	f(x)				
Czz	1.000000	f(x)				
Dxx	1.000000	<u>f(x)</u>				
🗹 Dyy	1.000000	<u>f(x)</u>				
🗹 Dgg	1.000000	f(x)				
Gxz	1.000000	<u>f(x)</u>				
🗹 Gyz	1.000000	<u>f(x)</u>				
🗹 Mass	1.000000	f(x)				
Keep selection						
<u>S</u> cale	A <u>d</u> d D <u>e</u> lete	<u>A</u> pply				

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.

•	¥ ?	Plate Attributes	×			
Edge	Global Pre	essure: MPa				
🗹 🗙	0.0		f(x)			
🗸 ү	0.0		f(x)			
🗹 Z	0.0		f(x)			
<mark>∕ St</mark> o	Store formulas					
Formu	ula units	<none> ~ <none></none></none>	~			
Global	Global XYZ [Cartesian] 🗸					
Keep selection Free edges only						
<u>S</u> ca	Scale Add Delete Apply					

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.

Strand7 R3.1 Feature Summary





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.

' ⊒~ ↓ ?	Edg	e Attributes	×			
Edge Cluster						
Auto ID						
🔘 Manual ID		1	0			
Cluster type		Rigid MPL	~			
Beam property type		Rigid MPL Pinned MP Master-Sla Interpolate	ve MPL ed MPL			
Auto origin	х	Beam Clust	ter			
O Manual origin	Y	0.0				
🚽 Origin	z	0.0				
Keep selection 🖌 Free edges only						
<u>S</u> cale A <u>d</u> d		D <u>e</u> lete	Apply			



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.



Strand7 R3.1 Feature Summary





STAGING

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

🛛 🖂 🔽 🛛 🖥	±,≣ ±	🖌 🗶 🐘 🐘 🖓 🕹	🔸 🗠 🚥	1				
Model ^	Stage (00 - Piling						
Tunnel A	ID	Name	Type	Morph Fixed Nodes	Rotate Clusters	Fluid	Stage Fluid Level (m)	Num Groups
Existing structure	1	Stage 00 - Piling	Unmorphed			Global Level	0.0	11
- H Crosshead	2	Stage 01 - Road	Morphed	\checkmark	~	Global Level	0.0	12
	3	Stage 02 - Encasement	Morphed		~	Stage Level	-3.0	13
Bottom	4	Stage 03 - Dual gauge & lid	Unmorphed			Stage Level	-10.0	16
- File caps	5	Stage 04 - Upper roof	Unmorphed			Stage Level	-10.0	16
Underpinning support	6	Stage 05.1 - Pile caps	Morphed	\checkmark		Stage Level	-10.0	17
- Primary beams	7	Stage 05.2 - Form & pour beam	Morphed			Stage Level	-10.0	18
- 🗐 Diaphragm A	8	Stage 05.3 - Pour diaphragms	Morphed			Stage Level	-10.0	18
- 🖽 Diaphragm B	9	Stage 05.4 - Jacking	Unmorphed	\checkmark		Stage Level	-10.0	18
Grout	10	Stage 05.5 - Grout & cut	Morphed			Stage Level	-10.0	19
Compression members	11	Stage 06 - Lid	Unmorphed			Stage Level	-10.0	20
Bearing plates	12	Stage 07 - Tunnel slab	Morphed			Stage Level	3.0	20
	13	Stage 08 - Roof	Unmorphed			Global Level	0.0	22
- Rock springs	14 Prope	Stage 08 - Reset rty display	Reset			Global Level	0.0	22
Pile caps	1: Sta	age 00 - Piling						

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



Strand7 R3.1 Feature Summary

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.

• <u>•</u> • •	Plate Attributes	×			
Plate Propert	у Туре				
Number	2	0			
Available Pro	perties				
2: Plate Prop	erty 2	\sim			
O Expression	n 0				
Switch at Sta	ge				
2: B		\sim			
Inherit Bir	th Stage				
Keep selection Update display on Apply					
<u>S</u> cale	A <u>d</u> d D <u>e</u> lete <u>App</u>	oly			

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.



Strand7 R3.1 Feature Summary



	Properti	es		Properties
Section Area	4161.28	mm ²	Section Area	4163.44 mm
111	4.24556 x 10 ⁷	mm ⁴	111	4.26844 x 10 ⁷ mm ⁴
122	2.66388 x 10 ⁶	mm ⁴	122	2.68848 x 10 ⁶ mm ⁴
J	212278.	mm ⁴	J	212151. mm*
Shear L1	-54.8258	mm	Shear L1	-50.7999 mm
Shear L2	0.0	mm	Shear L2	0.0 mm
Shear A1	0.0	mm ²	Shear A1	0.0 mm
Shear A2	0.0	mm ²	Shear A2	0.0 mm
Integration Points	5	0	Integration Points	5 0
Note 1: Set Shear A1 and Shea Note 2: MNL beams and tapere	r A2 to zero for the Thin Beam forn ed beams only use Integration Poir	nulation. Its.	Note 1: Set Shear A1 and Shea Note 2: MNL beams and taper	r A2 to zero for the Thin Beam formulation ed beams only use Integration Points.

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.

Assign default concrete (AS 3600)	fc = 20 MPa	Assign default concrete (AS 3600)	
Assign default steel (AS 3600)	fc = 25 MPa	Assign default steel (AS 3600)	fy = 400 MPa
Clear concrete	fc = 32 MPa	Clear concrete	fy = 500 MPa
Clear steel	fc = 40 MPa	Clear steel	fy = 600 MPa
	fc = 50 MPa		1
	fc = 65 MPa		
	fc = 80 MPa		
	fc = 100 MPa		

Strand7 R3.1 Feature Summary



Assign default concrete (EC 2)	fc = 12 MPa	Assign default concrete (EC 2)	
Assign default steel (EC 2)	fc = 16 MPa	Assign default steel (EC 2)	fy = 200 MPa
Clear concrete	fc = 20 MPa	Clear concrete	fy = 250 MPa
Clear steel	fc = 25 MPa	Clear steel	fy = 500 MPa
	fc = 30 MPa		fy = 650 MPa
	fc = 35 MPa	L	
	fc = 40 MPa		
	fc = 45 MPa		
	fc = 50 MPa		
	fc = 55 MPa		
	fc = 60 MPa		
	fc = 70 MPa		
	fc = 80 MPa		
	fc = 90 MPa		
Assign default concrete (ACI 318)	fc = 2.5 ksi	Assign default concrete (ACI 318)	•
Assign default steel (ACI 318)	fc = 3 ksi	Assign default steel (ACI 318)	► fy = 40 ksi
			6. C0 hri
Clear concrete	fc = 3.5 ksi	Clear concrete	TY = 60 KSI
Clear concrete Clear steel	fc = 3.5 ksi fc = 4 ksi	Clear concrete Clear steel	fy = 80 ksi
Clear concrete Clear steel	fc = 3.5 ksi fc = 4 ksi fc = 4.5 ksi	Clear concrete Clear steel	fy = 80 ksi fy = 80 ksi fy = 100 ksi
Clear concrete Clear steel	fc = 3.5 ksi fc = 4 ksi fc = 4.5 ksi fc = 5 ksi	Clear concrete Clear steel	fy = 80 ksi fy = 80 ksi fy = 100 ksi
Clear concrete Clear steel	fc = 3.5 ksi fc = 4 ksi fc = 4.5 ksi fc = 5 ksi fc = 6 ksi	Clear concrete Clear steel	fy = 60 ksi fy = 80 ksi fy = 100 ksi
Clear concrete Clear steel	fc = 3.5 ksi fc = 4 ksi fc = 4.5 ksi fc = 5 ksi fc = 6 ksi fc = 7 ksi	Clear concrete Clear steel	fy = 60 ksi fy = 80 ksi fy = 100 ksi
Clear concrete Clear steel	fc = 3.5 ksi fc = 4 ksi fc = 4.5 ksi fc = 5 ksi fc = 6 ksi fc = 7 ksi fc = 8 ksi	Clear concrete Clear steel	fy = 80 ksi fy = 80 ksi fy = 100 ksi
Clear concrete Clear steel	fc = 3.5 ksi fc = 4 ksi fc = 4.5 ksi fc = 5 ksi fc = 6 ksi fc = 7 ksi fc = 8 ksi fc = 9 ksi	Clear concrete Clear steel	fy = 60 ksi fy = 80 ksi fy = 100 ksi
Clear concrete Clear steel	fc = 3.5 ksi fc = 4 ksi fc = 5 ksi fc = 6 ksi fc = 7 ksi fc = 8 ksi fc = 9 ksi fc = 10 ksi	Clear concrete Clear steel	fy = 60 ksi fy = 80 ksi fy = 100 ksi
Clear concrete Clear steel	fc = 3.5 ksi fc = 4 ksi fc = 4.5 ksi fc = 5 ksi fc = 6 ksi fc = 7 ksi fc = 8 ksi fc = 9 ksi fc = 10 ksi fc = 11 ksi	Clear concrete Clear steel	fy = 60 ksi fy = 80 ksi fy = 100 ksi
Clear concrete Clear steel	fc = 3.5 ksi fc = 4 ksi fc = 4.5 ksi fc = 5 ksi fc = 6 ksi fc = 7 ksi fc = 8 ksi fc = 9 ksi fc = 10 ksi fc = 11 ksi fc = 12 ksi	Clear steel	fy = 80 ksi fy = 80 ksi fy = 100 ksi
Clear steel	fc = 3.5 ksi fc = 4 ksi fc = 4 ksi fc = 5 ksi fc = 6 ksi fc = 7 ksi fc = 8 ksi fc = 9 ksi fc = 10 ksi fc = 11 ksi fc = 12 ksi fc = 13 ksi	Clear steel	fy = 80 ksi fy = 80 ksi fy = 100 ksi

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.

General Attribute Property Model	1	-									
V Type Material Name	8	8≡~ 🎧 🔺									
Elastic Modulus	umns										
Poisson's Ratio		1.93 x 10 ⁵									
Density			Туре	Material	E	G	v	ρ	α	t _{mem}	tbend
Ihermal Expansion Membrane Thickness					MPa	MPa		t/mm ³	/C	mm	mm
Bending Thickness		1: 5mm FRP	Plate/Shell	Undefined material	8.20 x 10 ³		3.10 x 10 ⁻¹	4.27 x 10 ⁻⁹	0.00×10^{0}	5.00 x 10 ⁰	5.00 × 10 ⁰
o bending middless		2: 3mm FRP	Plate/Shell	Undefined material	8.00 x 10 ³		3.10 x 10 ⁻¹	4.27 x 10 ⁻⁹	0.00 x 10 ⁰	3.00 x 10 ⁰	3.00 x 10 ⁰
		3: 1.6mm Stainless Steel	Plate/Shell	Stainless Steel: Grade 304 (UNS No. S30400)	1.93 x 10 ⁵		2.50 x 10 ⁻¹	8.00 x 10 ⁻⁹	1.72 x 10 ⁻⁵	1.60 x 10 ⁰	1.60 x 10 ⁰
Structural	Q	5: 7mm FRP	Plate/Shell	Undefined material	8.20 x 10 ³		3.10 x 10 ⁻¹	4.27 x 10 ⁻⁹	0.00 x 10 ⁰	7.00 x 10 ⁰	7.00 x 10 ⁰
Laminate	ner	6: 3.2mm Stainless Steel	Plate/Shell	Stainless Steel: Grade 304 (UNS No. S30400)	1.93 x 10 ⁵		2.50 x 10 ⁻¹	8.00 x 10 ⁻⁹	1.72 x 10 ⁻⁵	3.20 x 10 ⁰	3.20 x 10 ⁰
Laminate Layup	itie	7: Bulkhead Connection (10mm FRP)	Plate/Shell	Undefined material	8.00 x 10 ³		3.10 x 10 ⁻¹	4.27 x 10 ⁻⁹	0.00 x 10 ⁰	1.00 x 10 ¹	1.00×10^{1}
Laminate Matrix	s	9: Thicker FRP (3mm+2mm)	Plate/Shell	Undefined material	8.00 x 10 ³		3.10 x 10 ⁻¹	4.27 x 10 ⁻⁹	0.00 x 10 ⁰	5.00 x 10 ⁰	5.00 x 10 ⁰
Nonlinear											
Heat											
Tables											
Tables											
		I I I I Structural Lamina	te 🗸 Laminate L	ayup 🖉 Laminate Matrix 🖉 Nonlinear 🖉 Damping	g 🖉 Heat 🖉 Tables			<			>

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,

- 0 `	?	Move by Increment	×
Glo	bal XYZ [Cartes	ian]	~
Inc	rements		
х	Y/2		f(x)
Y	0.0		f(x)
z	0.0		f(x)
×1	Y 🛃 Set by p	points	
	Keep selection		Apply

Strand7 R3.1 Feature Summary



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.

_ □ ~ ?	Move by Rotation	×
Global XYZ [C	artesian]	~
Increment Axis		
7005	Set origin by point	
	Set origin by point	
Origin X	0.0	
Origin Y	0.0	
Origin Z	0.0	
	🕒 Set angle by points	
Angle (deg)	0.0	
Keep sele	ction	Apply

Cut Elements

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



Strand7 R3.1 Feature Summary



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.

Strand7 R3.1 Feature Summary





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.

Strand7 R3.1 Feature Summary





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.

⊞~ ?	Subdivide Beams by Length	×
Subdivid	de beams	
Length	165.0	mm
	Apt	oly

Strand7 R3.1 Feature Summary

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.

Strand7 R3.1 Feature Summary



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.



Strand7 R3.1 Feature Summary



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.

*	×		
Sizes			
Maxim	ium edge length	0.5	mm
		3.57143	0 %
Transi	tioning:	0.4.1	
	ne 🔾 Auto 🕻	Custom	
⊡ Co	nsider surface cun	/ature	
Co	nsider nearby vert	ices	
Lengtl	h ratio (min/max)	0.1	
Maxim	ium increase	40	0 %
Min e	dges per circle	12	\odot
on e	dges longer than	0.0	mm
Res	et		
☐ Pre	view (approximate le faces after mesh	e) hing	<u>M</u> esh
Strand7 R3.1 Feature Summary



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**.

	× ?	Co	by by Increment	×
Glo	bal XYZ [C	artesian]		~
In	crements			
х	0.0			f(x)
Y	0.0			<i>f(x)</i>
z	5.0			f(x)
×	1/2 🛃 ! 3	Set by poin	nts	
	2	ant 1		^
	1	eat 1		0
	1/2	/ertex/	Attributes	
	1/3	t/Face	Attributes	
~	Increment	t String Gr	roup ID	
\checkmark	Increment	t Cluster II	D	
	Create Ne	w Group	for Copy	
	Keep selee	tion		Apply

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).

Strand7 R3.1 Feature Summary



🖽 🗠 Subdivide 🗙
Divisions
All Elements
A 1 0
Plate, Brick
в 1 🗘
Brick
c 1 🗘
Targets
Plate
Source 🗸
Tri3
Tri6
Quad4
Quad8
Quad9
Source
Tri
Ouad

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.

≧ × ?	(Clean Geometry									
Settings	Statistics	Mes	Messages								
SELECTION											
Act on			Duplicate faces								
Who	le model		Leave all faces								
◯ Selec	ted only		 Leave one face Delete all faces 								
Act on free edges only											
Clean groups independently											
TOLERANCES											
Edge Mer	ging Angle (deg)	160.0								
Minimum	Feature Leng	gth	0.0005								
Relativ	Relative Absolute (m)										
				<u>R</u> eset	<u>C</u> lea	n					

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.

DESCRIPTION	
CASE DEPENDENCE	
Property Temperature Dependence	
<none></none>	~
Cable Inertia Case Form stiffness matrix for every load case	
1: Load Case 1	~
Active Stage	
<all active="" groups=""></all>	~

Strand7 R3.1 Feature Summary



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 (KO) from stress results option is available to assign the average lateral stress by the calculation of KO.
- The **Solver control** group offers direct control of the solver iteration parameters for the in-situ calculation.

?	In-situ Stress Generation								
Fluid inside so	il elements								
Default Level	11.0	m							
Mass Density	1000.0	kg/m ³ Vater							
Cases									
Load case defi	ning gravity								
4: Gravity + H	ydrostatic (In-situ)	~							
Freedom case									
1: Freedom Ca	ase 1	~							
Stage									
<all a<="" groups="" td=""><td>ctive></td><td></td><td></td></all>	ctive>								
gi pi									
Options 📕									
Use existing	; in-situ attributes as init	ial conditions							
Calculate h	orizontel stress ratio (KO)	from stress results							
Solver control	¥								
Iteration Limit	20 C Allow sc	lver to add iterations							
Matrix Schem	ie	Node Ordering							
O Skyline	Direct sparse	None O Tree							
O Iterative (PCG)	Geometry AMD							
	Import Solve (Ba	itch) <u>S</u> olve (Wait) <u>C</u> lose							

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.

Strand7 R3.1 Feature Summary



Home	Matrix	Pa	arameters	Files	Results	Filtering	
GENERA	L TS		EIGENV	ALUE F	EXTRACT	ON (BUCKLING AND FREQUENCY) ✓	
EIGENV	ALUE		0.00001			Zero frequency (Hz)	
DYNAMICS 1.0 x 10 ⁻¹⁰			-10		Zero buckling Eigenvalue		
6 🗘		0	Additional working vectors 🗹 set automatically Iteration tolerance				
			50		0	Iteration light	
			Ensure Eigenvector co			nvergence	
			Auto	shift w	hen rigid l	pody motion detected	

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:

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

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
         Frequency
 Mode
                         P-DX
                                        P-DY
                                                       P-D7
                                                                     P-RX
                                                                                    P-RY
                                                                                                  P-R7
                                     SORT(N/mm)
                                                                  SORT(N.mm)
                                                                                 SORT(N.mm)
            (H7)
                       SORT(N/mm)
                                                    SORT(N/mm)
                                                                                               SORT(N.mm)
    1
      1.466385E+00
                     1.349967E+01 -3.509512E-02
                                                  2.096231E+01
                                                                 3.118482E+05
                                                                                1.338942E+04
                                                                                             -2.019440E+05
       2.061489E+00
                     2.518423E+01 4.155024E-01
                                                  -1.565447E+01
                                                                -2.463401E+05
                                                                               3.193650E+05 -3.913983E+05
    2
                                   -1.651454E-01
                                                  1.451089E+01
                                                                 2.480814E+05
                                                                               -2.798480E+05
       2.789282E+00
                     7.869350E+00
                                                                                             -1.411296E+05
    3
                                                                 1.333854E+04
       4.510150E+00
                     5.635333E+00
                                   5.257864E-02
                                                  9.112563E+00
                                                                               -2.681165E+04 -8.362777E+03
       7.475686E+00
                     2.500738E+00 -2.103694E-01
                                                  2.542251E+00
                                                                 1.082266E+04
                                                                               3.117561E+04 -9.645800E+03
       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
            (H_{7})
                          (+)
                                        (N/mm)
                                                        (%)
                                                                      (%)
                                                                                     (%)
                                                                                                  (%)
                                                                                                           (%)
                                                                                                                   (%)
                     2.651808E+02
                                    2.251117E+04
                                                  1.509234E+01
                                                                 1.020704E-04
                                                                               3.639055E+01
                                                                                                15.092
                                                                                                          0.000
                                                                                                                 36.391
    1
      1.466385E+00
                     5.826701E+02
       2.061489E+00
                                    9.775622E+04
                                                  5.252521E+01
                                                                 1.430715E-02
                                                                               2.029491E+01
                                                                                                52.525
                                                                                                          0.014
                                                                                                                 20.295
    2
                     1.968148E+02
                                                                               1.743807E+01
                                                                                                                 17.438
       2.789282E+00
                                    6.045086E+04
                                                  5.128474E+00
                                                                 2.260161E-03
                                                                                                 5.128
                                                                                                          0.002
    3
                                                                 2.290999E-04
                     2.490158E+02
                                                  2.629963E+00
    4
      4.510150E+00
                                    1.999717E+05
                                                                               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
         Frequency
                      Modal Mass
                                     Modal Stiff
                                                       P-RX
                                                                     P-RV
                                                                                    P-R7
                                                                                                 P-RX
                                                                                                          P-RV
                                                                                                                  P-RZ
 Mode
            (Hz)
                        (t.mm^2)
                                        (N.mm)
                                                        (%)
                                                                      (%)
                                                                                     (%)
                                                                                                  (%)
                                                                                                           (%)
                                                                                                                   (%)
                                                                               1.047697E+01
       1,466385E+00
                                                   3.377146E+01
                                                                 7.311858E-02
                                                                                                33.771
                                                                                                         0.073
                                                                                                                 10.477
    1
       2.061489E+00
                                                   2.107335E+01
                                                                 4.159862E+01
                                                                                3.935606E+01
                                                                                                21.073
                                                                                                         41.599
                                                                                                                 39.356
    2
    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
                                                   4.067535E-02
       7.475686E+00
                                                                 3.964004E-01
                                                                               2.390288E-02
                                                                                                 0.041
                                                                                                          0.396
                                                                                                                  0.024
      7.844652E+00
                                                   3.635775E-04
                                                                 1.136909E+01
                                                                               5.804985E-01
                                                                                                 0.000
                                                                                                        11.369
                                                                                                                  0.580
    6
 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.

Strand7 R3.1 Feature Summary



SPECTRUM TYPE			
Response			
⊖ PSD			
EXCITATION			
✓ Load Excitation			
Base Excitation			
COMBINED RESULTS SIGN			
Autosign			
○ Absolute			
	di i	Confirmation	×
	7	File: E:\Modal Participation Factors\A.SRA File exists.	
		Over <u>w</u> rite A <u>p</u> pend <u>C</u> ancel	

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.

Strand7 R3.1 Feature Summary



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]	~

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></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.

Strand7 R3.1 Feature Summary



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.

Strand7 R3.1 Feature Summary



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.





Strand7 R3.1 Feature Summary

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.

📑 Strand7 🗠 🛅 📑 📑	.000 🗟 ? VISUA	L TEXT C	ASES LAYOUTS NO	TES SUN	MMARY SOLVERS BROW	'SE Fillin	g Tank With Position Offset	⊡ ∼ — □ ×
► Solve 🐤 Batch 🖬 Batch Jot	Home Load Pat	hs Time	History Matrix Para	meters	Files Results Filtering			
LINEAR STATIC	1: Movement of Hyd	rostatic Load	1					~
LINEAR BUCKLING		Include	Factor vs Time Table	Add	Factor vs Position Table	Factor vs Position UCS	Factor vs Position Axis	Position Offset vs Time Table
LOAD INFLUENCE	1: Load Case 1	\checkmark	<none></none>		1: Hydrostatic Relationship	Global XYZ [Cartesian]	3	1: Movement of Hydrostatic Load
	1: Freedom Case 1	\checkmark	<none></none>		<none></none>	Global XYZ [Cartesian]	1	<none></none>
HARMONIC RESPONSE SPECTRAL RESPONSE LINEAR TRANSIENT DYNAMIC NONLINEAR TRANSIENT DYNAMIC STEADY HEAT TRANSIENT HEAT								
384 Nodes 0 Beams 360 Plates	0 Bricks 0 Links	0 Vertices	0 Faces 0 Paths n	n N k	g Pa K J (-56;0;26)	Model Cases: 2		.4

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.



Strand7 R3.1 Feature Summary





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 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.



Strand7 R3.1 Feature Summary



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

Strand7 R3.1 Feature Summary



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.



Strand7 R3.1 Feature Summary

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.

📫 Strand7 🕆 🗋 💼 🔒 📫	🗎 🚧 🐧 ? 🛛 VISU	AL TEXT C	SES LAYOU	JTS NOTES SUMM	IARY SOLVERS BR	OWSE Differe	ential Creep and Shrinkage Analysis_Q8	Ĩ~□ _ □ ×
► Solve 🐤 Batch 🗰 Batch Jo	Home Load Ma	atrix Paramet	ers Files	Results Filtering				
LINEAR STATIC	📄 👬 👬 👬	X 🗉 🛛 🗙	⊿ ⊠*	18				
LINEAR BUCKLING	Yes							~
NONLINEAR STATIC		Include	1	2	3	4		
QUASI STATIC	LABEL	1	increment	Increment	Increment	Increment		
NATURAL FREQUENCY	1: Load Case 1	\checkmark	1.0	1.0	1.0	1.0		
HARMONIC RESPONSE	1: Freedom Case 1	~						
SPECTRAL RESPONSE	PSEUDO TIME (s)	✓	2.592 x 10 ⁶	7.776 x 10 ⁶	3.1104 x 10 ⁷	3.1104 x 10 ⁸		
LINEAR TRANSIENT DYNAMIC	RESET MODEL		\checkmark	\checkmark	\checkmark	\checkmark		
NONLINEAR TRANSIENT DYNAMIC								
STEADY HEAT								
TRANSIENT HEAT								
1,184 Nodes 0 Beams 384 Pla	ates 0 Bricks 0 Lin	ks 0 Vertices	0 Faces	0 Paths mm N	t MPa C J (1	5;39;1) Model Incr	rements: 4	ii.

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**.

Strand7 R3.1 Feature Summary



GENERAL	AUTOMATIC SUB-STEPPING V								
SUB-STEPPING TERATION	○ None ○ Load scaling								
	 Displacement scaling v 	vith load reduction							
	Oisplacement scaling with displacement reduction								
	O Displacement control (arc length)								
	0.1	Minimum reduction factor							
	30.0	Maximum rotation increment (deg)							
	0.1	Maximum displacement increment (Ratio)							
	0.01	Maximum MNL strain increment							
	0.001	Minimum arc length reduction factor							
	2000.0	✓ Terminate if displacement exceeds (mm)							
	30.0	✓ Terminate if rotation exceeds (deg)							
	Save sub-steps								

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.

BASE EXCITATION						
O None		Table Factor	Displacement vs Time Table	Initial Disp (mm)	Initial Vel (mm/s)	Initial Acc (mm/s ²)
Acceleration	х	0.0	<none> ~</none>	0.0	0.0	0.0
	Y	0.0	<none> ~</none>	0.0	0.0	0.0
Velocity	z	0.0	<none> ~</none>	0.0	0.0	0.0
Usplacement						

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.

?	Results Options	×
Displacement	Envelopes	
Add initial	GNL displacements to display ement offset in displacement results	Transient Displacement Relative to base Total
Reaction MP O Use defir O Move ori	L in GNL - Moment Summation ned origin gin by average node displacement	Transient Velocity O Relative to base Total
NFA Modal E Unit Mod Engineer	Displacement Ial Mass ing Modal Mass	Transient Acceleration Relative to base Total
		<u>O</u> K <u>C</u> ancel

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.

LOAD TYPE
Base Acceleration
O Base Velocity
◯ Base Displacement
○ Applied Load
SOLUTION TYPE
⊖ vs Frequency
vs Time
© vs time

Strand7 R3.1 Feature Summary



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.

REFERENCE/INITIAL TE	MPERATURE
25.0	c
APPLY NODAL/GLOBAL	ACCELERATION TO
Structural mass	
Non-structural mass	
GEODAL INCIGIA LOAD	
O None	
Gravity	
O Acceleration and Veloc	ity
O Seismic	
	x 2
OX ®Y OZ	
ACCELERATION DUE TO) gravity 🗸
-9806.65	mm/s ²

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

 ✓ 1: Dead Load, G ✓ 2: Floor Live Loads, Q ✓ 3: Wind Loading, Wu 4: Seismic Case 1 	NON-STRUCTURAL MASS ADDED TO MASS MATRIX	
	 ✓ 1: Dead Load, G ✓ 2: Floor Live Loads, Q ✓ 3: Wind Loading, Wu ✓ 4: Seismic Case 1 	

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.

📑 Strand7 🗠 📄 🔚 📑 🛗 🤐 🔍 ? VISU	AL TEXT CASES LA	YOUTS NOTES SU	MMARY SOLVERS	BROWSE Bui	lding-Seismic	□ ×
Solve * Batch III Batch Jobs	Home Load Matrix	Parameters Files	Results Filtering	1		
LINEAR STATIC		Solver ge	nerates combinations f	ile		
LINEAR BUCKLING		1: Fixed Footings	2: Freedom Case 2			
	1: Dead Load, G	\checkmark	\checkmark			
	2: Floor Live Loads, Q	\checkmark	\checkmark			
	3: Wind Loading, Wu	\checkmark	\checkmark			
HARMONIC RESPONSE	4: Seismic Case 1	\checkmark	\checkmark			
SPECTRAL RESPONSE						
LINEAR TRANSIENT DYNAMIC						
NONLINEAR TRANSIENT DYNAMIC						
STEADY HEAT						
TRANSIENT HEAT						
2,496 Nodes 72 Beams 2,418 Plates 0 Bricks 0	Links 0 Vertices 0 Fa	ces O Paths mm	N t MPa C J	(-145;55;178) Mo	del Selected cases: 8	:

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.

Strand7 R3.1 Feature Summary



INITIAL CONDITIONS	
From File	~
Initial Conditions File	
C:\St7 Models\Dynamics of Tied Arch Roof.LTA	
5: 0.5 s	~
Initial Velocity From Load Case	
<none></none>	×
Time Steps:	
O Append all time steps	
Append remaining time	

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.

• ? SHB_2	26_transient_moving_lad_R3 - Available result files	×
🕒 Open 💼 Browse 🔗 Relink 📩 Dismiss 🗙 Delete 🖬	🛔 Files 🔅 Options 🔽 📃 🔽	
LINEAR BUCKLING (LBA)	NONLINEAR STATIC (NLA)	
Hide unconverged modes	Hide unconverged results	
Hide negative modes	Hide intermediate sub-steps	
NATURAL FREQUENCY (NFA)	QUASI-STATIC (QSA)	
Hide unconverged modes	Hide unconverged results	
Hide zero modes	Hide intermediate sub-steps	
Hide modes with mass participation below 0.0 🗘 %	NONLINEAR TRANSIENT (NTA)	
SPECTRAL RESPONSE (SRA)	Hide unconverged results	
Hide modal results	Hide intermediate sub-steps	
One file listed 0 files available		.i.

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 postprocessing.

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.



Strand7 R3.1 Feature Summary

P Buck	ding of Tied	d Arch Roof - 2: Rod pre-te	nsion 🗙	Buck	kling of Tied Arch Roof - 2: Rod pre-tension	×
• / 🔳 🛙	1: 20 m	nm Plates		• / 🔳 🖩	1: 20 mm Plates	
Plate 1	e Stored	e Spent		Plate 1	e Stored e Spent	
Quantity		e Stored (J/mm ²)	e Spent (J/mm ²)	Quantity	e Stored (J) e Spent (J)	
 Displacement 	Centroid	8.28949 x 10 ⁻⁶	8.28949 x 10 ⁻⁶	 Displacement 	Integral 0.055426 0.055426	
Stress	GP: 1	6.80972 x 10 ⁻⁶	6.80972 x 10 ⁻⁶	○ Stress		
🔘 Strain	GP: 2	9.77057 x 10 ⁻⁶	9.77057 x 10 ⁻⁶	🔿 Strain		
O Node Force	GP: 3	6.80587 x 10 ⁻⁶	6.80587 x 10 ⁻⁶	O Node Force		
Energy Density	GP: 4	9.77181 x 10 ⁻⁶	9.77181 x 10 ⁻⁶	C Energy Density		
	N[1]: 7	5.72807 x 10 ⁻⁰	5.72807 x 10 ⁻⁰	Energy Integral		
	N[2]: 3	1.08532 x 10 ⁻⁵	1.08532 x 10 ⁻⁵			
O Force	N[3]: 60	1.08586 x 10 ⁻⁵	1.08586 x 10 ⁻⁵	OForce		
Moment	N[4]: 12	5./181/ X 10 °	5./181/ X 10 *	 Moment 		
Curvature				Curvature		
OUser				🔿 User		
e Stored(C)				e Stored		
Min Max				Min Max		
Absolute				Absolute		
🔶 🗌 Highlights	Gauss P	oints 🗹 Nodes		+ Highlights		

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.



Strand7 R3.1 Feature Summary

		iea Arch Roof - 2: R	oa pre-tension		×
• / 🗖 🗄	1: 20 mm Plate	es			
Plate 1	Mid - z + z				
Quantity	xx yy xy xz	yz zz			
O Displacement	Surface: Mid	- 	107	207	
Stress	Centroid	-1.63813 x 10 ⁻⁵	6.40805 x 10 ⁻⁵	-2.36547 x 10 ⁻⁶	
Strain	GP: 1	-1.66041 x 10 ⁻⁵	5.82932 x 10 ⁻⁵	-2.38124 x 10 ⁻⁶	
O Node Force	GP: 2	-1.66039 x 10 ⁻⁵	6.98678 x 10 ⁻⁵	-2.47526 x 10 ⁻⁶	
Energy Density	GP: 3	-1.61589 x 10 ⁻⁵ 5.82932 x 10 ⁻⁵ -2	-2.24353 x 10 ⁻⁶		
	GP: 4	-1.61585 x 10 ⁻⁵	6.98678 x 10 ⁻⁵	-2.36185 x 10 ⁻⁶	
	N[1]: 7	-1.67671 x 10 ⁻⁵	5.40567 x 10 ⁻⁵	-2.40049 x 10 ⁻⁶	
⊖ Force	N[2]: 3	-1.67669 x 10 ⁻⁵	7.41043 x 10 ⁻⁵	-2.54793 x 10 ⁻⁶	
Moment	N[3]: 60	-1.59953 x 10 ⁻⁵	7.41043 x 10 ⁻⁵	-2.3669 x 10 ⁻⁶	
Curvature	N[4]: 12	-1.59962 x 10 ⁻⁵	5.40567 x 10 ⁻⁵	-2.14656 x 10 ⁻⁶	
🔿 User 🥒					
Charles Truns					
Defender					
Deformation					
O Total					

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.

CALCULATE V	
✓ Node Reaction	
🗹 Element Node Force 🦯	
🗹 Node Inertia Force 🦊	
☑ Node Velocity	
☑ Node Acceleration	
Beam Force/Moment/Stress	
Beam Strain/Curvature/Energy	
Beam Fibre Results (MNL and CNL)	
Plate Force/Moment/Stress	
Plate Strain/Curvature/Energy	
Brick Stress	
Brick Strain/Energy	
Link Node Force	

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).





?	Element Resul	ts Settings	3
/			
SYSTEM		^	Facet 15 0 dec
	Local		ucc
	Global		Absolute values
	UCS		
	Combined		
COMPOR	NENT		
	xx		
	YY		
	ZZ		
	ХҮ		
	YZ		
	ZX		
SURFACE			
	Mid Plane (or Visible Surfaces)		
	-z (or Visible Surfaces)		
	+z (or Visible Surfaces)		
EXTRAPO	DLATION		
	Centroidal Value		
	Nodal Values Extrapolated from Gauss Point	15	
	Gauss Point Values Placed at Nodes		<u>S</u> ettings
Marga	egends Apply to all views	OK	Cancel Apply

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.

?					Box	Girder - Cut	ting Planes
Planes Int	egrals Va	alues 👆 其 🕽	×: 🔒 🗐 😰 [□ 🗖 🙆			
Yes							
Plane ID	Show	Name	UCS	Plane	Axis 1	Axis 2	Axis 3
1	\checkmark	Cutting Plane	UCS 1 [Cylindrical]	ZR Plane	72.4	-74.0	-2.5
2	\checkmark	Cutting Plane	UCS 1 [Cylindrical]	ZR Plane	72.4	-80.0	-2.5
3	\checkmark	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	\checkmark	Cutting Plane	UCS 1 [Cylindrical]	ZR Plane	72.4	-98.0	-2.5
6	\sim	Cutting Plane	UCS 1 [Cylindrical]	ZR Plane	72.4	-104.0	-2.5
7	\checkmark	Cutting Plane	UCS 1 [Cylindrical]	ZR Plane	72.4	-110.0	-2.5
8	\checkmark	Cutting Plane	UCS 1 [Cylindrical]	ZR Plane	72.4	-116.0	-2.5
9	\sim	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.

Strand7 R3.1 Feature Summary





?		Box Girder - Cutting Planes										
Planes	Integrals	Values 🛃 Fo	r non-GNL re	sults, the un	deformed me	sh is used fo	r integrals.					
	Plane ID	Result Case	СХ	CY	CZ	F1	F2	F3	M1	M2	M3	
			m	m	m	N	N	N	N·m	N·m	N·m	
1	1	1: Load Case 1	8.386	1.61493	20.535	208.179	-3.27548 x 10 ⁶	-1.04796 x 10 ⁶	1.64242 x 10 ⁷	-343925.	4.90445 x 10 ⁶	
2	2	1: Load Case 1	6.63182	1.61493	12.9368	52.5823	-3.37481 x 10 ⁶	-2.66871 x 10 ⁶	4.24617 x 10 ⁷	-853489.	4.84127 x 10 ⁶	
3	3	1: Load Case 1	5.68148	1.61493	5.19686	83.2522	-3.16859 x 10 ⁶	-4.30766 x 10 ⁶	6.80773 x 10 ⁷	-1.35556 x 10 ⁶	4.72539 x 10 ⁶	
4	4	1: Load Case 1	5.54538	1.61493	-2.60001	-393834.	-2.65531 x 10 ⁶	-5.91842 x 10 ⁶	9.08847 x 10 ⁷	-3.14654 x 10 ⁶	897699.	
5	5	1: Load Case 1	6.22503	1.61493	-10.3684	-393740.	-1.84095 x 10 ⁶	-7.45471 x 10 ⁶	1.08509 x 10 ⁸	-6.28945 x 10 ⁶	403317.	
6	6	1: Load Case 1	7.71297	1.61493	-18.0232	-393773.	-736522.	-8.87141 x 10 ⁶	1.18639 x 10 ⁸	-9.36423 x 10 ⁶	-417166.	
7	7	1: Load Case 1	9.9929	1.61493	-25.4805	-393782.	640466.	-1.01249 x 10 ⁷	1.19079 x 10 ⁸	-1.23365 x 10 ⁷	-1.55456 x 10 ⁶	
8	8	1: Load Case 1	13.0398	1.61493	-32.6587	-393762.	2.26677 x 10 ⁶	-1.11743 x 10 ⁷	1.0779 x 10 ⁸	-1.51731 x 10 ⁷	-2.99622 x 10 ⁶	
9	9	1: Load Case 1	16.8204	1.61493	-39.479	-393958.	4.1135 x 10 ⁶	-1.19818 x 10 ⁷	8.29389 x 10 ⁷	-1.78428 x 10 ⁷	-4.72594 x 10 ⁶	
10	10	1: Load Case 1	21.2932	1.61493	-45.8668	-411156.	6.13451 x 10 ⁶	-1.2515 x 10 ⁷	4.29082 x 10 ⁷	-2.0329 x 10 ⁷	-6.76984 x 10 ⁶	

Cutting plane resultants for multiple result cases.

::: ?			Box	Girder - Cutti	ng Planes						×
Planes Integrals Values LIST: Single plane V Plane:1 Case:1: Load Case 1	Plane/Result Case	Point	X m	Y m	Z m	u (1) m	v (2) m	S1 Pa	S2 Pa	S3 Pa	^
Plane:2 Case:1: Load Case 1 Plane:3 Case:1: Load Case 1	Plane: 1 Case: 1: Load Case 1	1	3.33792	2.6875	21.9825	1.07257	5.25152	-2365.28	37340.4	1.362 x 10 ⁶	
Plane:4 Case:1: Load Case 1	Plane: 1 Case: 1: Load Case 1 Plane: 1 Case: 1: Load Case 1	2	3.41948 3.25927	2.75	21.9591 22.005	1.13507	5.33333	-1830.47 -2794.98	163127. 164179.	1.35/23 x 10° 1.39698 x 10 ⁶	
Plane:6 Case:1: Load Case 1	Plane: 1 Case: 1: Load Case 1 Plane: 1 Case: 1: Load Case 1	4	3.25733	2.625	22.0056	1.01007	5.33536	-906.049	-88295.2	1.36424 x 10 ⁶	
Plane:7 Case:1: Load Case 1 Plane:8 Case:1: Load Case 1	Plane: 1 Case: 1: Load Case 1	6	3.33792	2.6875	21.9825	1.07257	5.25152	-3097.27	32052.5	1.09477 x 10 ⁶	
Plane:9 Case:1: Load Case 1 Plane:10 Case:1: Load Case 1	Plane: 1 Case: 1: Load Case 1 Plane: 1 Case: 1: Load Case 1	7	3.41948 3.25927	2.75 2.75	21.9591 22.005	1.13507 1.13507	5.16667 5.33333	1097.41 -7321.77	160671. 161608.	1.08494 x 10 ⁶ 1.11461 x 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 x 10 ⁶	
	Plane: 1 Case: 1: Load Case 1 Plane: 1 Case: 1: Load Case 1	10	3.335	2.625	21.9602	0.947571	5.25456	-2773.62	-97634.2 -211362.	1.33116 x 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 x 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



Strand7 R3.1 Feature Summary

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

de	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 postprocessing of natural frequency results (graphical display, **Peek** and **LISTINGS**).

?	Results Options	×
Displacement	Envelopes	
☑ Add initial ☑ Consider e	GNL displacements to display lement offset in displacement results	Transient Displacement Relative to base Total
Reaction MP OUse defin OMove ori	PL in GNL - Moment Summation ned origin igin by average node displacement	Transient Velocity
NFA Modal I O Unit Mod O Engineer	Displacement dal Mass ring Modal Mass	Transient Acceleration Relative to base Total
		<u>O</u> K <u>C</u> ancel

Envelopes

Мо

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.

Strand7 ~) 🖬 🖬 📑 😁 🤐	Reference visual text	CASES LAYOUTS NO	TES SUMMARY LISTIN	GS BROWSE Building-R3 - Staged [Building-R3 - Staged	ed.HRA]
Load Freedor	m Combination Harm	onic Time Envelope	nfluence Combine Files			
Limit Envelopes	Combination Envelope	es Factors Envelopes	Sets			
* † * † X		¢∗ 24				
Limit MAG						
	1: Limit Min Envelope	2: Limit Max Envelope	3: Limit Abs Envelope	4: Limit Mag Envelope		
CASES	Limit MIN	Limit MAX	Limit ABS	Limit MAG		
1:0.0 s	~	\checkmark	\checkmark	~		
2: 0.040161 s	\checkmark	\checkmark	\checkmark	\checkmark		
3: 0.080321 s	\checkmark	\checkmark	\checkmark	\checkmark		
4: 0.120482 s	\checkmark	\checkmark	\checkmark	\checkmark		
5: 0.160643 s	\checkmark	\checkmark	\checkmark	\checkmark		
6: 0.200803 s	\checkmark	\checkmark	\checkmark	\checkmark		
7: 0.240964 s	~	\checkmark	\checkmark	\checkmark		
4 968 Noder	144 Reams A 926 Distor	O Bricke O Linke O	Vertices 0 Exces 0 Patt	or mm N + MPa	C I (154: 20:01) DS:10.0% Model Total Care	e: 4
4,500 NOUES	144 Deallis 4,000 Flates	U DHICKS U LIHKS U	venues uraces uraci	is initia in contra	C J (-134,-20,31) D3,10,0/6 WOULD TOTAL Case	3. 4

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.

Strand7 R3.1 Feature Summary



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.

?		Results Options		×
Displacement	Envelopes			
Averaging or	der			
🔿 Average, 1	then envelop	e		
Envelope,	then average			
Insert addit	ional stations	for beam forces and moments		
			<u>о</u> к	<u>C</u> ancel

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.

? Building-R3 - St	taged - 7: [Envelope:	Dead Load + Live Load +	Wind Loading] [Limit Max E	invelope 2] 🗙
• / 🔳 🖩	1: Ground Floor	Perimeter Columns : Solio	d Round	
Beam 73	End 1 [16] End 2 [2497]		
Quantity	LOCAL	End 1 [16]	End 2 [2497]	
 Displacement 	SFx (N)	5.295116 x 10 ³ (C.4)	2.890001 x 103 (C.5)	
End Force	SFy (N)	2.668187 x 103 (C.4)	2.668187 x 103 (C.4)	
O Section	AxiaIF (N)	3.265777 x 10 ⁵ (C.5)	3.063596 x 10 ⁵ (C.5)	
	BMx (N · mm)	1.077595 x 10 ⁶ (C.2)	7.095422 x 10 ⁶ (C.5)	
Oncease	BMy (N·mm)	4.885316 x 10 ⁵ (C.2)	4.988933 x 10 ⁶ (C.4)	
O Plane 1	Torque (N·mm)	4.112845 x 104 (C.5)	4.112845 x 10 ⁴ (C.5)	
O Plane 2	PRINCIPAL	End 1 [16]	End 2 [2497]	
O Plane x	SF1 (N)	5.295116 x 10 ³ (C.4)	2.890001 x 103 (C.5)	
O Plane v	SF2 (N)	2.668187 x 103 (C.4)	2.668187 x 103 (C.4)	
O Avial	AxiaIF (N)	3.265777 x 10 ⁵ (C.5)	3.063596 x 10 ⁵ (C.5)	
Axiai	BM1 (N·mm)	1.077595 x 10 ⁶ (C.2)	7.095422 x 10 ⁶ (C.5)	
User	BM2 (N·mm)	4.885316 x 10 ⁵ (C.2)	4.988933 x 10 ⁶ (C.4)	
Expanded	Torque (N·mm)	4.112845 x 10 ⁴ (C.5)	4.112845 x 10 ⁴ (C.5)	
\leftrightarrow 🗹 Cases				

Strand7 R3.1 Feature Summary



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.

?		Element Results Settings		×
/				
SHOW A	S		^	Facet 15 û deg
	None		_	
	Contour			Absolute values
	Vector			Subtract supports
QUANTI	γ		_	
	Displacement			
	Node Reaction			
	Stress		_	
	Element Force			
	Bending Moment			
	User-defined Result			
SYSTEM			- 1	
	Global 🥒			
	ucs 🖌			
	Restraint UCS			
COMPON	JENT			
	F(1)			
	F(2)			
	F(3)			
	M(1)			
	M(2)		~	<u>Settings</u>
Merge I	egends Apply to all view	5	<u>O</u> K	<u>C</u> ancel <u>Apply</u>

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.

?		I	Beam User-	defined Res	ult		×
H × I	× 1 🕯 🗉	•					
Name: Na	me cannot l	be blank.					
							~
Equation:							
0							^
							~
Force com	ponents:						
SF1	SF2	SFx	SFy	Axial			
Moment co	omponents						
BM1	BM2	BMx	BMy	Torque			
Strain com	ponents:						
CurvD.1	CurvD.2	AxiaID	CurvT.1	CurvT.2	AxialT	Twist	
Section co	mponents:						
111	122	lxx	lyy	lxy			
Α	J	SA1	SA2	S11	S22	Sxx	Syy
Z11+	Z11-	Z22+	Z22-	Zxx+	Zxx-	Zyy+	Zyy-
Operators	and consta	ints:					
+ -	*	/ ^	() E	πε	2	
Functions:							
Sqrt	Sqr	Sin	Cos	Tan	Min	Angular	Units
Abs	Sign	Arcsin	Arccos	Arctan	Max	Degr	rees
Round	Trunc	Ln	Log	Exp		OBC	
IfPos	lfPosB	lfNeg	lfNegB	lf			ans
							Close

Strand7 R3.1 Feature Summary

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).



Strand7 R3.1 Feature Summary

?		Base	Displacement	vs Time rigid I	inks + DXYZ - Rel	To Base - Graph
Graph 2						
2D Graph	Lines	Points	3D Graph	Filtering		
Line - Point		х	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 Displacemen	nt vs Time rig	d 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).



Strand7 R3.1 Feature Summary

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.

Strand7



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.

Strand7 R3.1 Feature Summary



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.