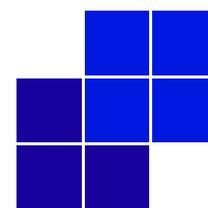


News.St7



Newsletter for Strand7 and Straus7 users

Strand7 is marketed as Straus7 in continental Europe

Issue 8, September 2010

Welcome to another issue of News.St7. The articles in this issue focus on some of the many new features in Strand7 R2.4. We present an overview and example of the Soil In-Situ Stress tool, which calculates the steady-state (initial) stress state in soil models taking into account fluid content and pore pressure in the soil; it is an important step in soil analysis. We also cover in more detail the entities, attributes and tools available for attaching dissimilar meshes together, in particular the edge attachment options for plate elements. New functionality in the Strand7 API is also presented - namely, the ability to launch the Strand7 Model Window through the API and to make it part of your own application.

As usual, there are new *Did You Know?* items plus information about Strand7 Training, and our participation at upcoming exhibitions around the world. We are also considering to run a Strand7 course in Abu Dhabi in late November or early December. The course is expected to run for five days and to cater for both new and experienced Strand7 users. The course can only go ahead subject to there being sufficient participation, so please contact us as soon as possible if you are interested in attending.

Since Strand7 R2.4 was released last December, we have issued two point updates - the latest release is now Strand7 R2.4.3. If you did not receive our emails announcing the updates, please send us your contact details so that we can amend our mailing list. This will ensure that you receive a timely notification for future updates.

We hope you enjoy reading this issue of News.St7 and look forward to hearing from you with comments on its contents, or with suggestions for topics you would like us to cover in future issues

If you have any feedback or suggestions regarding the content of News.St7 or you would like to contribute a case study showing how you use Strand7, please email us at news.st7@strand7.com. If you would like to receive your copy of News.St7 automatically by email, simply send a blank email to newsletter-subscribe@strand7.com. All care is taken to ensure that information in News.St7 is accurate and up to date at the time of publishing. However Strand7 Pty Ltd accepts no responsibility for inaccuracies in, or changes to, such information.

In this Issue

• Soil In-Situ Tools	1
• API Windowing Tools	5
• Attaching Dissimilar Meshes	6
• Free Body Diagrams	7
• Factor vs Position Table	9
• API Troubleshooting	11
• Training and Exhibitions	12

Soil In-Situ Tools

In Strand7 Release 2.4 the soil modelling capabilities have been greatly expanded from the previous releases. This release has seen the addition of new soil material models as well as several modules and element attributes that help the user to define the soil's initial stress state. This article takes a look at these tools.

The behaviour of a soil model in a geotechnical analysis is highly dependent on the in-situ (initial or in position) conditions of the soil, i.e. its stress state. The soil in-situ conditions represent the stress history of the sample, and are generally specified in the soil material models by input parameters to define the influence of the stress history upon the soil behaviour under the current loading situation.

In Strand7, the soil in-situ state can be defined using dedicated tools. The **Auto Assign Soil In-situ Stress**

module (**Tools/Auto Assign/Soil In-situ Stress**) is designed to automatically calculate and assign the soil in-situ stress state that gives the initial equilibrium condition. Two Strand7 element attributes are also available for the definition of soil model-specific parameters, the **In-situ Stress** and the **In-situ Ratios**.

Soil In-Situ Stress State

The soil in-situ stress state is the three-dimensional stress distribution within a body of soil that maintains the equilibrium condition. This can be considered as two separate components: the vertical, or the direction in which the gravity is acting, and the horizontal, or the direction that is perpendicular to the gravity. In theory, the vertical component can be more easily calculated when compared to the horizontal component, as the mechanics in the vertical direction are relatively simple to describe. Therefore the horizontal stress component is generally expressed as a factor of the vertical component.

Trapped Fluid and Soil Material Parameters

Real life soil samples often contain micro-pores or cavities in the soil skeleton structure, which are mostly filled with air and/or fluid. Under loading, and depending on the permeability of the soil, the fluid may flow through the soil structure or may be partially retained in the pores. The two limiting cases of this behaviour result in the so-called *drained* and the *undrained* states of the soil.

The undrained state is where the fluid within the voids of the soil is not allowed to displace out of the soil structure under loading. This means that any change in load will change both the stress in the soil skeleton and the pressure in the fluid. The drained state, on the other hand, is where the fluid is completely free to displace out of the structure under loading. This means that any change in load will change only the stress in the soil skeleton, with the fluid pressure remaining constant. In reality, the response of soil to changes in load is a transient one, where the soil state is somewhere between the drained and undrained states. Nevertheless, the assumption of undrained behaviour is usually a good approximation to the short term effects of loading changes on soil, whilst the drained behaviour is usually considered to be representative of the long term response.

The soil mass therefore experiences both the stress carried by the soil skeleton (known as the *effective stress*), and the stress in the fluid (known as the *pore pressure*). The combination of these two stresses is known as the *total stress* of the soil. In Strand7, these two components are clearly separated when using soil elements; stress results for soil elements are given as effective stresses, with the pore pressure also available as an output option. Effective stresses are also used for the in-situ stress attributes applied to elements.

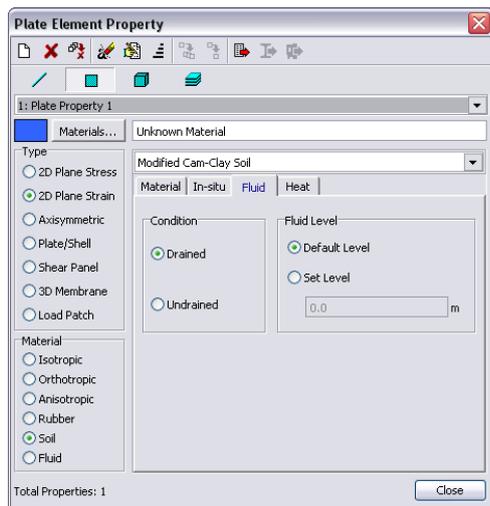


Figure 1.1: Strand7 Soil Material dialog, Fluid tab.

It is important to know that in laboratory tests of the shear strength, soil samples can be tested under drained or undrained conditions. This can result in different material properties, which will affect the subsequent calculation of results. In Strand7, the element property definition requires the drained material properties; the effect of the fluid for the undrained case is added to the element behaviour based on the defined fluid characteristics (i.e. bulk modulus, density, level). Fluid properties may be

defined in the **Fluid** tab of the soil property dialog, with the default parameters set via the **Soil/Fluid** option of the solver dialog.

In addition to the drained/undrained soil state, real life soil samples can also be classified by the degree of soil saturation, or the percentage of the pores filled with fluid. This is a significant factor in the transient analysis, where the fluid displaces from the soil skeleton due to consolidation. Currently Strand7 considers only two states of soil saturation: the fully saturated state, where all pores are filled by fluid, and the unsaturated state, where all pores are filled by air.

Did You Know?

Middle Button for Dynamic Rotate

As of R242 you can now quickly switch to dynamic rotate mode by simply clicking the middle mouse button. The equivalent keyboard shortcut is the F4 key. Refer to the Online Help for other shortcuts.



Soil In-Situ Stress Calculations

Assuming the drained condition, the vertical stress component of the in-situ stress state, S_v , can be calculated from the equilibrium condition between the weight of the soil and the buoyancy force due to any fluid within the voids of the soil structure. This gives the following equilibrium form.

$$S_v = -\rho_{soil}gh_{soil} + \rho_{fluid}gh_{soil_sub} \quad (1)$$

where

- ρ_{soil} = density of the soil skeleton;
- g = gravitational acceleration constant;
- h_{soil} = height of the soil element measured from the ground level;
- ρ_{fluid} = effective density of the trapped fluid, taking into account that the volume of fluid is a function of the voids in the soil; Strand7 offers the *void ratio*, e , to define this; and
- h_{soil_sub} = height of the soil element that contains trapped fluid, measured from the fluid reference level.

In the unsaturated case, as the voids are filled with air, only the soil skeleton is contributing to the stress state of the soil. This gives the following equilibrium form.

$$S_v = -\rho_{soil}gh_{soil} \quad (2)$$

The horizontal stress component is calculated from the following relationship.

$$S_h = K_0 S_v \quad (3)$$

where

$$K_0 = \text{ratio of horizontal to vertical stress.}$$

The stress ratio K_0 is generally obtained from the laboratory testing of the soil sample. A common assumption made about this parameter is based on the drained friction angle ϕ' of the sample.

$$K_0 = 1 - \sin \phi' \quad (4)$$

This relationship can be used if no laboratory tests have been conducted. (See Bowles, J.E., *Foundation Analysis and Design*, McGraw-Hill, 4th Edition, 1988.)

Soil In-Situ Stress Attribute

The plate and brick **Soil In-situ Stress** attribute is applicable to all soil models to describe the soil in-situ conditions discussed above. This attribute defines the effective (not total) vertical stress (i.e. the stress experienced by the element in the direction of gravity) and the horizontal stress ratio, which is an estimate of the stress in the direction perpendicular to the applied gravity.

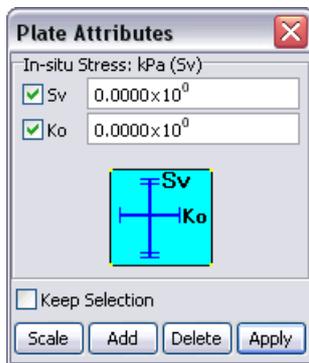


Figure 1.2: Strand7 Plate In-situ Stress attribute dialog.

If applied, the parameter K_0 in this dialog overwrites the value of the same parameter defined in the **Element Property** dialog.

Auto Assign Soil In-Situ Stress Module

The **Auto Assign Soil In-situ Stress** module (**Tools/Auto Assign/Soil In-situ Stress**) can perform the stress calculation and assignment automatically. This module can be applied to any of the Strand7 soil models and considers both soil and non-soil elements contained in the model.

To perform the stress calculation, the module requires the following parameters:

1. The type of fluid trapped within the soil structure – The calculation requires the fluid density to be entered. The default option is water, which can be entered automatically by clicking the **Water** button.
2. The reference level for the trapped fluid – This specifies the global reference level for the fluid. If the fluid reference level is set at the element property level, then the property values will be used instead.

This allows for different parts of the mesh to have different fluid reference values.

3. The load case that contains the gravity definition – The **Load case defining gravity** dropdown list in the **In-situ Stress Generation** dialog is used to select a load case that specifies the direction and magnitude of gravity used for the in-situ stress state calculation.
4. The boundary conditions – The **Freedom case** dropdown list in the **In-situ Stress Generation** dialog specifies the restraint conditions of the mesh. This represents the restraints on the structure at the in-situ equilibrium state (i.e. before any additional loads are added or any changes are made to boundary conditions or staged construction changes occur).
5. The stage for which the in-situ calculation is to be performed – This option is used in construction sequence analysis, where a stage representing the initial situation is used to calculate the in-situ conditions.

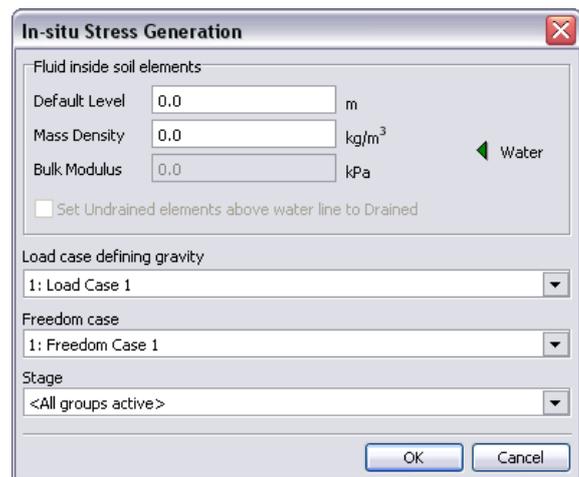


Figure 1.3: Strand7 Soil In-situ Stress Generation dialog.

The module calculates the stress distribution according to the defined settings, assuming drained conditions, and automatically applies the element **In-situ Stress** attribute to all elements within the selected stage. It aims to determine the in-situ stresses that equilibrate the applied gravity loads (soil weight and fluid pressure), such that the net vertical displacement is approximately zero. It is possible that, in some meshes, the stress calculation will return a warning message about the convergence of the calculation. This indicates that the applied **In-situ Stress** attribute has to be checked to ensure that the correct magnitude is being applied to the mesh.

Because the soil elements generally require the application of initial in-situ stress, when these are not applied, Strand7 automatically calculates an approximate in-situ stress distribution before the solver is run - this distribution is approximate because it considers only the geometry and not the stiffness of the elements. Strand7 will issue a warning message to notify the user about this automatic, approximate stress calculation.

Soil In-Situ Ratios Attribute

The **Soil In-situ Ratios** attribute allows for the specification of the initial void ratio and the Over-Consolidation Ratio (OCR). OCR is the ratio of the maximum stress in the history of the soil to the current stress and is applicable only to the Modified Cam-Clay soil model. This attribute, if applied, takes precedence over the default values specified in the **Element Property** dialog.

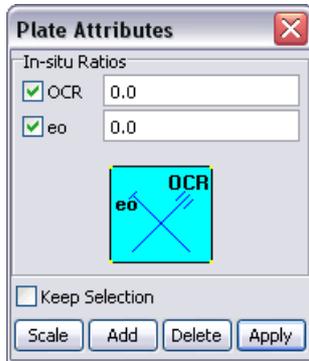


Figure 1.4: Strand7 dialog for plate In-situ Ratios attribute.

Example – Fluid State in a Soil Sample

The following example is a simple demonstration of the Strand7 soil in-situ calculations. We will consider three cases of soil condition.

- The soil is saturated through the entire depth.
- The top half of the soil is unsaturated while the bottom half is saturated.
- The entire depth of soil is unsaturated.

The soil sample is treated as a 2D Plane Strain case 10 metres deep and 2 metres wide. The mesh of each soil sample is shown in the figure below.

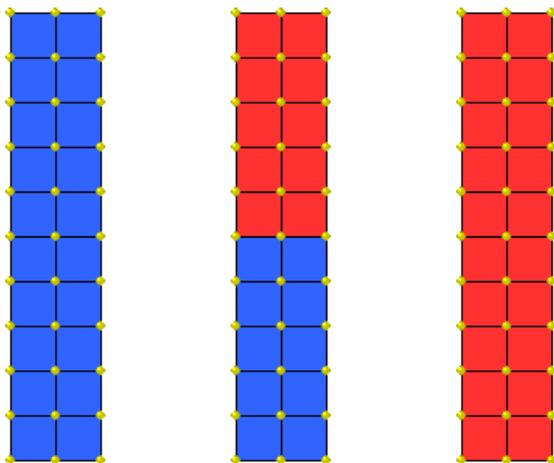


Figure 1.5: The mesh representing the column of soil. The blue elements represent saturated soil, and the red elements represent unsaturated soil.

The soil void ratio for the saturated soil is taken to be 0.2, while for the unsaturated soil the void ratio will not be relevant. A horizontal stress ratio of 0.5 is assumed. The bulk density of the soil is taken to be 1500 kg/m³ (this is the mass density of the dry soil), and the fluid filling the void is taken to be water.

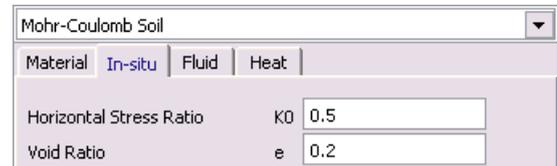


Figure 1.6: Soil In-situ parameters for saturated soil defined in the Strand7 Plate Property Definition dialog. We can apply a Void Ratio of 0.0 for unsaturated soil.

Now we apply the in-situ stress using the **Auto Assign** tool described above.

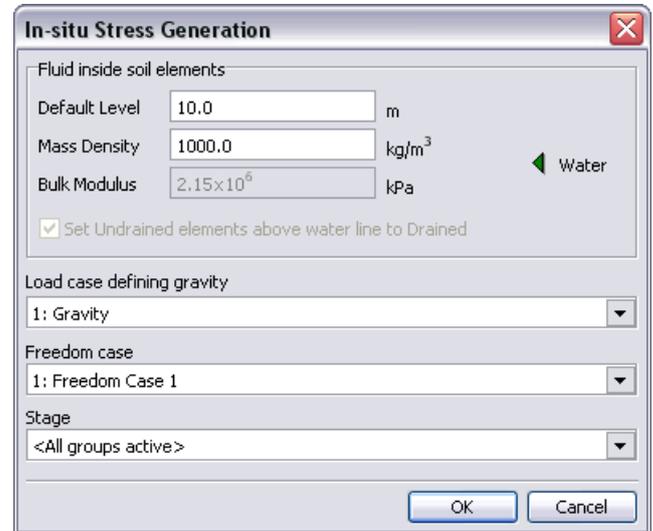


Figure 1.7: The In-situ Stress Generation dialog, using the settings described above.

From the soil in-situ stress contours shown below we can observe the difference between the saturated and unsaturated conditions.

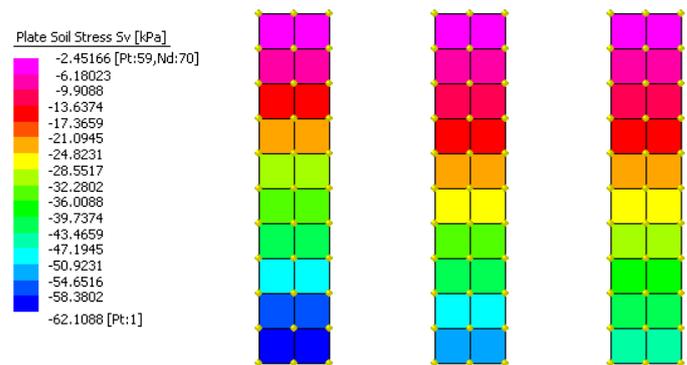


Figure 1.8: Vertical In-situ stress contour for the three cases.

When the model is solved under the action of gravity only, including the automatically calculated in-situ stress values, the displacement contours below show that the vertical displacement in all three cases is zero or very small. This agrees with the in-situ condition definition discussed above.

Did You Know?

User Contour of Principal Stress with Largest Magnitude

For plate and brick elements a User Contour based on stress and strain result quantities can be defined in Results Settings. This is useful for plotting contours of derived quantities based on the elemental stress and/or strain, such as a contour of principal stress with largest magnitude according to:

$$\sigma_{\max} = \text{MAX}(\sigma_{11}, -\sigma_{33})$$

This allows you to display both maximum principal stresses in the same contour. Enter the following formula into the user defined contour for plates:

$$\text{IFPOSB}(\text{ABS}([S11]) - \text{ABS}([S22])) * [S11] + \text{IFNEGB}([S11] - \text{ABS}([S22])) * [S22]$$

This produces the signed principal stress with largest magnitude.

IFPOSB(X) evaluates to +1.0 if X is positive and 0.0 if X is negative or zero.

IFNEGB(X) evaluates to +1.0 if X is negative and 0.0 if X is positive or zero.

To plot the absolute value of the principal stress with largest magnitude enter:

$$\text{IFPOSB}(\text{ABS}([S11]) - \text{ABS}([S22])) * \text{ABS}([S11]) + \text{IFNEGB}([S11] - \text{ABS}([S22])) * \text{ABS}([S22])$$

Substitute [S33] for [S22] when contouring for bricks.

The signed principal stress with largest magnitude for bricks is shown below.

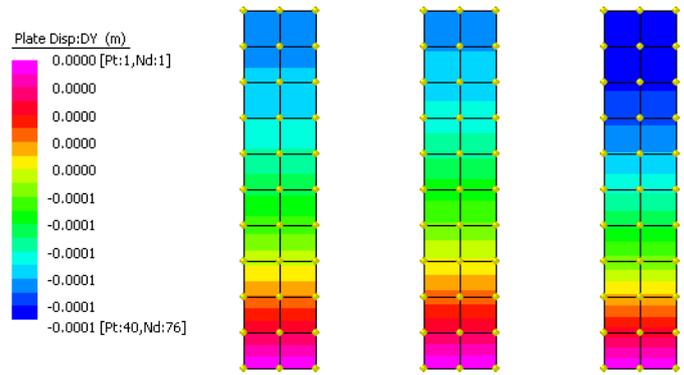
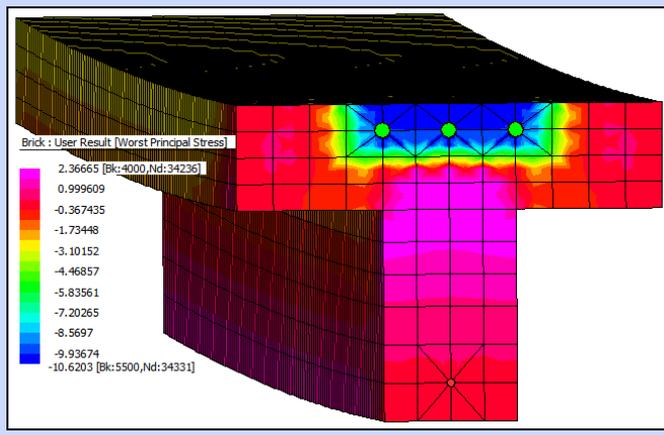
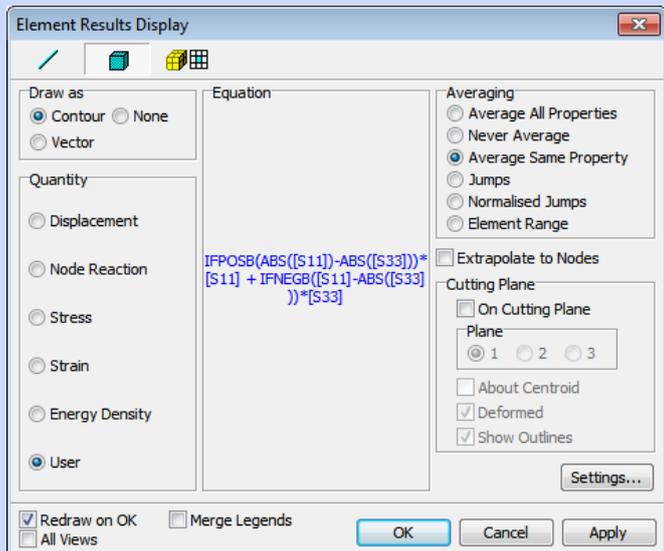


Figure 1.9: Displacement contours of the soil under gravity only.

In a typical soil analysis, this would be the starting condition for the simulation. Normally the first increment would be the gravity + in-situ stress conditions, with subsequent increments applied to simulate the effects of construction, tunnelling, etc.



Figure 1.10: Relative Displacements dialog.

A feature of the **Results Settings** in Strand7 that is particularly useful in the analysis of soil problems is the **Reference Displacement** option (**Results/Reference Displacement**). This option can be used to select a reference result case, whose displacements are subtracted from all subsequent increments so that relative displacements can be visualised easily.

API Windowing Tools

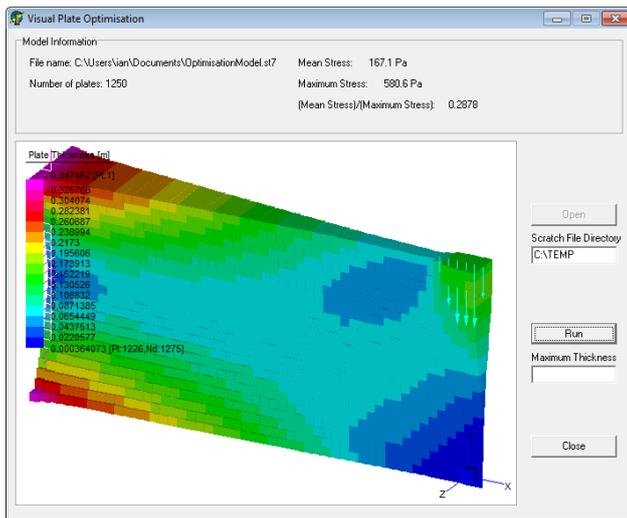
Strand7 R24 introduces new API functionality to allow you to open interactive Strand7 model view windows, giving instant graphical feedback on your models and results.

To display a model window in your application, it is necessary to first open a new or existing Strand7 model. A model window can then be created by calling **St7CreateModelWindow**. The window is not displayed at first. To show the window call **St7ShowModelWindow** and to hide the window call **St7HideModelWindow** or simply close the window. The window is destroyed when the model is closed. You can explicitly destroy a model window by calling **St7DestroyModelWindow** prior to closing a model.

The model window is displayed as a resizable window that moves independently of your application. It is possible to embed the model window within a window in your application for an integrated appearance. This can

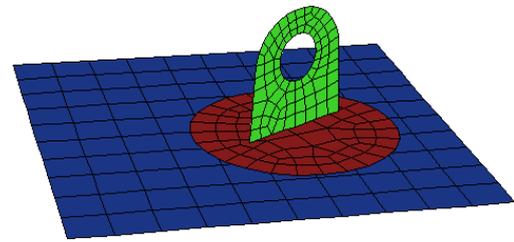
be achieved by passing a window handle to the function **St7SetModelWindowParent** or by getting the model window handle using the function **St7GetModelWindowHandle** and assigning it as a child to an existing window. It is even possible to customise the appearance of the model view window. Functions exist to show and hide the top panel, toolbar, status bar and right-click popup menu.

When changes are made to a model, the API does not automatically update the model window. It is necessary to call **St7RedrawModel** to update the display. Some model changes such as changes to beam and plate properties are not detected on redraw and a call to **St7UpdateElementPropertyData** is first necessary to synchronise the display database.



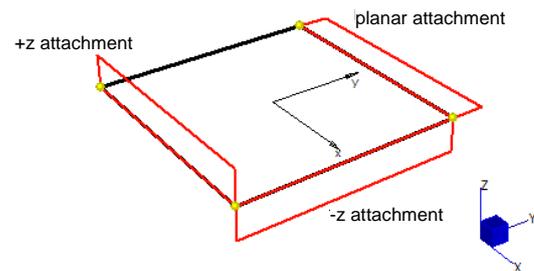
Entity selection can be performed using the model view window. The selection tools are available from the Selecting menu option on the right-click popup menu. From your program code you can then use the function **St7GetEntitySelectState** to check whether a given entity has been selected.

An example has been created to demonstrate the windowing tools available in the API. This example is an extension of the plate optimisation demo that ships with Strand7. The demo is available as a Microsoft Excel spreadsheet written in VBA and can be downloaded from www.strand7.com/news.st7.htm

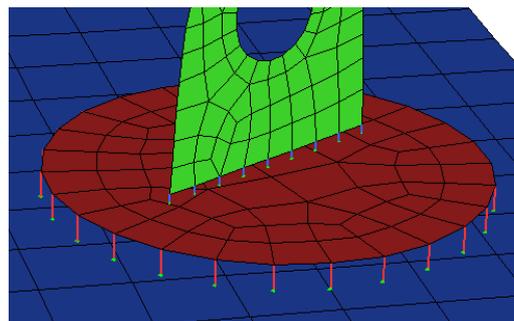


Although it is possible to generate compatible meshes to connect the parts together conventionally, one quick and easy solution is to use the **Plate Edge Attachment** attributes to stitch the parts together.

The **Plate Edge Attachment** attribute is accessed via **Attributes/Plate/Attachment/Edge** and can also be applied to geometry edges via **Attribute/Edge/Attachment**. The edge attachment can be defined in three different directions: **Planar**, **+z** or **-z**. The distance that the attachment should use to search for elements to attach to is defined as part of this attribute.



The **Planar** attachment is used to attach the lug to the round panel whereas the **+z** or **-z** attachment (depending on the orientation of the panel) is used to define the attachment between the edges of the round panel and the base plate.



The attachments should cover the distance between the mid-planes of the attached parts to account for their thicknesses. The attachments can be made **Flexible** to approximate the weld stiffness. **Rigid** and **Direct** connections are also available if suitable for the type of analyses.

After the attributes have been applied it is then necessary to activate the attachment process. This is done using **Tools/Attach Parts**. For a **Flexible** attachment, this tool creates beam elements up to the maximum distance defined in the attribute (it creates rigid links if using **Rigid** type) to coincide with the plane of the elements being attached to. A special link is also created at the end of each beam element (an **Attachment Link**) which directly couples the displacements at the node on the beam to the shape functions on the appropriate plate element. The link ensures that the connection is a fully compatible one

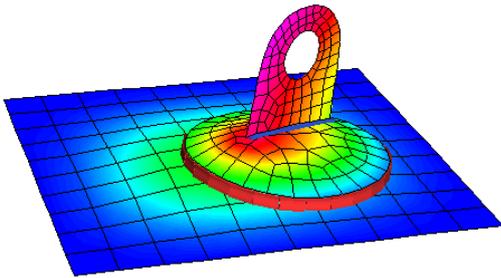
Attaching Dissimilar Meshes

Using Edge Attachment Attributes to Attach Dissimilar Meshes Together

Sometimes the geometry of a problem dictates the layout of the finite element mesh. For instance, the following model consists of independent meshes from a base plate, a round panel and a lug which are quite regular but are not compatible at the interfaces. The lug is supposed to be welded onto the round panel while the round panel is welded around its edges onto the base plate.

that takes into account all the active degrees of freedom on the beam and on the connecting plate. The link couples the two together and transfers loads into the plate element from the beam element.

With the attachments created, a lifting load applied to the lug hole not only lifts the lug, but also deforms the attached part underneath.



In summary, attaching parts together is a two-step process:

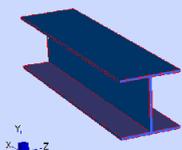
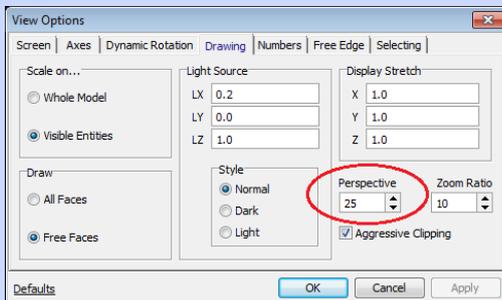
1. Apply the appropriate type of attachment attributes to the elements,
2. Attach the parts together by using **Tools/Attach Parts**.

Note that these attachments act to make the motion between the two bodies compatible and represent the stiffness of the joint. They should generally be used in cases where you need to transfer load. For a detailed analysis of the connection itself, a compatible mesh is preferable .

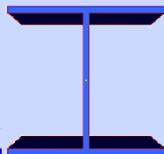
Did You Know?

Perspective in 3D Drawing

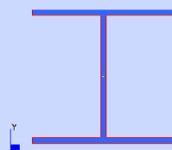
Strand7 allows you to change the perspective setting in a 3D model (**View/Options/Drawing**) so that you can see the 3D model on a plane view (say YZ plane view) more clearly. The setting is simply a number that ranges from 0 (no perspective) to 200 (very high perspective, like a fish-eye lens). It can be very useful when viewing/editing a complex 3D model.



Isometric view with Perspective set to 25 (default)



YZ plane view with Perspective set to 25 (default)



YZ plane view with Perspective set to 0 (parallel view - no perspective)

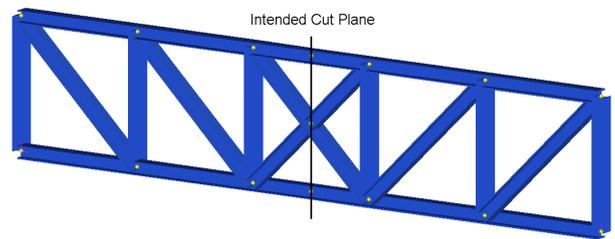
Free Body Diagrams

How to get Free Body Diagrams without a Submodel

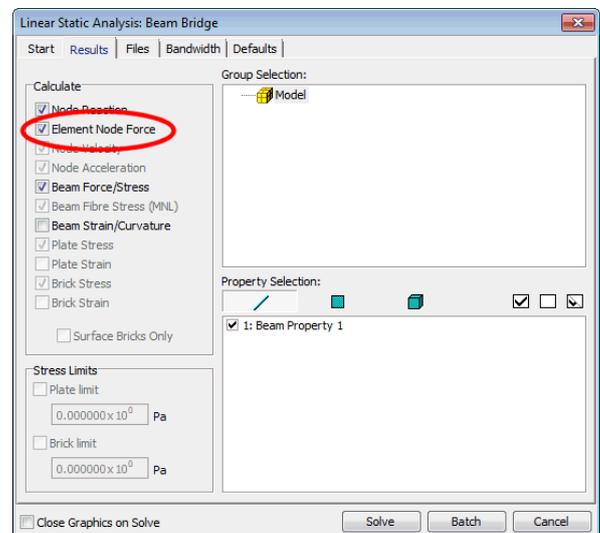
Extracting forces and moments from simple models made from beam elements has always been straightforward with Strand7. All that needs to be done is draw a force/moment diagram and take the values at the required locations.

In previous versions of Strand7, to extract a total of the forces and moments carried by multiple beam, plate and brick elements in a cross-section it was necessary to create a submodel (creating equivalent prescribed displacements), analyse the submodel and then use the **Peek** tool to calculate the force and moment total at the restrained nodes. If several locations needed to be investigated, then a submodel was required for each one.

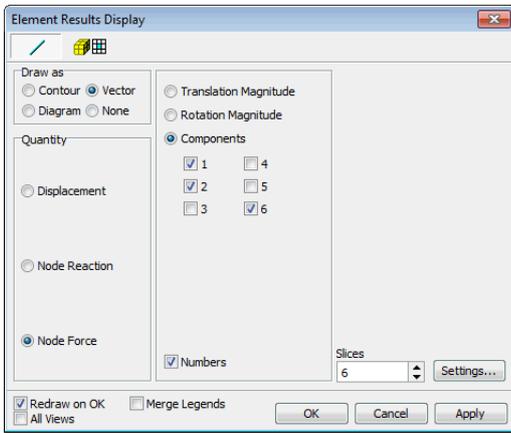
A new calculated result is available for all element types in Strand7 R24. This is the **Element Node Force** quantity, which can be displayed as a **Vector** plot. Before analysing the model, ensure that there is a node or a line of nodes at the location where the forces and moments are to be calculated.



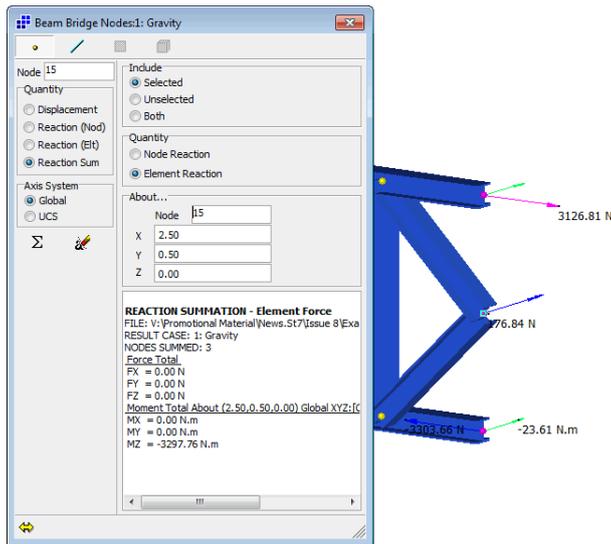
1. In the solver, calculate the **Element Node Force** results.



2. Hide the unwanted elements. Groups are often used for this.
3. Show a **Vector** plot of **Node Force**.



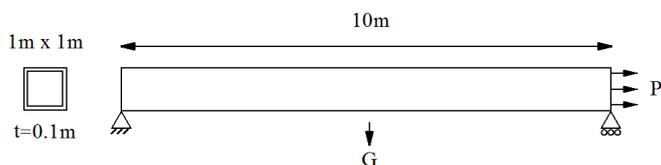
4. Select the nodes to be used for the calculation.
5. Use the **Peek** tool to calculate the force and moment sum.



The moment summation needs to be about a specific point, the default being the Global origin. If there is a node in the model which should be used, click the **Node** box to activate the Hotpointer and select the relevant node. If there is no node then the co-ordinates can be entered manually. In this example, the node in the middle of the section was used.

The **Peek** tool allows you to sum the selected nodes, the unselected nodes or all the nodes. The option to include all nodes is a useful way to check that a group of connected elements is in equilibrium (zero total reactions should be calculated for a static problem).

In a second example, a brick mesh was used to model a square hollow section beam defined as follows:



$$L = 10m$$

$$A = 0.36m^2$$

$$\rho = 7870kg/m^3$$

The beam was simply supported at each end at the centre of the hollow square section.

The applied loads were gravity, $g = 9.80665 \text{ m/s}^2$ and an axial pressure at the end, $P = 100 \text{ kPa}$ putting the beam in tension.

Force/Moment Calculations

Load per unit length due to gravity

$$w = gA\rho = 9.80665 \times 0.36 \times 7870 = 27784.2N/m^2$$

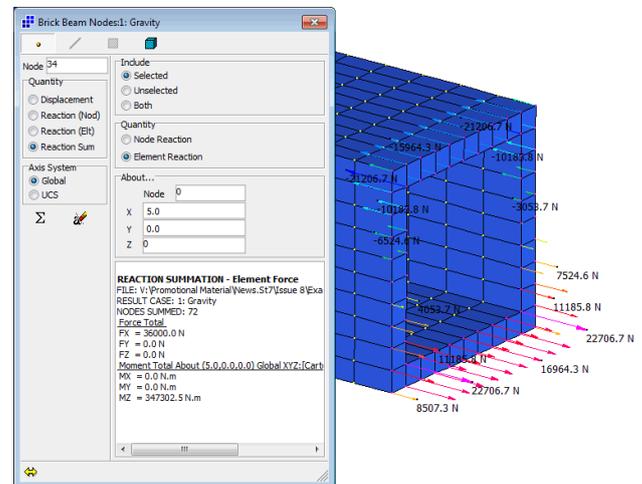
Maximum bending moment expected in the middle of the simply supported beam

$$M = \frac{wL^2}{8} = \frac{277842 \times 10^2}{8} = 3473025Nm$$

Axial force expected in the beam

$$F = PA = 100000 \times 0.36 = 36000N$$

The model's **Element Node Force** results were calculated and the axial force **Vectors** plotted for half of the model. The nodes in the middle of the beam were selected and the **Peek** tool opened. There was no node in the middle of the beam cross-section, so the **About...** location was entered manually in the **Global XYZ** system.



The force and moments summation shows a total axial force of 36000N and the bending moment 347302.5Nm, both agreeing exactly with the expected results.

This method can also be used in models where there are beams, plates and bricks at the cut. It is a dynamic tool in that as you hide elements, the nodal forces and moments contributed by the hidden elements are automatically subtracted from the remaining elements shown in the model window. Conversely, as elements are added to the display, the resulting vectors automatically update to add back the element's nodal force contribution.

When the whole model is shown, reaction vectors are expected only at restrained nodes. For nonlinear analysis, the reaction vectors are also a good way of visualising any unbalanced force remaining in the model due to the iteration process .

Factor vs Position Table

In Strand7 R24, a new type of table, **Factor vs Position** table, is available. There are two typical applications of this table. The first is for applying a complex pattern of attributes, such as distributed loads on beams, edge pressures on plates, etc. The second is for modelling situations where forces depend on position, such as buoyancy. The following examples demonstrate these two applications.

Applying Attributes of a Complex Pattern onto a Model

Often one needs to apply loads which are of a complex pattern such as surcharge loads on soil or wind pressure on a building. If the complex pattern does not conform to a relatively simple equation so that the functions in Strand7 can be used to define it, the **Factor vs Position** table can be a very helpful option.

Suppose that we want to apply a surcharge pressure load onto the soil modelled as plates as shown in Figure 5.1. The distribution of the surcharge load is difficult to describe with a simple equation and so we use the **Factor vs Position** table to perform the task.

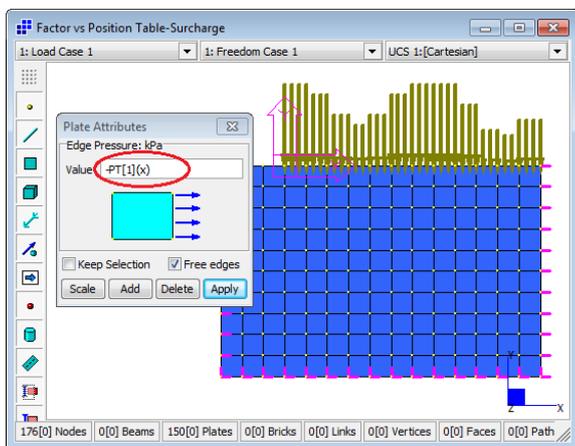


Figure 5.1: Soil model with surcharge load.

Firstly, we define a **Factor vs Position** table by using **Tables/Factor vs Position** to represent the pattern to be applied as shown in Figure 5.2.

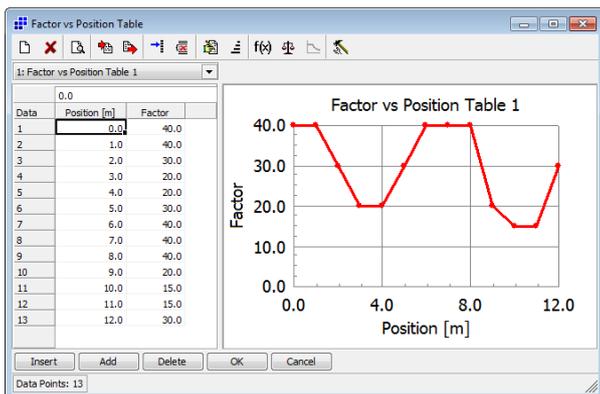


Figure 5.2: Defining a Factor vs Position table to represent the loading pattern.

Next we assign the surcharge pressure described by the table curve as plate edge pressure by referring to the defined table with an expression of the form **PT[1](x)** where **PT** represents Position Table, 1 is the table identification number, and **x** is the X coordinate at the point being interpolated (note here that UCS1 is the current coordinate system and so the table refers to this system as it interpolates **x** from the horizontal axis of the table - see Figure 5.1). This expression, **PT[1](x)**, is entered directly into the attribute dialog edit box.

The **Factor vs Position** table follows the extrapolation conventions of the other table types. Therefore, when the to-be-applied length is outside the range of the defined table, extrapolated data will be used.

Used for Modelling Buoyancy

Suppose we want to find the steady state position and the internal stress distribution within the structure of a loaded boat when it is stationary in water. How can we do this using Strand7?

The boat is in a stable (equilibrium) state under the action of gravity (self weight of the boat plus cargo) and buoyancy force.

The challenging part is the modelling of the buoyancy force. According to Archimedes' principle, the buoyancy force depends on the submerged volume (or depth) of the boat. For a curved hull, it is not easy to work out this submerged depth (i.e. where the water line is on the hull).

By using the **Factor vs Position** table together with the nonlinear transient dynamic solver, this problem can be solved without much difficulty.

For the sake of simplicity and clarity, a simple example is used to demonstrate the modelling of buoyancy. More information and examples can be found in the Strand7 Verification Manual (VNT11) and in the Online Help (**Special Topics\Factor vs Position Table\Example 2**).

Let's consider the 2D plate model shown in Figure 5.3, which represents the cross-section of a long block being lowered into the water.

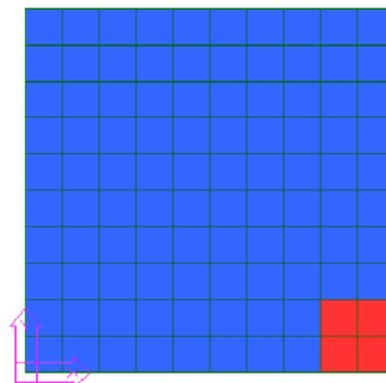


Figure 5.3: 2D plate model at initial position representing the cross-section of the block.

The whole block consists of two types of material as indicated by blue and red properties. The blue part has a negligible density and the whole weight of the block (set to be half of the total volume of the block multiplied by the density of the water and acceleration due to gravity) is concentrated in the red part. Initially the bottom face of the block is in contact with the surface of the water marked by the X axis of the UCS, i.e. the water level is at $Y=0$ in the UCS reference system. Our task is to find the final position of the block in the water.

As mentioned earlier, floating forces depend on the submerged depth. This position dependence can be modelled by using the **Factor vs Position** table which scales the element load attributes according to the position of the element with respect to the reference level.

We first define a **Factor vs Position** table as shown in Figure 5.4. This table represents the relationship between the hydrostatic pressure and the depth, i.e. $p = \rho gh$ where p is the pressure, ρ is the density of water (1000 kg/m³ in the example), g is the acceleration due to gravity (10 m/s² in the example) and h is the depth (the position of a point of the plate in the Y direction of the UCS reference). We then apply (**Attributes/Plate/Edge Load/Pressure**) a unit edge pressure on the four edges of the plate (the plate may be submerged totally into the water therefore the pressure is applied to all four edges, see Figure 5.5). By associating this unit pressure with the defined table on the nonlinear transient dynamic solver (**Solvers/Nonlinear Transient Dynamic/Load Tables**), buoyancy force will be automatically handled by the transient solver (see Figure 5.6). Note that in the table in Figure 5.4, when the position is greater than zero, the factor is zero. That is, if any part of the edges of the plate is above the water level, the pressure on it will be zero. Also note that in Figure 5.6, the **Factor vs Position** table is assigned to axis 2 (the Y axis) of UCS1 system. The table can only be assigned to one axis of a reference system in one load case, but it can be used by more than one load case.

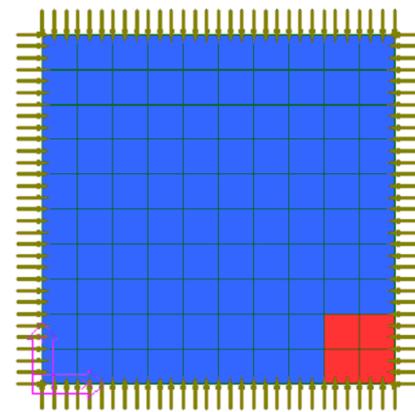


Figure 5.5: 2D plate model with unit edge pressure.

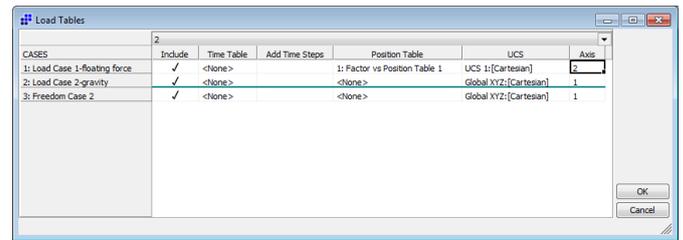


Figure 5.6: Assign the table to the unit pressure load case.

Solving the model using the nonlinear transient dynamic solver, the final position of the block in the water and its stress distribution are calculated. Since the block undergoes large displacements, the **Nonlinear Geometry** option on the solver panel should be set. Also, damping is needed to allow the model to eventually come to rest so that the block reaches a stable state. Rayleigh damping is used for this example (**Solvers/Nonlinear Transient Dynamic/Added Damping/Rayleigh**).

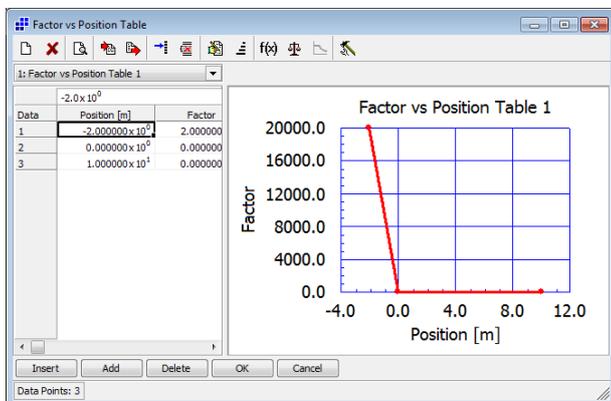


Figure 5.4: Factor vs Position table representing buoyancy force.

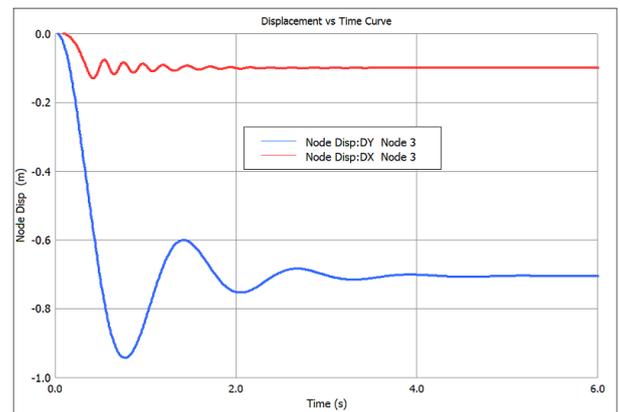


Figure 5.7: Displacement curves at node 3.

Figure 5.7 shows the vertical and horizontal displacements at node 3 (at the bottom right corner of the red part). Initially, gravity load is greater than the buoyancy force and so the block quickly moves downwards. Inertial force will make the block move beyond the static equilibrium line. With the buoyancy force increasing, the block starts to move back up. After a period of oscillation, the block finally reaches its static equilibrium position as indicated by the level part of the displacement curves. At this position the weight is balanced by the buoyancy force. Note that due to the uneven distribution of the mass, the block also rotates and

swings as it sinks. Figure 5.8 shows the final stable position of the block. Click Figure 5.9 for an animation showing the whole process.

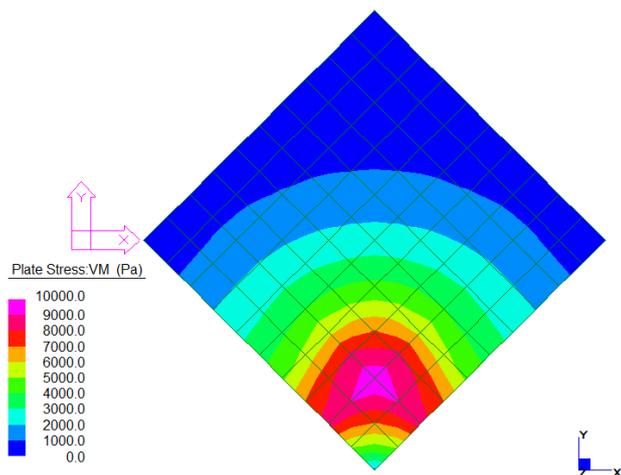


Figure 5.8: Final stable position of the block.

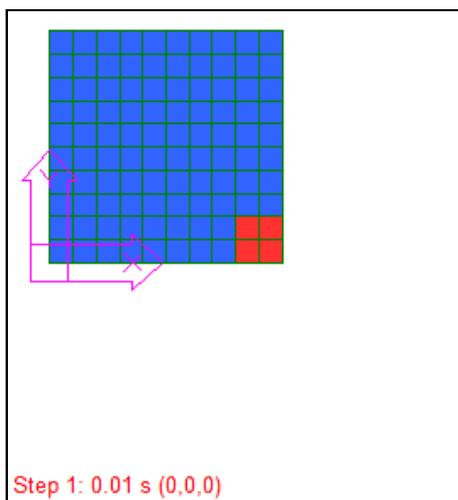


Figure 5.9: Animation of process. Click to play.

It can be seen that the heavier red part is at the lowest level and half of the block is submerged into the water 

How does your program find ST7API.DLL?

Windows searches for a DLL in the following order:

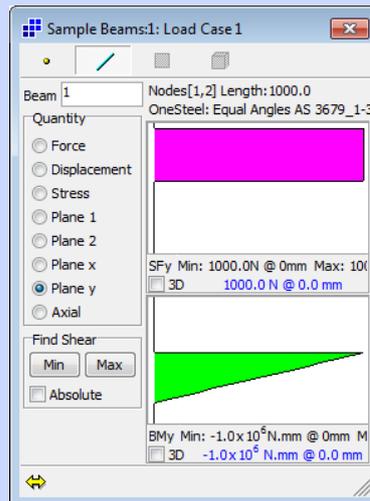
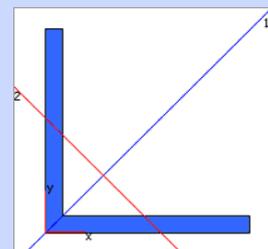
1. The directory where the application executable is located.
2. The Windows system directory.
3. The Windows directory.
4. The current directory.
5. The directories listed under the Path environment variable.

For computers with multiple installations of Strand7, it may not be obvious as to which ST7API.DLL is being used by your program. Two API functions are available to help identify the DLL in use. The function **St7GetPath** returns the path to the DLL and the function **St7APIVersion** returns the Strand7 version of the DLL.

Did You Know?

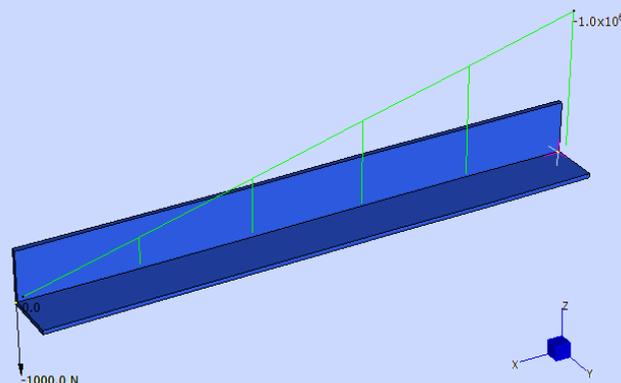
Beam Local and Principal Results

Cross sections in beam elements contain both local and principal axes and they do not necessarily coincide (e.g. in the case of unsymmetrical or user defined sections). For example, the principal axes of an equal angle section are oriented at a 45 degree angle from its local axes.



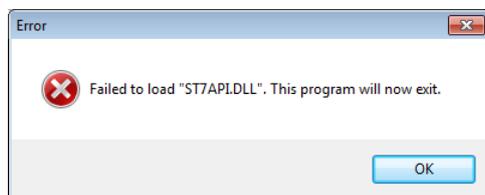
To help make the extraction of beam results more versatile, Strand7 reports beam forces and moments in both the local (x,y) and principal (1,2) axes.

For example, the highest bending moment at the root of the cantilever beam shown is reported in plane y if the load is applied in the Global Z direction.



API Troubleshooting

When an API application is launched, Windows searches for ST7API.DLL so that the entry points of the API functions can be resolved. When Windows is unable to locate the DLL, an error message such as the following is usually displayed:

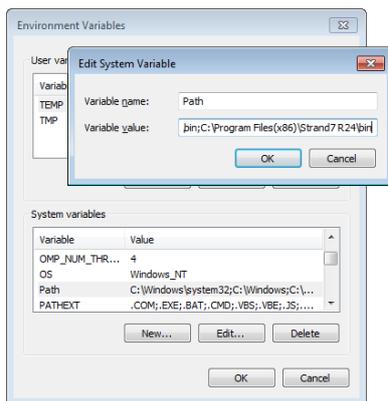


How can I make my program find ST7API.DLL?

Do not copy the DLL. This approach does not work as the DLL relies on other Strand7 installation files. By copying the DLL, the API is no longer able to locate the other files that it depends on.

The recommended approach is to update your Path environment variable so that the Strand7 installation is found regardless of where you run your program from. To update your Path environment variable, complete the following steps:

1. Click Start and select Control Panel. Double click System.
2. If you are using Windows Vista or Windows 7, click the Advanced system settings link. If you are using an older version of Windows, simply choose the Advanced tab.
3. Click Environment Variables.
4. Under System Variables, scroll down to Path and click Edit.
5. Identify the Strand7\bin directory path. For a local installation this might be C:\Program Files\Strand7 R24\bin. For a network installation this might be \\Server\Apps\Strand7 R24\bin or Z:\Apps\Strand7 R24\bin.
6. Modify the Path environment variable value string by adding the Strand7\bin directory path identified in the previous step. Please note that values in this string need to be separated by semi-colon (;) characters. Take care not to remove existing paths from this string.



7. Click OK and close the System Properties dialog box.
8. Restart the computer.

Still having problems?

If you still receive an error message relating to a missing ST7API.DLL after completing the above procedure then you can download a Strand7 API diagnostic tool from www.strand7.com/news.st7.htm

Training

The training calendar for the remainder of 2010 is still being finalised, but the following courses have so far been scheduled:

Date	Course Title	Location
11-13 October	Strand7 Essentials	Cambridgeshire, UK
14-October	Nonlinear Analysis with Strand7	Cambridgeshire, UK
15-October	Dynamic Analysis with Strand7	Cambridgeshire, UK

To register, please go to www.strand7.com and follow the links.

Abu Dhabi Training

We are also planning to run a Strand7 Introductory and Advanced Training course in Abu Dhabi, UAE, in late November or early December. This course will only go ahead if there is sufficient participation. If you are interested in attending, please contact us via info@strand7.com

Exhibitions

Between now and end of 2010, we will be exhibiting Strand7 at the following events:

18-20 Oct	The Infrastructure Show	NEC, UK
21-22 Oct	Civil Expo 2010	Seoul, Korea
3-5 Nov	SNAME	Seattle, USA

Links to these exhibitions are available on our website. Other exhibitions will be announced soon on the Strand7 website and in the next edition of News.St7

Head Office

Strand7 Pty Ltd

Suite 1, Level 5
65 York Street
Sydney NSW 2000
AUSTRALIA

Tel +61 2 9264 2977
Fax +61 2 9264 2066
Email info@strand7.com
Web www.strand7.com