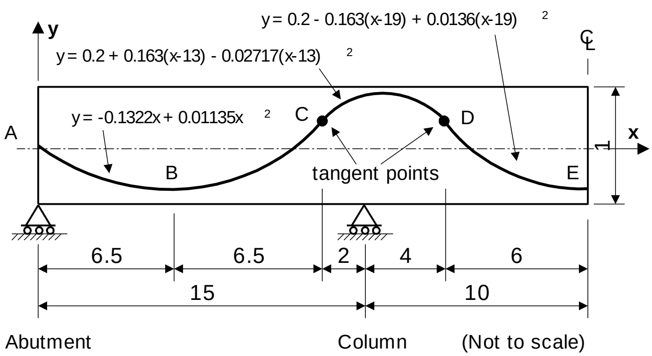
Linear Analysis of a Post Tensioned Bridge

For LUSAS version:	16.0
For software product(s):	LUSAS Civil & Structural or LUSAS Bridge
With product option(s):	None.

Description

A 3-span concrete post tensioned bridge is to be analysed using the single tendon prestress wizard. The bridge is idealised as a 1 metre deep beam with a tendon profile as shown in the adjacent image, which shows a half model of the bridge.



Half-model of bridge

The initial post tensioning force of 5000 kN is to be applied from both ends. Tendons with a cross section area of 3.5e3mm^2 are located every 2 metres across the section allowing the analysis to model a 2 metres effective width.

Three loadcases are to be considered; self-weight, short term losses, and long term losses.

Units of kN, m, t, s, C are used throughout the analysis. Note that tendon cross-sectional area is specified in mm^2.

Objectives

The required output from the analysis consists of:

- The maximum and minimum long and short term stress in the concrete due to post tensioning.

Keywords

2D, Inplane, Beam, Plane Stress, Single Tendon Prestress Wizard, Post tensioning, Beam Stress recovery.

Associated Files

- **post_ten_modelling.vbs** carries out the modelling of the example up until the point of section titled **Defining Prestress with Short Term Losses**. After running the script continue from that point in the example.
- **post_ten_profile.csv** carries out the definition of the tendon.

Modelling

Running LUSAS Modeller

For details of how to run LUSAS Modeller see the heading ***Running LUSAS Modeller*** in the *Introduction to LUSAS Worked Examples* document.

Note. This example is written assuming a new LUSAS Modeller session has been started. If continuing from an existing Modeller session select the menu command **File> New** to start a new model file. Modeller will prompt for any unsaved data and display the New Model dialog.

Creating a new model

- Enter the file name as **post_ten**
- Use the default User-defined working folder.
- Select Analysis type as **Structural**
- Select Analysis Category as **2D Inplane**
- Select model units of **kN,m,t,s,C**
- Ensure the timescale units are **Seconds**

- Select startup template **2D Plane Stress**
- No layout grid is to be used.
- Enter the title as **Post-tensioning of a bridge** and click the **OK** button.

Note. Save the model regularly as the example progresses. This allows a previously saved model to be re-loaded if a mistake is made that cannot be corrected easily.

Note also that the Undo button may also be used to correct a mistake. The undo button allows any number of actions since the last save to be undone

Feature Geometry

Geometry
Line >
By Coords...

Enter coordinates of (**0, 0**), (**15, 0**) and (**25, 0**) to define two Lines representing half the bridge. Use the **Tab** key to move to the next entry field on the dialog. Click the **OK** button

- Select all the visible Points and Lines using the **Ctrl** and **A** keys together.

Geometry
Group >
New Group...

To make the manipulation of the model easier create a group of the bridge

- Enter the group name as **Bridge**
- In the Treeview select the group **Bridge** with the right-hand mouse button and select the **Invisible** option to hide these features from the display.

Defining the tendon profile

When using the Prestress utility the tendon geometry is determined from a Line definition which, in practice, is usually a spline curve. The geometry of the tendon may be input into the model directly by manually entering the coordinates, or, as in this example, by copying values from a comma separated file (.csv) which has been opened in a spreadsheet application and pasting these values into the Point coordinates dialog in Modeller. This is done using standard Ctrl + C and Ctrl + V keys.

Note. To prevent the point representing the left-hand end of the bridge and the end point of the tendon profile from becoming stored as just a single point in the model the geometry should be made unmergable.

File
Model Properties…

- On the Geometry tab select the **New geometry unmergable** option and click **OK**

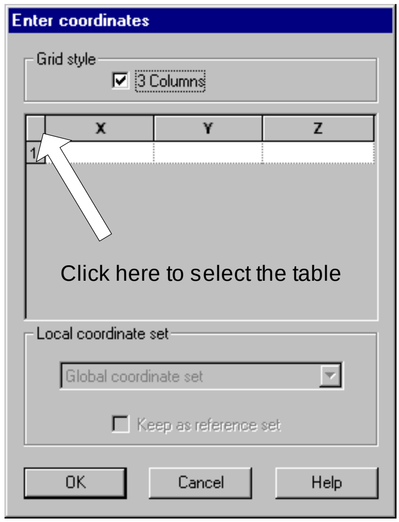
- Read the file **\<LUSAS Installation Folder>\Examples\Modeller\post_ten_profile.csv** into a spreadsheet.

- In the spreadsheet, select the top left-hand corner of the spreadsheet grid to select all the cells and press the **Ctrl + C** keys together to copy the data.

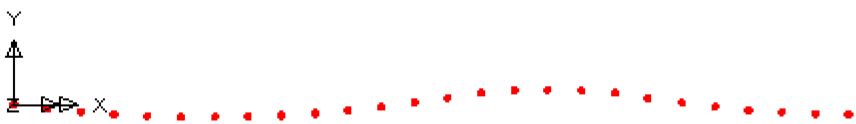
Geometry >
Point >
By Coords…

- Select the **3 Columns** option to show X, Y and Z columns.

- Select the top left-hand corner of the Enter Coordinates dialog and press the **Ctrl + V** keys together to paste the coordinates defining the tendon profile into the table. Note that the use of 'Paste' using a mouse button is not enabled. Click **OK**.



The points defining the tendon profile should appear as shown.



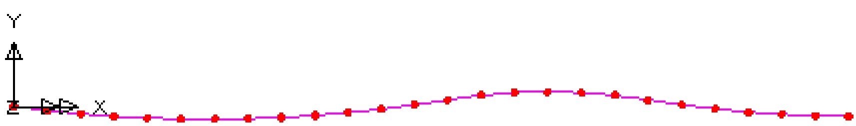
File
Model Properties…

On the Geometry tab de-select the **New geometry unmergable** option and click **OK**

- Use **Ctrl** and **A** keys together to select all the visible Points.

Geometry
Line >
Spline >
By Points…

- Click **OK** to define a spline that is defined in point selection order.



- Use **Ctrl** and **A** keys together to select the spline line and points.

Geometry
Group >
New Group

Make the model definition easier by placing the prestress tendon into a group.

- Enter the group name as **Tendon** and click **OK**

Note. As well as defining the tendon manually using coordinates the Prestress Wizard (accessed using **Bridge** (or Civil) > **Prestress Wizard** > **Tendon Profile**) allows you to define the profile of the tendon in 2D or 3D space as a series of straight lines, arcs, splines and parabola. The Tendon Profile dialog is used in the worked example 'Segmental Construction of a Post Tensioned Bridge'.

Defining the Mesh

With the prestress analysis type to be used in this example (the beam analysis method) the tendon profile does not need to be meshed.

- In the Treeview select **Bridge** and pick the **Set As Only Visible** option. This turns off the display of the tendon to make bridge assignments easier.

The concrete bridge is to be represented by 2D beam elements. A suitable 2D thick beam element mesh is already provided in the Attributes Treeview.

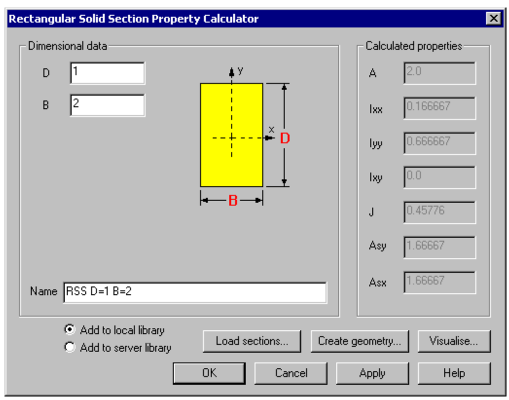
- With the lines representing the bridge selected, (press **Ctrl** and **A** keys together) drag and drop the mesh dataset **Thick beam, linear order, div = 4** from the Treeview onto the selected features.

Defining the Geometric Properties

The beam idealisation represents a two metre width of the bridge which is one metre deep. A section of this size will be created and added to the local library for subsequent assigning to the model.

Tools
Section Property Calculator >
Rectangular Sections > Solid

- Select the **Rectangular Solid Section** (RSS)
- Enter a depth (D) of **1** and a breadth (B) of **2** and click **OK** to add the section to the local section library with the default name given.

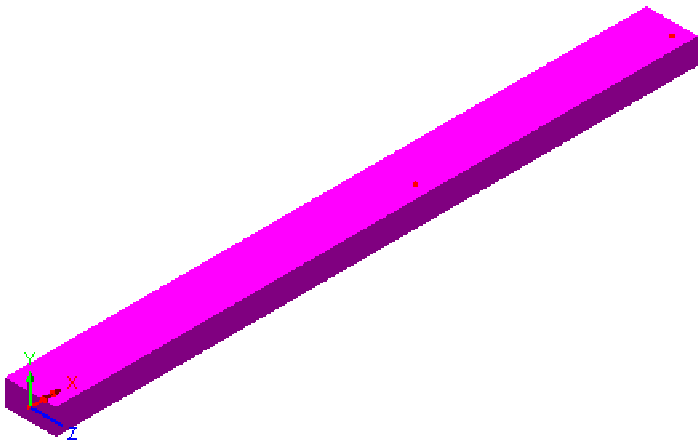


To use this section it must be added to the Treeview:

Attributes
Geometric >
Section Library..

- Select **User Sections** from the top-most drop-down list, select **Local** library, then select **RSS D=1 B=2**.
- Enter the attribute name as **Beam Properties** and click **OK**
- With the lines representing the bridge selected, drag and drop the geometric dataset **Beam Properties (RSS D=1 B=2 major z)** from the Treeview onto the selected features.

Select the Isometric button to view the visualised beam.



Select the fleshing on/off button to turn-off the geometric visualisation.

Select the Home button to return the model to the default view.

Defining the Material

A suitable concrete material is already provided in the Attributes Treeview.

- With the lines representing the bridge selected, drag and drop the material dataset **Concrete (Ungraded | Concrete)** from the Treeview onto the selected features and assign to the selected Lines by clicking the **OK** button.

Defining the Supports

Since the bridge is symmetrical about the centre, the analysis considers only half of the structure. Two support types are required: one that models the symmetry and prevents longitudinal displacement and vertical rotation, and one that provides simple support in the Y direction. Both support types are provided in the Treeview.

- With the point at the right-hand side of the model selected drag and drop the support dataset **Symmetry parallel to Y (Fixed X and Thz)** from the Treeview onto the selected feature. Choose options to **Assign to points** for **All analysis loadcases** and click **OK**
- Then select the point at the left-hand abutment and the point at the column. (Hold the **Shift** key to add to the initial selection) and drag and drop the support dataset **Fixed in Y** onto the selected feature. Choose options to **Assign to points** for **All analysis loadcases** and click **OK**

Loading

Three loadcases are to be applied. The first represents self-weight of the structure. The second represents the prestress with short term losses only. The third represents the prestress with short and long term losses.

Defining the Self-weight

Gravity can be applied to any loadcase from the Treeview.

- Select **Loadcase 1** from the Treeview with the right-hand mouse button and select **Gravity**.

Note. When gravity is applied to a loadcase as an option, load arrows representing self-weight are not shown in the View window.

- Select **Loadcase 1** from the Treeview with the right-hand mouse button and pick the **Rename** option and change the loadcase name to **Self Weight**

Turn-on the display of the tendon

Now that the bridge attribute assignments have been made the tendon can be re-displayed.

- In the Treeview select **Tendon** with the right-hand mouse button and select the **Visible** option.

If rebuilding a model, or creating a model using the supplied file.

If a previous analysis of this example has failed you need to return to this point to continue after having run the supplied file stated.

If you created the model using the supplied associated file you need to continue from this point to complete the modelling required.

Modelling Prestress in LUSAS

The single and multiple tendon prestress wizards in LUSAS calculate equivalent nodal loading due to post tensioning and assign these forces automatically (and using search areas) to selected lines (and hence nodes and elements) of the model for the current active loadcase.

The Single tendon prestress wizard (as used in this example) does not take into account any stressing or unstressing of any other tendons. It is for use with beam, tendon, plane stress and solid element modelling of concrete

A separate example 'Segmental construction of a post-tenstioned bridge' shows the use of the multiple tendon prestress wizard

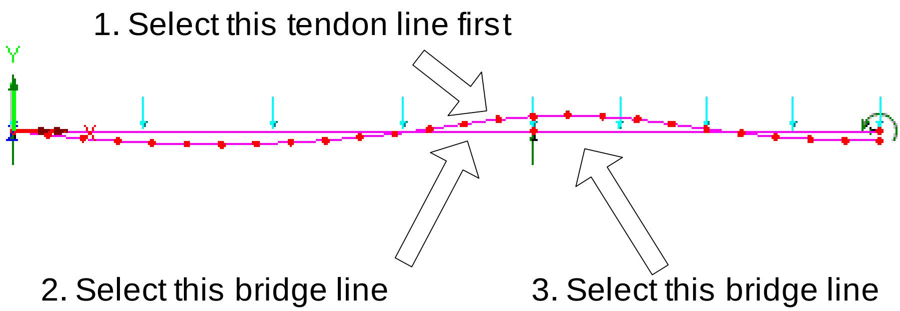
Defining Prestress with Short Term Losses

Analyses
Loadcase…

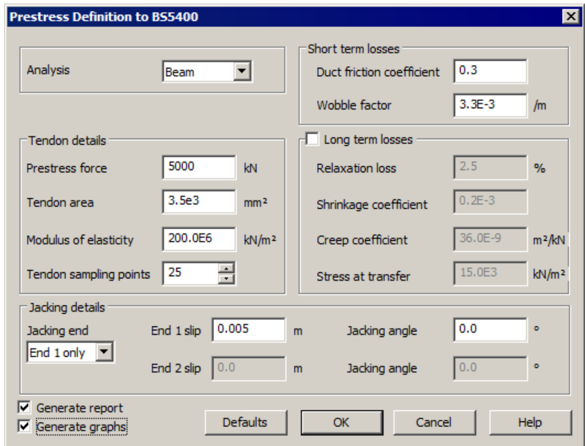
- Ensure that the ‘Automatically add gravity to this loadcase’ option is <u>not</u> selected.
- Change the loadcase name to **Short Term Loss** and click **OK**
- Select **Short Term Loss** from the Treeview with the right-hand mouse button and pick the **Set Active** option.

Caution. Prior to running the Single Tendon Prestress Wizard to calculate short term and long term tendon losses the correct Short Term or Long Term loadcase must be set active to ensure that the values are written into the correct Tendon Load assignment attributes as held in the Treeview. In addition, prior to running the Prestress Wizard, the <u>spline line representing the tendon</u>, then <u>any lines representing the concrete bridge</u>, must be selected <u>in that order</u>. So, with the **Short Term Loss** loadcase set active:

- In the model view select the spline line representing the tendon
- Select the 2 beams representing the concrete bridge. (Hold the **Shift** key to add the other 2 lines to the initial selection)



Bridge
Prestress Wizard >
Single Tendon >
BS5400-4:1990



- Select the **Defaults** button.
- Ensure the Analysis type is set to **Beam**
- In the Tendon details section enter the Prestress force as **5000**
- Enter the Tendon area as **3.5E3** (note that this is entered in mm2)
- Leave the Modulus of the elasticity of the tendon as **200E6**
- Set the number of Tendon sampling points to **25**
- In the Short term losses section set the duct friction coefficient to **0.3**
- Ensure the **Long term losses** option is deselected.
- In the Jacking details section ensure the jacking is defined for **End 1 only** with an End 1 (anchorage) slippage of **0.005**
- Click the **Generate report** option. When the wizard is run this will generate a report in HTML format in the **working** directory summarising the prestress tendon forces.

- Click the **Generate graphs** option. When the wizard is run this will generate graph datasets in the Treeview for subsequent graphing of tendon losses to be carried out.

- Click **OK**

Caution. Subsequent modification of the current tendon profile or beam lines will not update the calculated loading until the prestress wizard is run again with the same analysis loadcase set active.

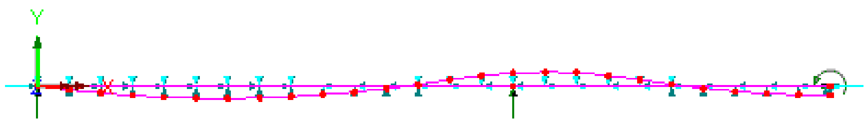
- Click **OK** to dismiss the warning message displayed.

Equivalent nodal loading is then calculated from the values entered on the Prestress Definition dialog.

On completion, an HTML report will be opened automatically in the default web browser. This report is also saved into the working directory. The report is a complete summary of the tendon profile, properties and loading assignments made. On closing the report the tendon loading assignments on the model can be seen for the active loadcase.

Modeller will assign the datasets to the selected Lines representing the concrete beams for the current loadcase and will also add two load datasets to the Treeview.

- In the Treeview double-click the **Utilities** entry and on the Tendon profile tab click the **None** button to stop the display of the tendon profile setting-out details.



Note. Tendon sampling points determine the number of points used when graphing and creating reports on short and long term losses.

Defining Prestress with Short and Long Term Losses

Analysis
Loadcase…

- Change the loadcase name to **Long Term Loss** and click **OK**

- Select **Long Term Loss** from the Treeview with the right-hand mouse button and pick the **Set Active** option.

Caution. Forgetting to set the correct loadcase active prior to running the Prestress Wizard will write values to an incorrect loadcase.

- Re-select the tendon profile and the two lines representing the bridge, making sure that the tendon profile is selected first.

With the Long Term Loss loadcase active:

Bridge
Prestress Wizard >
Single Tendon >
BS5400-4:1990

The values previously entered on the dialog will be retained.

- Select the **Long term losses** option. Leave all other values as they were set for the short term loss loadcase.
- Click the **Generate report** and the **Generate graphs** options. This generates a report in HTML format and graph datasets in the Treeview for subsequent graphing to be carried out.
- Click **OK** to apply the prestress force to the current loadcase.

Equivalent nodal loading is calculated from the values entered on the Prestress Definition dialog.

- Click **OK** to dismiss the warning message about subsequent modification of the current tendon profile or beam lines not updating the calculated loading until the prestress wizard is run again.

Graphing of Prestress Force in the Tendon

When the Generate graphs option is selected on the Prestress dialog two graph datasets are created. One contains the distance along the beam and the other contains the force in the tendon at each point. To graph these:

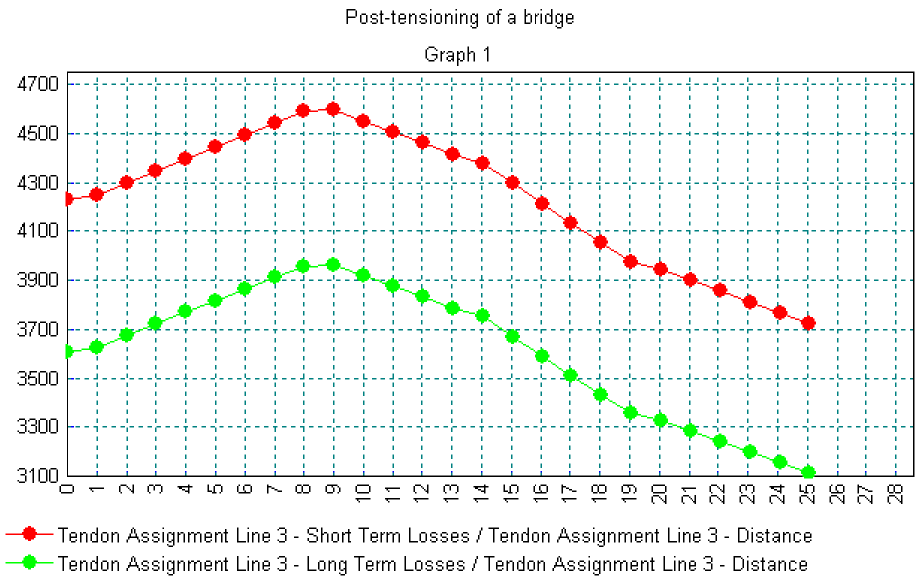
Utilities
Graph Wizard…

- Select the **Specified datasets** option and click **Next**
- From the first drop down list select **Tendon Assignment Line 3 Distance** and from the second select **Tendon Assignment Line 3 - Short Term Losses** and click **Next**
- Click **Finish** to display the graph

Now add a graph of the Force after long term losses:

Utilities
Graph Wizard…

- Select the **Specified datasets** option and click **Next**
- From the first drop down list select **Tendon Assignment Line 3 Distance** and from the second select **Tendon Assignment Line 3 - Long Term Losses** and click **Next**
- In the Include existing graphs panel ensure that **Graph 1** is selected, then click **Finish** to create a graph with both curves.



 Close the graph window.

Saving the Model

File
Save

 Save the model file.

Running the Analysis

 Press the **Solve Now** button to run the analysis.

A LUSAS Datafile will be created from the model information. The LUSAS Solver uses this datafile to perform the analysis.

If the analysis is successful...

LUSAS loadcase results will be added to  Treeview.

In addition, 2 files will be created in the Additional Model Data folder, where the model file resides:

- **post_ten.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.

❑ **post_ten.mys** this is the LUSAS results file which is loaded automatically into the Treeview to allow results processing to take place.

If the analysis fails...

If the analysis fails, information relating to the nature of the error encountered can be written to an output file in addition to the text output window. Any errors listed in the text output window should be corrected in LUSAS Modeller before saving the model and re-running the analysis.

Rebuilding a Model

If it proves impossible for you to correct the errors reported a file is provided to enable you to re-create the model from scratch and run an analysis successfully.

❑ **post_ten_modelling.vbs** carries out the modelling of the example up to the point of defining the prestress.

File
New…

Start a new model file. If an existing model is open Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **post_ten**

File
Script >
Run Script...

- To recreate the model up until the point of defining the prestress, select the file **post_ten_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Now return to the section entitled **Defining Prestess With Short Term Losses** earlier in this example and re-define the tendon properties.

Viewing the Results

Selecting the Results to be Viewed

The results will be loaded on top of the current model and the loadcase results for the last solved loadcase (Long term loss) will be set to be active by default.

- In the Treeview right-click on **Self Weight** and select the **Set Active** option.

Deformed Mesh Plots

A deformed mesh plot helps highlight any obvious errors with an analysis before progressing to detailed results processing. The deformed shape will usually show up errors in loading or supports and may also indicate incorrect material property assignments (e.g. where the results show excessive displacements).

Deformed Mesh Plot for Self Weight

- If present, turn off the display of the **Geometry, Attributes** and **Mesh** layers from the Treeview.

The **Deformed mesh** layer should be already present in the Treeview, and the View window will be showing the deformed mesh for the self-weight alone.

- With no model features selected, click the right-hand mouse button in the graphics window and select **Values**
- Select entity **Displacement** and component **DY**
- Select the **Values Display** tab.
- Select the **Show values of selection** option.
- Change the number of significant figures to **4** and click **OK**
- Select the node on the centre line to add the value of vertical displacement at mid-span to the display.

Deformed Mesh Plot for Short Term Loss

- In the Treeview right-click on **Short Term Loss** and select the **Set Active** option.

Defining a Basic Combination

Combinations can be created to view the combined effects of the self weight and short term loss loadcases on the structure.

Analyses
Basic
Combination…

The combination properties dialog will appear.

- Select the **Self Weight** and **Short Term Loss** loadcases from those available and click the >> button to add them to the load combination. Change the combination name to **Self Weight and Short Term** and click the **OK** button to finish.

Deformed Mesh Plot for Combination

- In the Treeview right-click on **Self Weight and Short Term** and select the **Set Active** option.

The deformed mesh plot will show the effect of the combined loading on the structure.

Fibre Locations

Force/moment and stress diagrams can be plotted for pre-defined fibre locations on members. All sections created by the Section Property Calculator will have extreme fibre locations pre-defined. These fibre locations can be seen by expanding the Geometric Line entry **Beam properties (RSS D=1 B=2 major y)** in the Treeview, and they can be visualised by double-clicking the Geometric Line attribute name and then selecting the **Visualise** button on the dialog. In doing so, it can be seen that, by default, Fibre S1 is the active fibre and Fibre S1 and Fibre S4 are the upper and lower extreme fibres for which diagram and stress results plots should be created.

Diagram Stress Results at Fibre Locations

- Delete the **Values** layer from the Treeview.

- In the Treeview right-click on **Short Term Loss** and select the **Set Active** option.

- With no features selected right-click in a blank part of the graphics window and select **Diagrams** to add the diagrams layer to the Treeview.

The diagram properties will be displayed.

- Select **Stress - Thick 2D Beam** results of stress **Sx(Fx, Mz)** in the beam.

- Select the **Diagram Display** tab and ensure that **Peaks only** is selected.

- Change the angle of the text to **45**, set the number of significant figures to **4** and click **OK** to display the stress diagram at Fibre location S1 for the Short Term Loss loadcase

Diagram Results for Short Term Loss

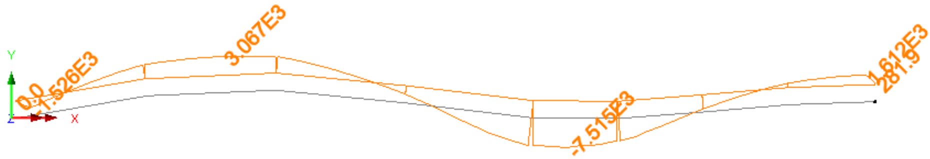


Diagram Results for Combination

- In the Treeview right-click on the combination **Self Weight and Short Term** and select the **Set Active** option to display the stress diagram at Fibre location S1 for the Self Weight and Short Term combination

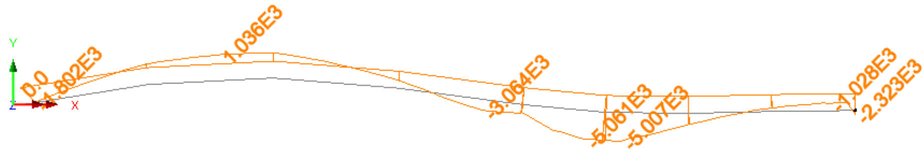


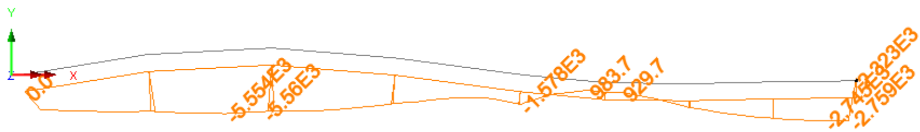
Diagram Results for Combination at Fibre 4

Once a loadcase has been set results can be plot for any defined fibre location. In this example, because of their location and because of the applied loading, Fibre S1 and Fibre S2 will give the same results at the top of the beam. Fibre S3 and Fibre S4 will both also give the same results at the bottom of the beam – so only a diagram for Fibre S4 needs to be viewed.

With the combination **Self Weight and Short Term** still set active in the Treeview:

- In the Treeview expand Geometric Line entry **Beam Properties (RSS D=1 B=2 major z)** and right-click on **Fibre S4** and select **Set Fibre Active**.

The diagram display will update to show the stresses at Fibre S4:



Viewing Results on Fleshed Members

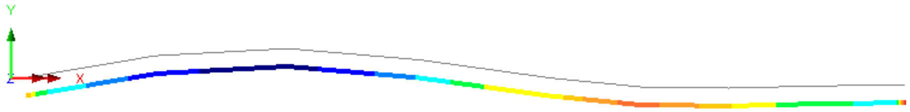
As an alternative to plotting diagram results, stresses in beams can also be plot on fleshed members.

- Turn off the display of the **Diagrams** layer from the Treeview.

Contour Results for Short Term Loss

- In the Treeview right-click on **Short Term Loss** and select the **Set Active** option.
- Click in a blank part of the graphics window to deselect the members.
- With no features selected right-click in a blank part of the graphics window and select **Contours** to add the contours layer to the Treeview.
- Select **Stress – Thick 2D Beam** results of axial force and moment results in the members **Sx(Fx, Mz)** and click **OK**

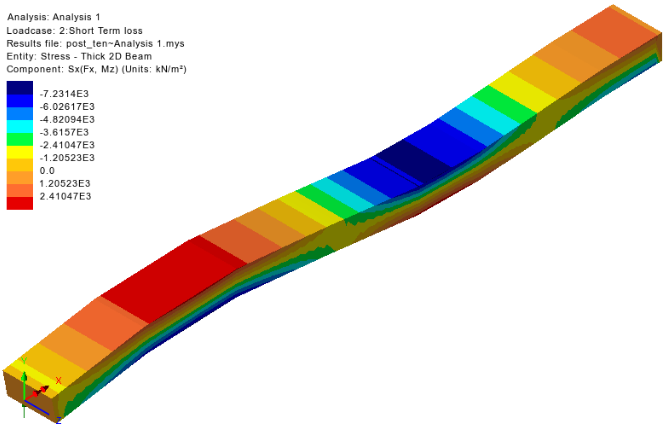
Initially the contour properties will be displayed for the fibre location (Fibre S4) that is currently set active.



To see the stresses on the fleshed beam member:

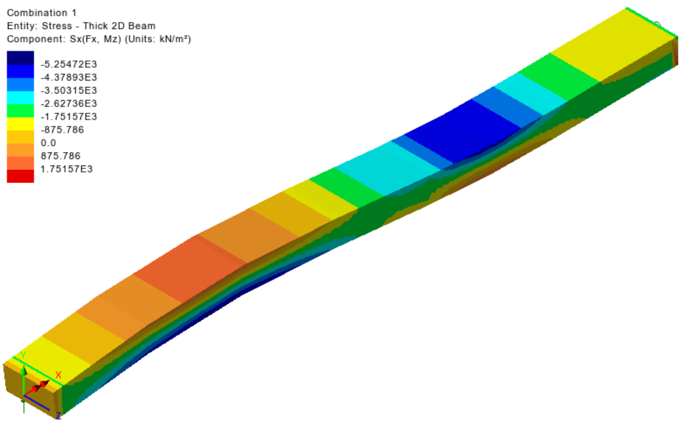
Select the isometric button.

Select the fleshing on/off button to visualise the stress results for the fully fleshed cross-section.



Contour Results for Combination

- In the Treeview right-click on the combination **Self Weight And Short Term** and select the **Set Active** option.



Note. Diagrams and contours may be displayed at the same time to show results for selected fibre locations.

This completes the example.

Discussion

This example illustrates the use of the single tendon prestress wizard, which does not take into account any stressing or unstressing of any other tendons, should they also be present in the model, which in this example they are not.

The **Multiple tendon prestress wizard** should generally be used instead of the single tendon wizard since it takes into account elastic shortening due to stressing of other tendons according to the selected design code or user-defined percentage losses, and is more suited to staged construction analysis.