# Linear Analysis of a Post Tensioned Bridge

For LUSAS version:	19.0
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 prestress modelling facilities. 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
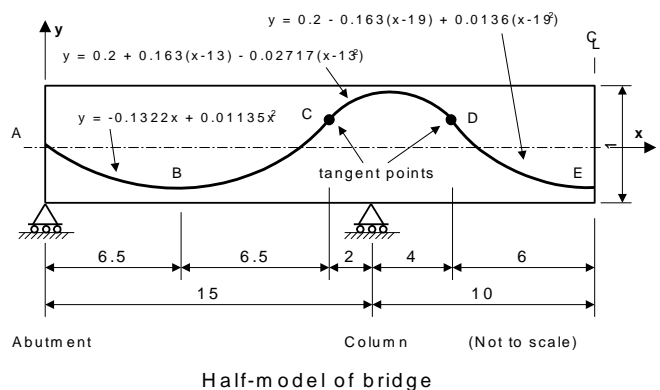
The initial post tensioning force of 5000 kN is to be applied from both ends. Tendons with a cross section area of 3.5e3mm^2 are located every 2 metres across the section allowing the analysis to model a 2 metres effective width.

Three loadcases are to be considered; self-weight, short term (instantaneous) 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.

The design code used in this example is **BS5400-4:1990.** This code restricts the example to using the 'approximate losses' method.

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, Prestress Loading, Tendon Profile, Tendon Properties, Tendon Loading, Post Tensioning, Approximate Losses, Short term, Instantaneous, Long term, 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 (Instantaneous) 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 the Analysis type to be **Structural**

- Select Analysis category to be **2D Inplane**

- Select model units of **kN,m,t,s,C**

- Ensure the timescale units are set as **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
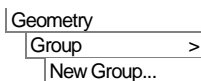
Feature Geometry

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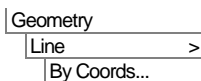
Enter coordinates of (**0, 0**), (**15, 0**) and (**25, 0**) to define two Lines representing half the bridge. Use the **Tab** key to move to the next entry field on the dialog. Click the **OK** button

- 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

Tendon profile data can be defined manually on a dedicated dialog, be copied and pasted from a spreadsheet, or be generated by selecting lines, arcs or splines that are defined or imported into LUSAS Modeller.

In this example, the tendon profile is defined by copying values from a supplied 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.
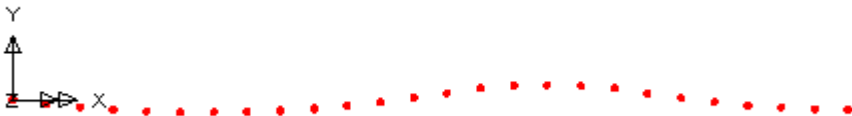
- On the Geometry tab select the **New geometry unmergable** option and click **OK**

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

- 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**.
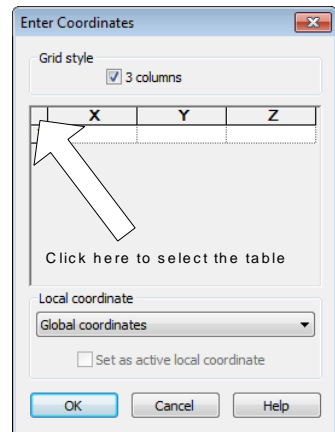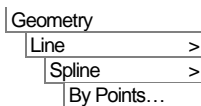
The points defining the tendon profile should appear as shown.

File
Model Properties…

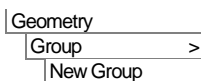On the Geometry tab now de-select the **New geometry unmergable** option and click **OK**

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

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

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

Defining the Mesh

- 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.

The spline line defining the tendon profile does not need to be meshed. It is merely used to define the tendon profile.

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**, a breadth (B) of **2,** and click **OK** to add the section to the local section library with the default name given.
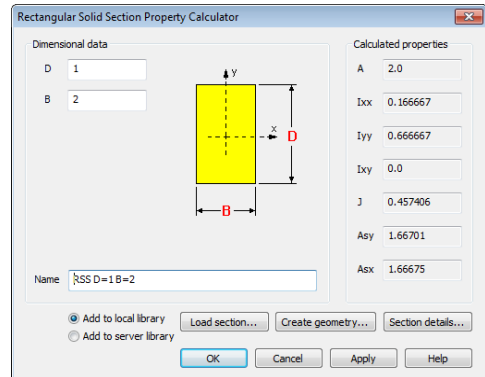
To use this section it must be added to the ⚛ Treeview:

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

⚡ Turn-on / off fleshing as necessary.
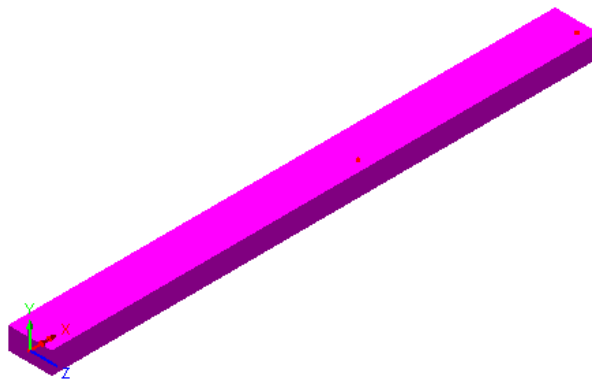
🏠 Select the Home button to return the model to the default view.

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.

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. Ensure **Assign to points** for **All analysis loadcases** are selected 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. Ensure **Assign to points** for **All analysis loadcases** are selected 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 (instantaneous) 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.

Defining Prestress with Short Term (Instantaneous) Losses

<div style="border: 1px solid;">Analyses
   Loadcase…</div>

- 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.

Defining the Tendon Profile

- In the model view select the spline line representing the tendon
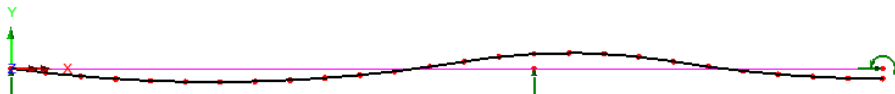
The tendon profile dialog grid will be populated with the coordinates of the drawn tendon.

- Click the **Advanced** button.

- Set the 'Minimum total number' of sampling points to be **20** and ensure the 'Minimum per profile' segment is set to **1** and click the **OK** button.

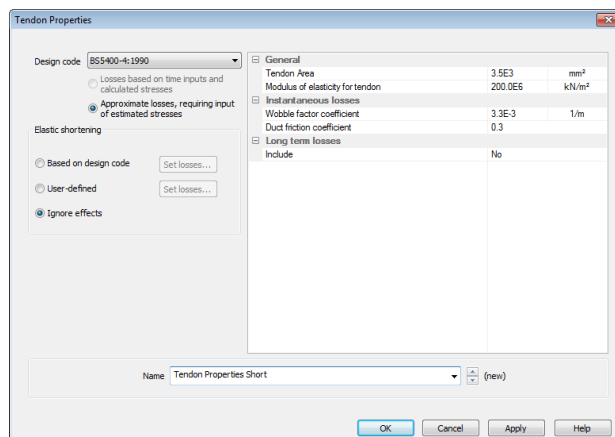- Change the name to **Tendon Profile** and click the OK button.
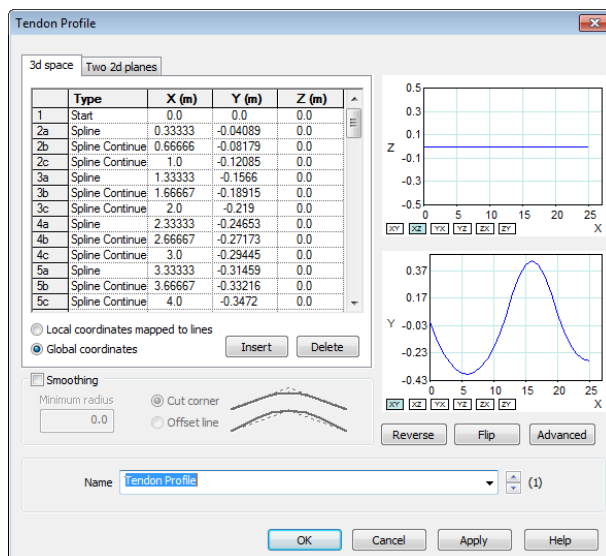
Note. Tendon sampling points are the locations along a tendon profile at which equivalent tendon loads are calculated and applied to the model. Tendon sampling points are also the locations used when graphing and creating reports on short and long term losses.

Defining the Tendon Properties

- Choose the **BS5400-4:1990** design code entry from the drop down list.

- In the Elastic shortening panel, select the **Ignore effects** option.

- 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.**

- In the Instantaneous losses section, set the duct friction coefficient to **0.3.**

- In the Long term losses section, change Include to **No**.

- Change the name to **Tendon Properties Short** and click **OK.**

Defining the Tendon Loading

Attributes
   Loading…

- Select the **Tendon** option and click **Next**

- Enter a Prestress force of **5000**

- Ensure **Jacking at end 1** is selected and enter a value for anchorage slip of **0.005**

- Enter the attribute name as **Tendon Short**

- Click the **Finish** button.


Caution. Subsequent modification of the current tendon profile will not update the calculated loading until the model has been solved.

Assigning the Tendon Loading

- Select the two straight lines representing the bridge deck that the loading **Tendon Short** will act on.

- Drag and drop **Tendon Short** from the ⚙ Treeview onto the selected lines.

- Ensure the **Single loadcase** option is chosen and select **Short Term Loss** in the loadcase combo box. Click **OK**.
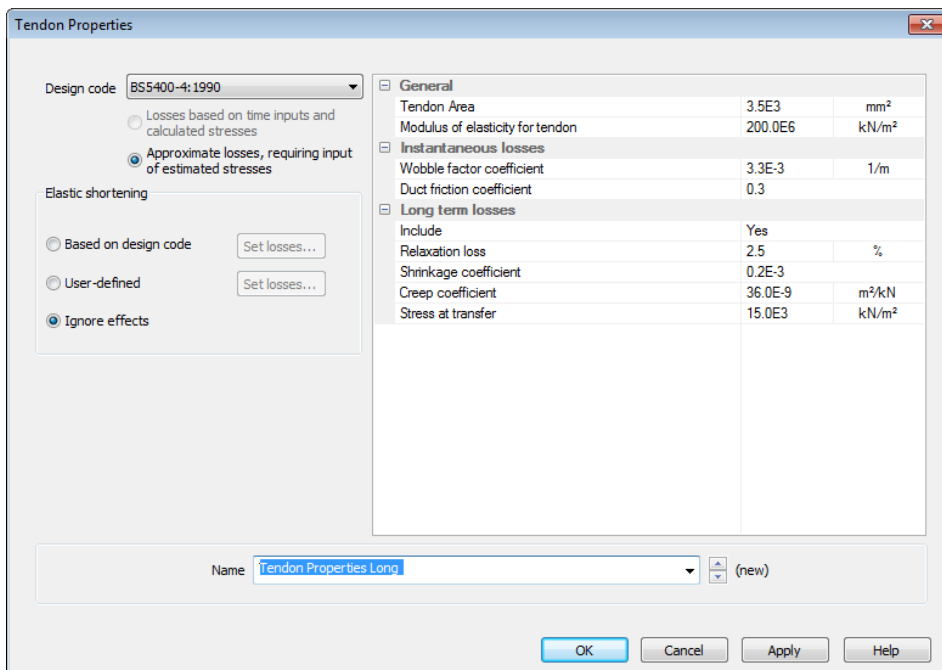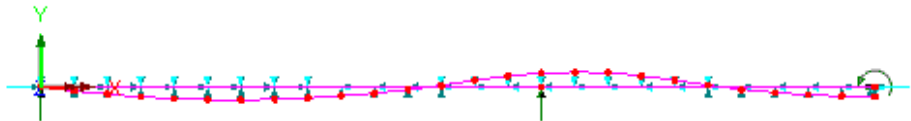

Defining Prestress with Short (Instantaneous) and Long Term Losses

Analysis
   Loadcase…

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

- In the ⚒ Treeview, double click on the **Tendon Properties Short**.

- In the Long term losses section change 'Include' to **Yes**. Accept the default values.

- Change the name to **Tendon Properties Long** and click **OK**.

Tendon Properties			
Design code	BS5400-4:1990		
⊙ Losses based on time inputs and calculated stresses			
⦿ Approximate losses, requiring input of estimated stresses			
Elastic shortening			
⊙ Based on design code	Set losses...		
⊙ User-defined	Set losses...		
⦿ Ignore effects			

⊟ General		
Tendon Area	3.5E3	mm²
Modulus of elasticity for tendon	200.0E6	kN/m²
⊟ Instantaneous losses		
Wobble factor coefficient	3.3E-3	1/m
Duct friction coefficient	0.3	
⊟ Long term losses		
Include	Yes	
Relaxation loss	2.5	%
Shrinkage coefficient	0.2E-3	
Creep coefficient	36.0E-9	m²/kN
Stress at transfer	15.0E3	kN/m²

Name Tendon Properties Long ▾ ▲▼ (new)

[OK] [Cancel] [Apply] [Help]

Defining the Tendon Loading

Attributes
   Loading…

- Select the **Tendon** option and click **Next**

- Ensure that the **Tendon Profile** and **Tendon Properties Long** options are selected

- Enter the Prestress force as **5000**

- Ensure **Jacking at end 1** is selected and enter a value for anchorage slip of **0.005**

- Enter the attribute name as **Tendon Long** and click the **Finish** button

Assigning the Tendon Loading

- Select the two straight lines representing the bridge deck that the loading **Tendon Long** will act on.

- Drag and drop the loading attribute **Tendon Long** from the 🔵 Treeview onto the selected lines.

- Select the **Single loadcase** option and select **Long Term Loss** in the Loadcase combo box. Click **OK**.

Note. Forces calculated in prestress tendons will differ if prestress loading is assigned individually to several adjacent loadcases, compared with it being assigned to a range of loadcases. This is simply a function of the implementation of this facility. Tendon loads for multiple loadcases should be assigned to a range of loadcases (e.g. Loadcase 1 to Loadcase 5) and not individually to those five loadcases. A warning will be issued if the same tendon load is individually assigned to the same features in two adjacent loadcases.

Saving the Model

File
Save

🖫 Save the model file.

Running the Analysis

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

A LUSAS Datafile and a PrestressResuls 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.
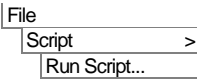
❑ **post_ten_modelling.vbs** carries out the modelling of the example up to the point of defining the prestress.
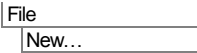
File
New…

⬜ 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.
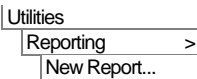
- Enter the file name as **post_ten**

File
Script   >
    Run Script...

- To recreate the modelup 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

After the Solve, 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.
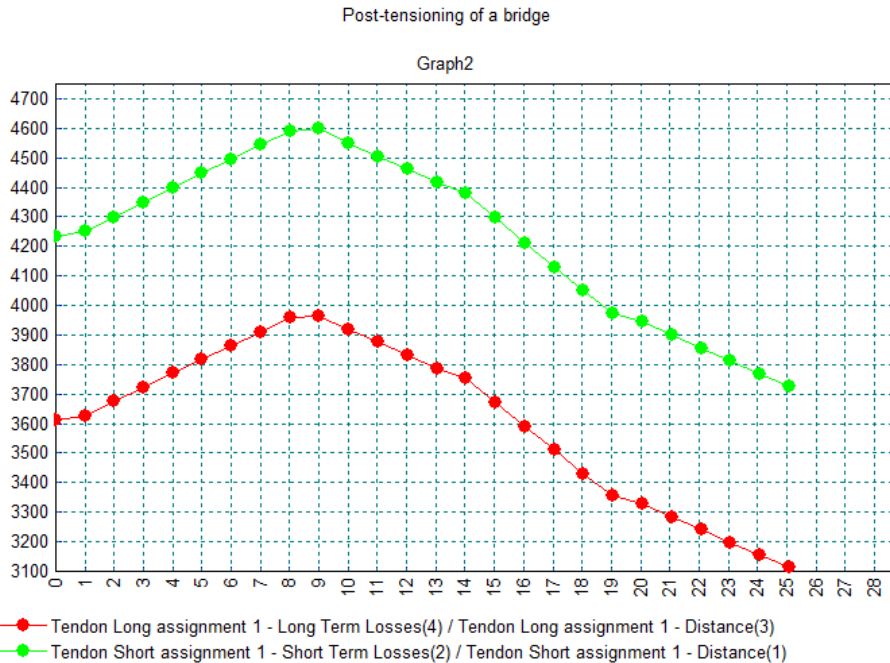
Generating a Report showing Prestress Details

Utilities
Reporting   >
    New Report...

- Enter the Title as **Post Ten**, change the name to **Prestress Report** and click **OK**.

- In the ▤ Treeview, right-click on **Prestress Report** and choose **Add Chapter**.

- On the Chapter dialog, go to the **Prestress** tab and tick the **Prestress** check box and press **OK**.

- In the ▤ Treeview, right-click on **Prestress Report** and choose **View Report**.

A PDF report is created giving details of the tendon profile, properties and loading assignments.

Graphing of Prestress Force in the Tendon

- Select the **Prestress** option and click **Next**

- Ensure the check boxes for **Tendon Short assignment 1** and **Tendon Long assignment 1** are selected click **Next**

- Click **Finish** to display the graph

Post-tensioning of a bridge

- Tendon Long assignment 1 - Long Term Losses(4) / Tendon Long assignment 1 - Distance(3)
- Tendon Short assignment 1 - Short Term Losses(2) / Tendon Short assignment 1 - Distance(1)

Close the graph window.

Selecting the Results to be Viewed

- 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)** 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.
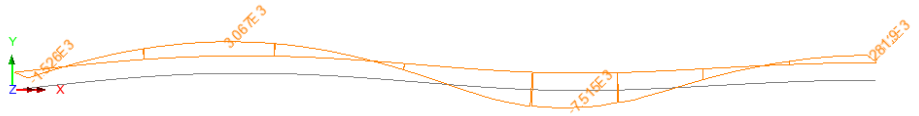
Diagram Stress Results at Fibre Locations

- 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** 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
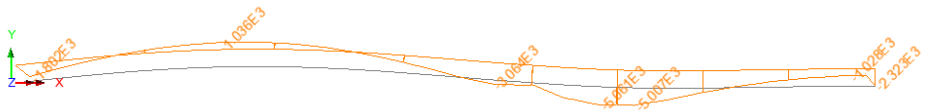
Diagram Results for Short Term Loss

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 plotted 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)** and right-click on **Fibre S4** and select **Set Fibre Active**.

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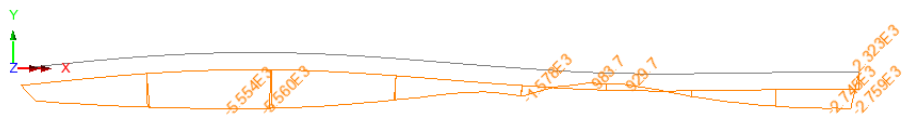

Viewing Results on Fleshed Members

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** 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.
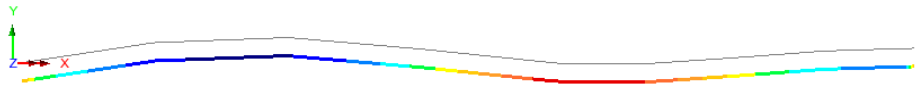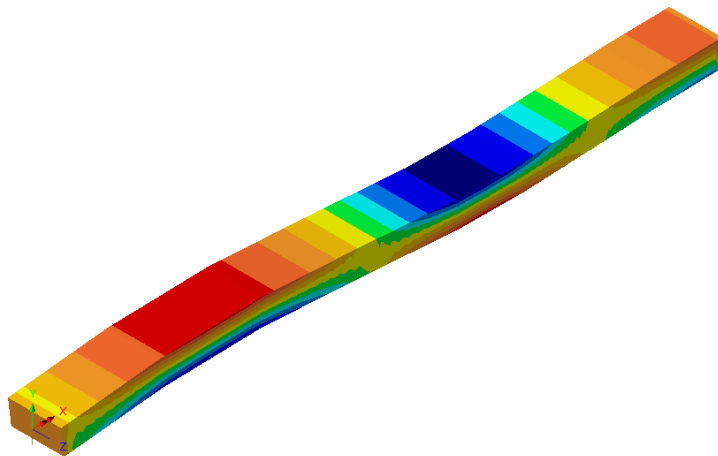

To see the stresses on the fleshed beam member:
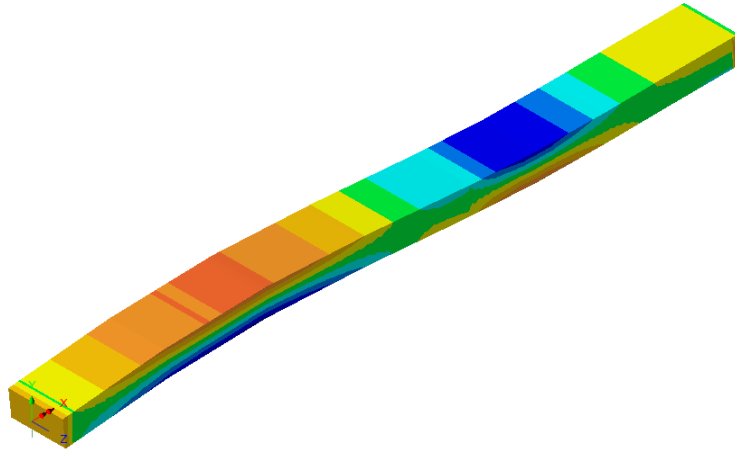
🔘 Select the isometric button.

⚡ 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 demonstrates the use of the approximate losses method for design code is **BS5400-4:1990**, where an estimate of the concrete stress at transfer is required. Other supported codes could be used.

The worked example 'Balanced Cantilever Construction of a Segmental Post-Tensioned Bridge' illustrates the use of losses based on time inputs and calculated stresses. This method can be used with **EN1992-1-1:2014 / 2014 Eurocode 2** and **AASHTO LRFD 5th to 7th Editions**.