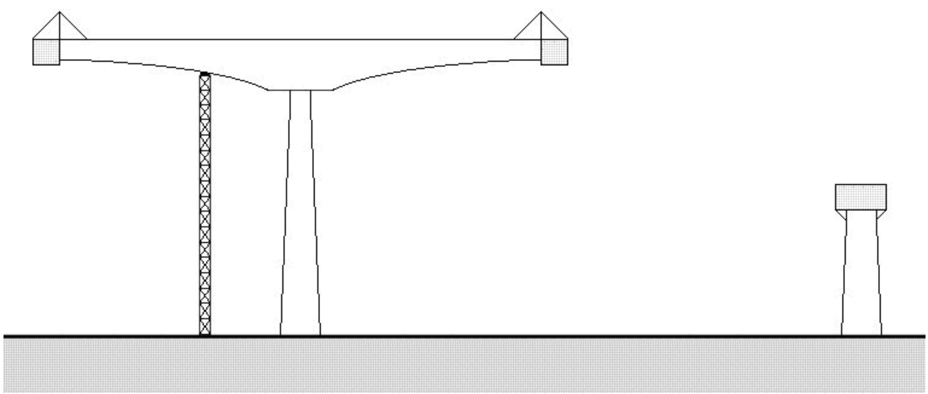
Segmental Construction of a Post Tensioned Bridge

For LUSAS version:	17.0
For software product(s):	LUSAS *Civil & Structural Plus* and LUSAS *Bridge Plus*
With product option(s):	Nonlinear

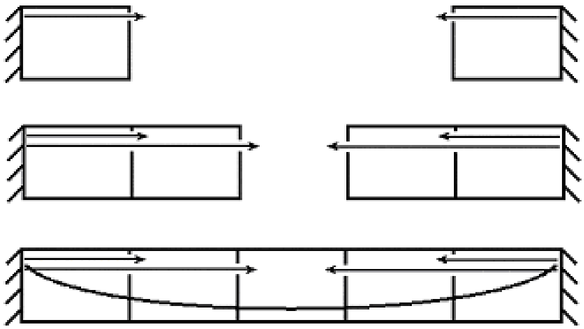
Description

The staged construction of a balanced cantilever segmentally constructed bridge is to be modelled using a beam analysis.



A complete analysis of a balanced cantilever segmentally constructed bridge is a complex and large analysis to undertake. In this example the geometry has been simplified to concentrate on the definition of the staged construction process and to illustrate the definition and assignment of tendon profiles, properties and loading.

An internal span of the bridge is modelled with segments being placed from adjacent piers, with a final closing segment to join the two constructions together as shown on the next diagram.



Each stage of the construction analysis considers a 6m long section being added to the construction. These are: Stage 1 (top-left), Stage 2 (top-right), Stage 3 and Stage 4 (middle) and Stage 5 (bottom).

Tendon

Units used are kN, m, t, s, C throughout.

The example incorporates staged construction and the assignment of prestress loading.

Objectives

The output requirements of the analysis are:

- Maximum moments during construction

Keywords

3D, Beam, Staged Construction, Age, Time Management, Prestress Loading, Tendon Profile, Tendon Properties, Tendon Loading, Post Tensioned.

Associated Files

- **segmental_bridge_modelling.vbs** carries out the modelling of the structure up to the section titled **Prestress Loading** where tendon properties are defined. Modelling. After running the script continue from that point in the example.

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 **Segmental Bridge**
- Enter a file path for the working folder.
- Select **Analysis type** as **Structural**
- Select **Analysis Category** as **3D**
- Select model units of **kN, m, t, s, C**
- Ensure the timescale units are **Days**
- Select the Startup template **None** from those available in the drop-down list.
- Leave **Layout grid** as **None**
- Enter the title as **Segmental bridge including prestress and creep**
- Click the **OK** button.

Defining the Geometry

Geometry >
Line >
By Coords…

Enter coordinates of (**0, 0, 0**), and (**6, 0, 0**), to define the first segment of the bridge and click **OK**

- Select the line just drawn.

Geometry
Line >
Copy…

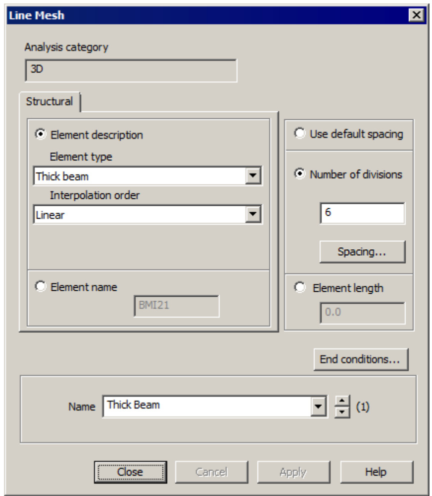
Enter a translation in the **X** direction of **6**

  - Enter the number of copies required as **4** and click **OK**

Defining and Assigning Mesh Attributes

Attributes
Mesh >
Line…

- The bridge is to be modelled with **Thick beam** elements (BMI21 elements)
- Ensure that **Linear** interpolation order is selected.
- Enter **6** for the number of mesh divisions.
- Enter the attribute name as **Thick Beam** and click **OK**
- Select all lines on the model and drag and drop the **Thick Beam** mesh from the Treeview onto the selected features.
- Click **OK** to accept default element orientation.



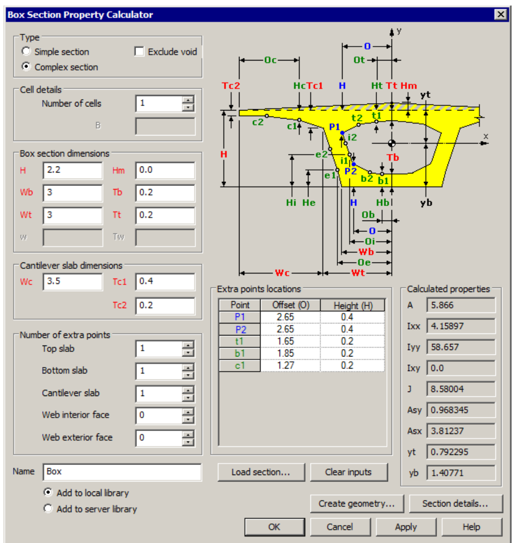
Defining and Assigning Geometric Properties

The bridge is formed from box sections of a constant width and depth. Five construction stages are to be modelled. Each of the stages uses the same section properties.

Tools
Section Property Calculator >
Box Section…

The box section property calculator will be used to define the box section required.

- Choose to define a **Complex section** type, and then enter the section dimensions as shown, before clicking **OK** to add the section definition to the user local library.

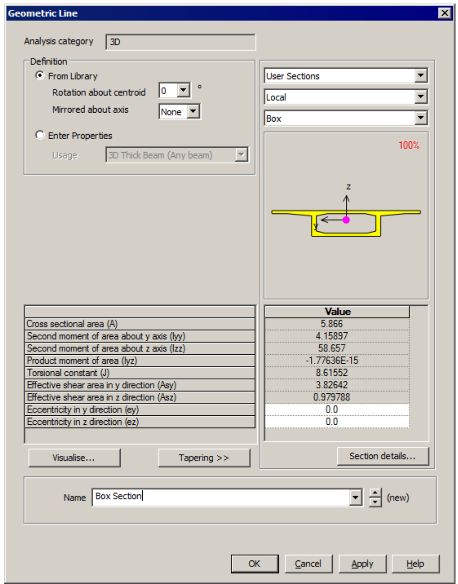


Attributes
Geometric >
Section Library…

- Select **User Sections**, **Local** and **Box** from the drop-down selection lists.

- Enter the attribute name as **Box Section** and click **OK**.

- Select all the lines on the model (hold down **Ctrl + A** keys) and drag and drop the Box Section geometric attribute from the Treeview into the main window to assign the property to the selected lines.



The box section will be fleshed and can be seen if the isometric button is selected.

The view window currently shows a plan view of the model – as viewed along the Z-axis.

Y: N/A Select the View along Y-axis button from the Status bar to view a side view of the model.

Turn off the fleshing.

Note. When time dependent effects are to be considered (as in this example) the geometric attribute must contain fleshing information that can be used to determine section properties such as the perimeter exposed to drying, as required in shrinkage loss calculations.

Defining the Material

Attributes
Material >
Material Library...

- Select material **Concrete**, region **Europe,** standard **EN1992-1-1:2004/2014** and grade **Advanced define** from the drop down lists.

- Accept the Advanced Define defaults, enter the attribute name as **Concrete** and click **OK** to add the material attribute to the Treeview.

- With all the lines on the model selected, drag and drop the **Concrete (…)** material from the Treeview onto the selected features.

Note. Advanced define concrete materials provide additional properties, such as cement type, that are required by the prestress to calculate time dependent effects. Since we will consider these effects in this example we must use the correct material.

Supports

Attributes
Support…

- Translations in the **Y** and **Z** and rotations about the **X, Y** and **Z** axes must be **Fixed**.
- For the **Translation in X** support enter a **spring stiffness of 160e3**
- Enter a attribute name of **Fully Fixed** and click **OK**
- Select the points at both ends of the bridge model and, from the Treeview, drag and drop the support attribute **Fully Fixed** onto the selected features, ensure the **All analysis loadcases** option is selected and click **OK**

Note. This simplified example considers only the span between two piers. The connection to the piers is idealised using rigid supports for simplicity. Since the prestress loading acts in the X direction some freedom must be allowed in order for the prestress to be mobilised in the beam elements, hence the use of the spring stiffness value. A fully rigid restraint would prevent the prestress load acting on the beam.

Loading

As well as the self-weight of the structure the effects of the prestress force will also be considered in this analysis. Firstly, the self-weight will be defined.

Gravity can be applied to any loadcase from the Treeview by selecting the loadcase with the right-hand mouse button and selecting gravity. Alternatively gravity can be assigned to all loadcases in an analysis by right clicking the analysis and selecting the Add gravity menu item.

Creating Activation and Deactivation Datasets

In order to carry out a staged construction analysis the birth and death facility is used.

Attributes
Birth and Death…

- Select the **Activate** option and click **Next**
- Enter the attribute name as **Activate** and click **Apply**
- Click **Back** so the deactivate attribute can be defined.
- Select the **Deactivate** option and click **Next**

- Ensure the stiffness reduction factor is set to **1e-6**
- Select **Percentage to redistribute** and ensure the value is **100**%. Enter the attribute name as **Deactivate** and click **Apply**
- Change the Stiffness reduction factor to **1e-12** and change the name to **Deactivate closing segment** and click **Apply**, and then **Finish**

Note. Deactivated elements are still present in the analysis but their stiffness is reduced such that they do not contribute to global stiffness. The deactivated elements undergo deformations for reasons of compatibility. Where two parts of a structure are initially separate and are finally joined during staged construction it is important that deformations in one part of the structure do not affect the other. For example two cantilevered segmental structures may be constructed in different sequences before finally being joined in a final stage. In order to prevent deformations from one structure affecting the other a second Deactivate attribute is applied to the final closing segment. The stiffness of this segment is severely reduced by comparison to the other deactivation attribute used so that deformations of one structure are properly isolated from the other. In this way, segments are introduced tangential and in the as-drawn (straight) shape apart from the closing segment which will fit between the two cantilevers. In a real project, avoiding a 'kink' at closure is typically an important design consideration.

Setting the percentage to redistribute as 100% ensures that the forces in the deactivated elements are fully redistributed to the active elements. See the online help for more information

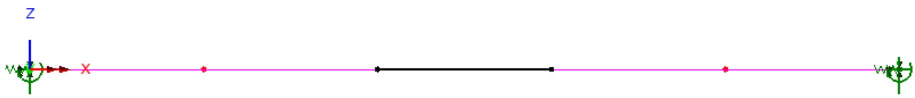
Modelling of Construction Stage 1

- In the Treeview expand **Analysis 1** then rename **Loadcase 1** to be **Stage 1** by selecting Loadcase 1 with the right-hand mouse button and using the **Rename** option.

All elements in the model that are not required for the first stage analysis need to be de-activated.

- In the graphics window select the two lines defining the segments in stage 2.

- From the Treeview and with the two segments selected, assign the deactivation attribute **Deactivate** ensuring that it is assigned to loadcase **Stage 1** and click **OK**
- Now select only the centre line which represents the closing segment



- From the Treeview assign the deactivation attribute **Deactivate closing segment** ensuring that it is assigned to loadcase **Stage 1** and click **OK**

Defining Loadcase Properties

- In the Treeview select **Stage 1** using the right-hand mouse button and from the **Controls** menu select **Nonlinear and Transient**.
- On the Nonlinear & Transient dialog select the **Nonlinear** option in the top-left hand corner, leave the incrementation type as **Manual** and click **OK** to accept all default entries.

Note. The nonlinear analysis control is required when birth and death of elements are specified. It is only required on the first loadcase and will be continued through to the subsequent loadcases.

Construction Stages 2 to 3

Stages 2 and 3 require loadcases to be generated to specify the duration of the construction process. Lines on the model must be selected according to the construction stage being considered and activation attributes must be assigned to these lines.

Stage 2

The elements in the second construction stage must now be activated

- In the graphics window select the segments to be added in Stage 2 of the construction sequence.

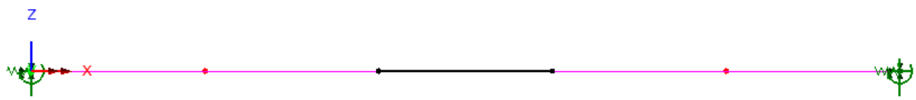
- Assign the activation attribute **Activate** from the Treeview. Enter **Stage 2** in the loadcase combo box and ensure that **Set as the active loadcase** is checked. Click **OK** to finish activation of the selected sections.

Note. As the **Stage 1** loadcase has been set as nonlinear with manual control the subsequent loadcase controls do not need to be defined as they will take the properties defined in loadcase 1.

Note. If you need to verify loading assignments such as self-weight or to check when particular lines (and hence elements) become active in an analysis select a line on the model and then, using the right-hand mouse button, select Properties. Expand the droplist for **Activate** and select **Details**. The analysis and loadcase to which the activation is assigned is shown.

Stage 3

The elements in the third construction stage now need to be activated.



- In the graphics window select the segment to be added in Stage 3 of the construction sequence.
- With the middle segment selected, assign the activation attribute **Activate** from the Treeview. Enter **Stage 3** in the loadcase combo box and ensure the **Set as the active loadcase** is checked. Click **OK** to finish the activation of the selected section.

Note. The active elements at each stage can be visualised by turning off the Geometry layer, selecting the **Show only activated elements** option on the view properties dialog and then activating each loadcase in turn.

Long term

A final loadcase will be added to consider the long term losses at the end of service.

Analyses
Loadcase…

- Enter **Long term** as the loadcase name and click OK.

Assigning Gravity Loading

- In the Treeview right click on **Analysis 1** and select **Add Gravity**. This adds gravity to all loadcases in the analysis.

Concrete age

To calculate the time dependent losses the prestress wizard needs to know the age of the concrete at the time it is activated in the model. This is achieved by assigning age attributes.

Attributes
Age…

- Enter the **Age at activation time** as **14 days**
- Enter the **Age when shrinkage begins** as **3 days**
- Enter the name as **Age 14 days** and click **Apply**
- Change the **Age at activation** to be **5 days**
- Change the **Age when shrinkage begins** to **5 days**
- Enter the name as **Age 5 days** and click **OK.**
- Select all the lines on the model (hold down Ctrl + A keys) and drag and drop the **Age 14 days** attribute from the Treeview into the main window to assign the property to the selected lines.
- Now select only the centre line and drag and drop the **Age 5 days** attribute from the Treeview to central closing segment.
- Click elsewhere in the view window to deselect the previously selected line.

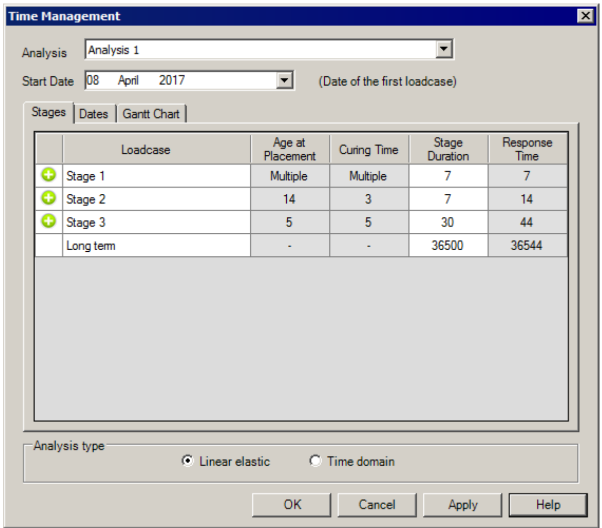
Time management

The time management dialog is used to specify the duration of each construction stage. Loadcases representing stages in a construction sequence are indicated with a green “Plus” symbol in the dialog, as shown below. Age at placement (start of the loadcase) and curing time are displayed based on the age attributes assigned to the activated elements for each stage where available.

The stage durations are required by the prestress wizard to calculate prestress losses for each loadcase. The response time is the total analysis time at the end of each loadcase, the values of prestress force are calculated at this time.

Bridge
Time Management…

- Enter the stage durations (in days) for each loadcase as shown, ensure a **linear elastic** analysis type is selected and click **OK**.



This completes the definition of the model geometry and the staged construction process.

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.

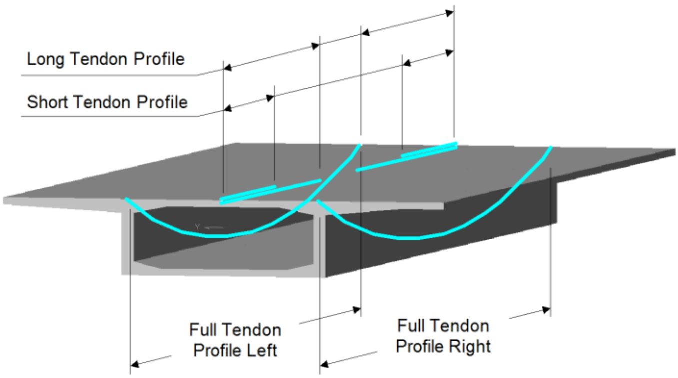
Prestress Loading

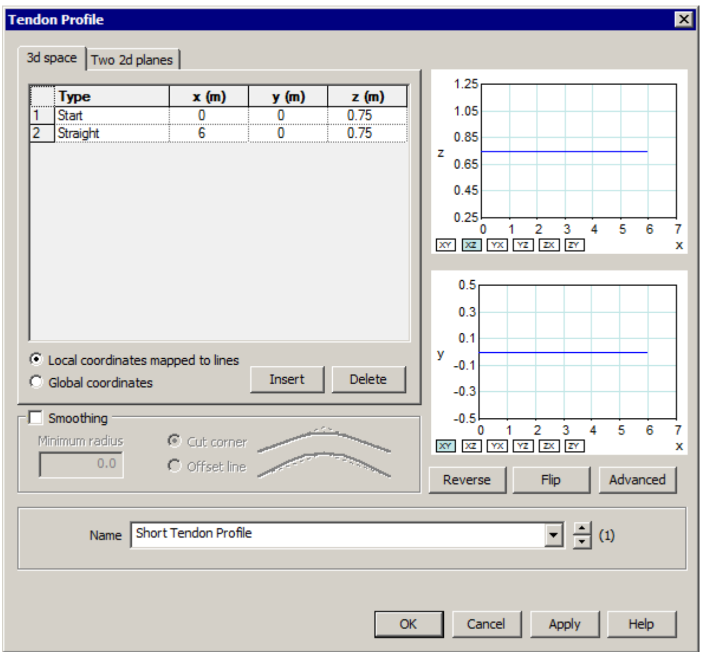
To model prestress loading requires the following:

1. **Definition of tendon profiles**
2. **Definition of tendon properties**
3. **Definition of tendon loading attributes**
4. **Assignment of tendon loading attributes**

Defining the Tendon Profiles

Four tendon profiles are to be defined: one short and one long located in the top slab, and one located in each side of the box section. The profiles are to be defined as local coordinates with respect to the relevant line(s) representing the box section, to which they will be assigned to.





Short tendon profile

Utilities
Prestress >
Tendon Profile...

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

- Select the option **Local coordinates mapped to lines**
- Enter the tendon profile into the grid as shown in the following table:

Type	x (m)	y (m)	z (m)
Start	0	0	0.75
Straight	6	0	0.75

- Click **Apply**

The tendon profile is added to the Tendon Profile folder in the Treeview.

Long tendon profile

- Now, to start entering another tendon profile, enter the name as **Long Tendon Profile**
- Enter the tendon profile into the grid as follows:

Type	x (m)	y (m)	z (m)
Start	0	0	0.75
Straight	12	0	0.75

- Click **Apply**

The tendon profile is added to the Tendon Profile folder in the Treeview.

Full tendon profile left

- Enter the attribute name as **Full Tendon Profile Left**
- Enter the tendon profile into the X, Y, Z part of the grid as follows (Use the **Tab** key to create each new line):

Type	x (m)	y (m)	z (m)
Start	0	2.8	0.75
Parabola Bulge	15	2.8	-1
Parabola End	30	2.8	0.75

- Change the Type settings from Straight to be as shown by clicking inside the cell and selecting **Parabola Bulge** for the second row and **Parabola End** for the last row.

- Click **Apply**

Full tendon profile right

- Enter the attribute name as **Full Tendon Profile Right**
- Enter the tendon profile into the X, Y, Z part of the grid as follows (Use the **Tab** key to create each new line):

Type	x (m)	y (m)	z (m)
Start	0	-2.8	0.75
Parabola Bulge	15	-2.8	-1
Parabola End	30	-2.8	0.75

- Click **Apply** and then click **OK**

The long tendon profiles are added to the Tendon Profile folder in the Treeview.

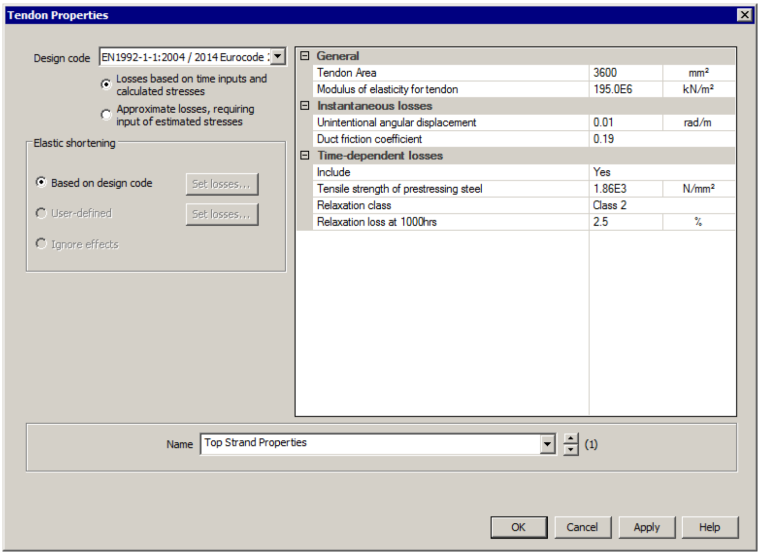
Note. Tendon profile definitions can be reviewed before proceeding further by clicking the up and down buttons adjacent to the tendon profile name.

Defining the Tendon Properties

- Ensure that the **EN1992-1-1:2004 / 2014 Eurocode 2** design code entry is selected from the drop down list.
- Ensure **Losses based upon time inputs and calculated stresses** is selected

Utilities
Prestress >
Tendon Properties...

The Multi-strand prestressing system being considered here consists of 12, 15mm diameter strands that give a nominal area of 1800mm^2. For simplicity only a single tendon is modelled in the top slab and this will be given the properties of two tendons, i.e. an area of 3600mm^2

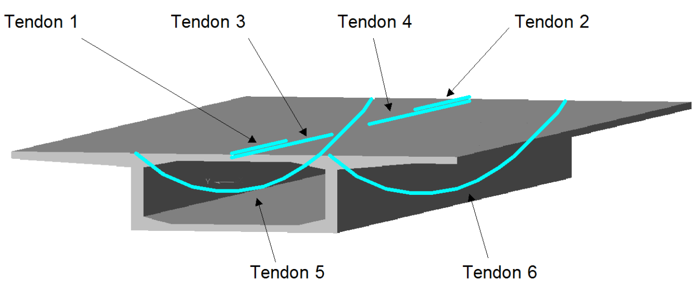


- Enter the attribute name as **Top Strand Properties**
- Enter the tendon area as **3600**
- Accept the defaults for the remaining properties
- Click **Apply**

The tendon property is added to the Tendon Properties folder in the Treeview.

- Change the name to **Web Strand Properties**
- Enter the tendon area as **1800**
- Click **Apply** and then click **OK**

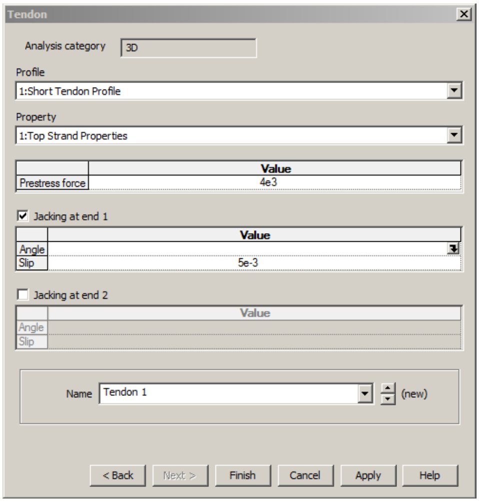
Defining the Tendon Loading



For tendon 1

Attributes
Loading…

- Select the **Tendon** option and click **Next**
- Select **Short tendon profile** from the Profile drop-down list
- Select **Top Strand Properties** from the Property drop-down list.
- Enter a prestress force of **4E3**
- Ensure that jacking at **End 1** is the only jacking detail selected and enter the slip to be **5E-3**
- Enter the attribute name as Tendon 1
- Click **Apply**.



This completes the tendon loading for tendon 1.

For tendon 2:

- Ensure the profile combo box is set to **Short tendon profile** and the property combo box is set to **Top Strand Properties**
- Ensure the prestress force is set to **4E3**
- Ensure that jacking from **End 2** is the only jacking detail selected and enter the slip as **5E-3**
- Enter the attribute name as **Tendon 2**.
- Click **Apply**.

This completes the tendon loading for tendon 2.

For tendon 3:

- Change the profile combo box so that **Long tendon profile** is displayed and the property combo box so that **Top Strand Properties** is displayed.
- Ensure the prestress force is set to **4E3**
- Ensure that the jacking from **End 1** is only selected and enter the slip as **5E-3**
- Enter the attribute name as **Tendon 3**.
- Click **Apply**.

This completes the tendon loading for tendon 3.

For tendon 4:

- Ensure the profile combo box is set to **Long tendon profile** and the property combo box is set to **Top Strand Properties**
- Ensure the prestress force is set to **4E3**
- Ensure that the jacking from **End 2** is only selected and enter the slip as **5E-3**
- Enter the attribute name as **Tendon 4**.
- Click **Apply**.

This completes the tendon loading for tendon 4.

For tendon 5:

- Change the profile combo box so that **Full Tendon Profile Left** is displayed and change the property combo box so that **Web Strand Properties** is displayed
- Enter the prestress force as **2E3**
- Ensure that the jacking from **End 1** and **End 2** is selected and enter the slip as **5E-3** at each end.
- Enter the attribute name as **Tendon 5**.
- Click **Apply**.

This completes the tendon loading for tendon 5.

For tendon 6:

- Change the profile combo box so that **Full Tendon Profile Right** and the property combo box is set to **Web Strand Properties**
- Ensure the prestress force is set to **2E3**
- Ensure that the jacking from **End 1** and **End 2** is selected and enter the slip as **5E-3** at each end.
- Enter the attribute name as **Tendon 6**.
- Click **Apply**.

This completes the tendon loading for tendon 6.

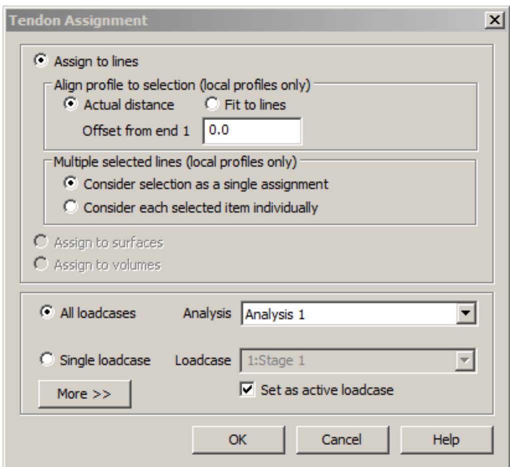
Note. Tendon loading assignments can be reviewed and corrected if necessary before proceeding further by clicking the up and down buttons adjacent to the tendon profile name.

- Click **Finish**

Assigning the Tendon Loading

- Select the line tendon 1 acts on.

- Drag and drop **Tendon 1** from the Treeview onto the selected line.
- On the Tendon Assignment dialog ensure the **All loadcases** option is selected and click the **OK** button.



- Select the line tendon 2 acts on.

- Drag and drop **Tendon 2** from the Treeview onto the selected line
- On the Tendon Assignment dialog ensure the **All loadcases** option is selected and click the **OK** button.

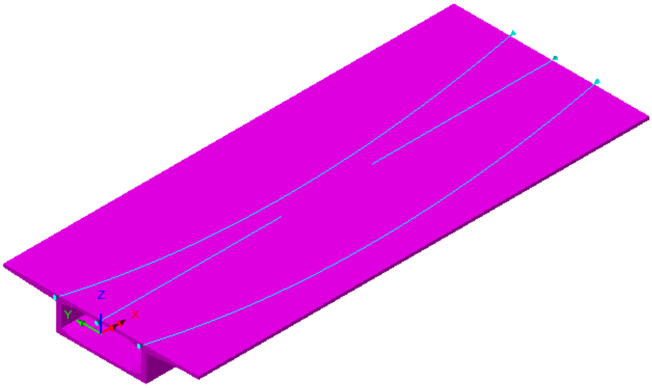
- Select the two lines tendon 3 acts on.

- Drag and drop **Tendon 3** from the Treeview onto the selected line
- On the Tendon Assignment dialog select the **More** button. Choose **Range of loadcases from** and select **Stage 2** in the combo box. Choose **Until end of analysis**. Click the **OK** button.
- Select the two lines tendon 4 acts on.

- Drag and drop **Tendon 4** from the Treeview onto the selected line
- Choose **Range of loadcases from** and select **Stage 2** in the combo box. Choose **Until end of analysis**. Click the **OK** button.
- Select the lines tendon 5 acts on. (All the lines in the model.)
- Drag and drop **Tendon 5** from the Treeview onto the selected line
- Choose **Range of loadcases from** and select **Stage 3** in the combo box. Choose **Until end of analysis**. Click the **OK** button.
- Ensure all lines in the model are still selected
- Drag and drop **Tendon 6** from the Treeview onto the selected line
- Choose **Range of loadcases from** and select **Stage 3** in the combo box. Choose **Until end of analysis**. Click the **OK** button.

Turn the fleshing on using the **Fleshing on/off** button if it is not already turned on

Turn the loading on using the **Loading on/off** button if it is not already turned on



- Turn off the fleshing and the loading.

Note. Calculation of prestress loading including time dependent effects involves an iterative approach to determine concrete stresses at tendon locations used to estimate creep losses. The calculation is therefore performed as part of the analysis.

Note. Tendon profile visualisations can be controlled by accessing the **Attributes** layer Properties dialog and on the Loading tab pressing the **Settings** button to change the visualisation settings.

Notes on editing Prestress data

- ❑ Multiple tendon loads can be viewed and edited by selecting the loads in the Treeview, right clicking and selecting **Edit Assignments**.
- ❑ Tendon profile data is stored in the Treeview.
- ❑ Editing of tendon profile data is permitted, requiring a re-solve to be undertaken if done.

Save the model

File
Save

Save the model file.

Running the Analysis

With the model loaded:

Open the **Solve now** dialog. Ensure **Analysis 1** and **Update prestress loading** is selected and press **OK** 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...

Analysis loadcase results are added to the Treeview.

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

- **segmental bridge.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- **segmental bridge.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. Select **Yes** to view the output file. Any errors listed in the text output window should be corrected in LUSAS Modeller before saving the model and re-running the analysis. Note that a common error is to forget to assign attribute data (such as geometry, mesh, supports, loading etc.) to the model. If the errors cannot be identified the model may be rebuilt and the prestress loading redefined.

Rebuilding the Model

If it proves impossible for you to correct the errors reported a file is provided to enable you to re-create the model up to the point of defining the Prestress Loading.

- **segmental_bridge_modelling.vbs** carries out the geometric modelling of this example.

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 **Segmental Bridge**

File
Script >
Run Script...

- To recreate the model, select the file **segmental_bridge_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

The geometry of the model has been created. Now return to the section entitled **Prestess Loading** earlier in this example and re-define the tendon properties.

Viewing the Results

Analysis loadcase results are present in the Treeview for each stage. The first solved loadcase (Increment 1) will be set active.

Bending moments of My over the segments are to be investigated for each stage of the construction process. A summary of results on each results plot allows a comparison of maximum displacements for each stage.

In the following images the tendon profile visualisations are not shown. They can be turned off by clicking on the button.

Generating a report showing Prestress Details

Utilities
Reporting >
New Report...

- Enter the Title as **Segmental Bridge**, 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.

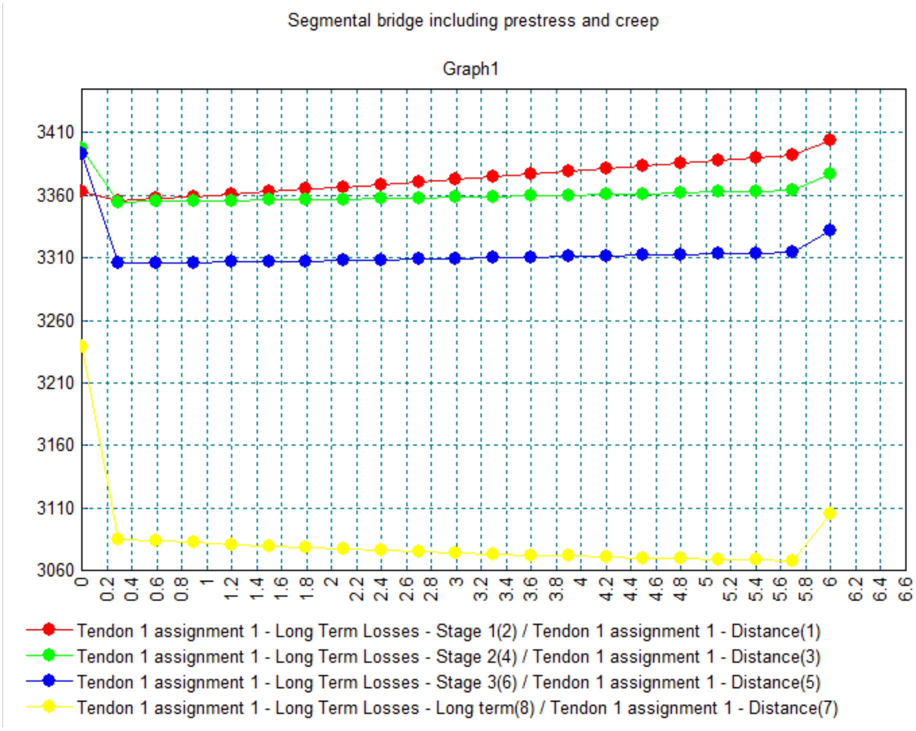
Graphing of Prestress Force in Selected Tendons

Tendon 1

Create a graph of the prestress force for Tendon 1.

Utilities
Graph Wizard…

- Select the **Prestress** option and click **Next**
- Select the check box for **Tendon 1 assignment 1** and ensure all other check boxes are deselected. Click **Next**
- Click **Finish** to display the graph of prestress force in the tendon.



This shows the remaining prestress force (long term) in the tendon following each stage of construction.

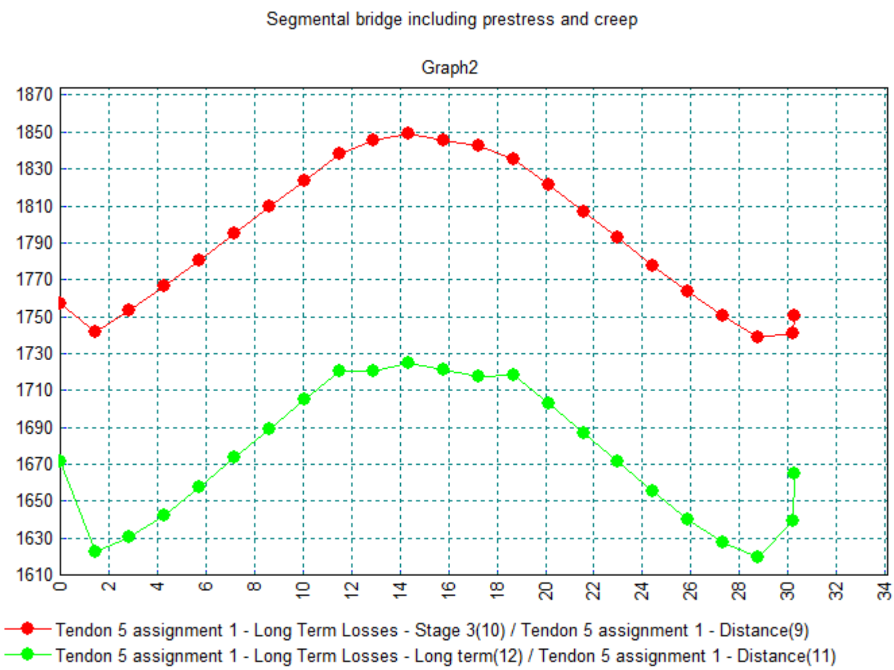
- Close the Graph.

Tendon 5

Now create a graph of prestress force for Tendon 5:

Utilities
Graph Wizard…

- Select the **Prestress** option and click **Next**
- Select the check box for **Tendon 5 assignment 1** and ensure all other check boxes are deselected. Click **Next**



Close the graph window.

Stage 1 results

- In the Treeview right-click on **Stage 1** and select the **Set Active** Option.
- Turn off the display of the **Mesh, Geometry**, **Utilities** and **Attributes** layers from the Treeview.
- If not there already, add the **Deformed mesh** layer to the Treeview.
- Using the **Deformations...** button at the bottom of the Treeview and specify factor as **1E3**. Click the **Window summary** option.
- Add the **Diagrams** layer to the Treeview. On the dialog, select **Force/Moment - Thick 3D Beam** results from the entity drop-down list and component **My.** Select the **Diagram Display** tab.
- Ensure the **Label values** option and the **Peaks only** option is selected.
- Click **OK** to display the bending moment diagram for the active elements for Stage 1.

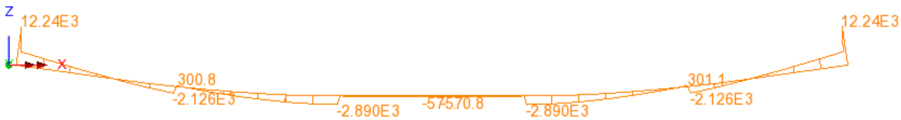
Note. The maximum deflection and maximum and minimum bending moments are shown in the window summary.

Stage 2 results

- In the Treeview right-click on **Stage 2** and select the **Set Active** option.

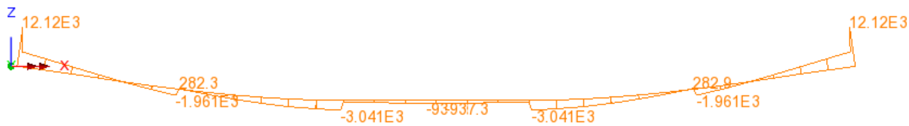
Stage 3 results

- In the Treeview right-click on **Stage 3** and select the **Set Active** option.



Long term results

- In the Treeview right-click on **Long term** and select the **Set Active** option.



Note. The bending moment diagrams exhibit a sharp drop in end moment next to the fixed support. This occurs because the prestress loading is applied to the element very close to the support but not at the support. Hence the end moment in the beam matches the reaction at the support and then drops rapidly where the prestress loading is applied.

This completes the example.

Notes on time dependency

This example demonstrates calculation of prestress losses at specified times such as is required in a staged construction analysis. For each stage of construction the prestress losses are determined for the time at the end of each loadcase. The losses account for the construction history by considering the change in stress between the loadcases (or stages of construction) and determining the cumulative effect.

The linear analysis carried out only considers time dependent effects in determination of the prestress forces. This type of analysis however does not capture the deformations arising from the time dependent effects. The creep and shrinkage strains that occur in the concrete are not included and therefore the final deflections will be inaccurate for long term effects.

More accurate deformations may be computed by using a material model capable of capturing these effects in a transient analysis. For this example the linear elastic concrete material could be replaced with the Concrete Creep EN1992 material (accessed from the Attributes > Material > Specialised menu item). In addition, selecting 'Time Domain' from the Time management dialog (accessed from the context menu for an analysis entry in the Analyses Treeview) would be required. This will apply the required nonlinear controls to the staged construction loadcases to carry out the transient analysis.