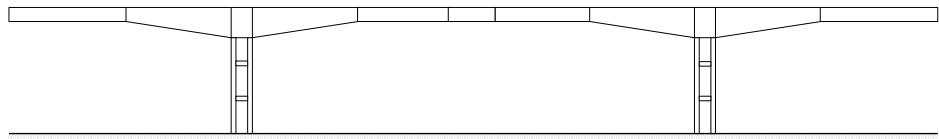


Balanced Cantilever Construction of a Segmental Post Tensioned Bridge

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

Description

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

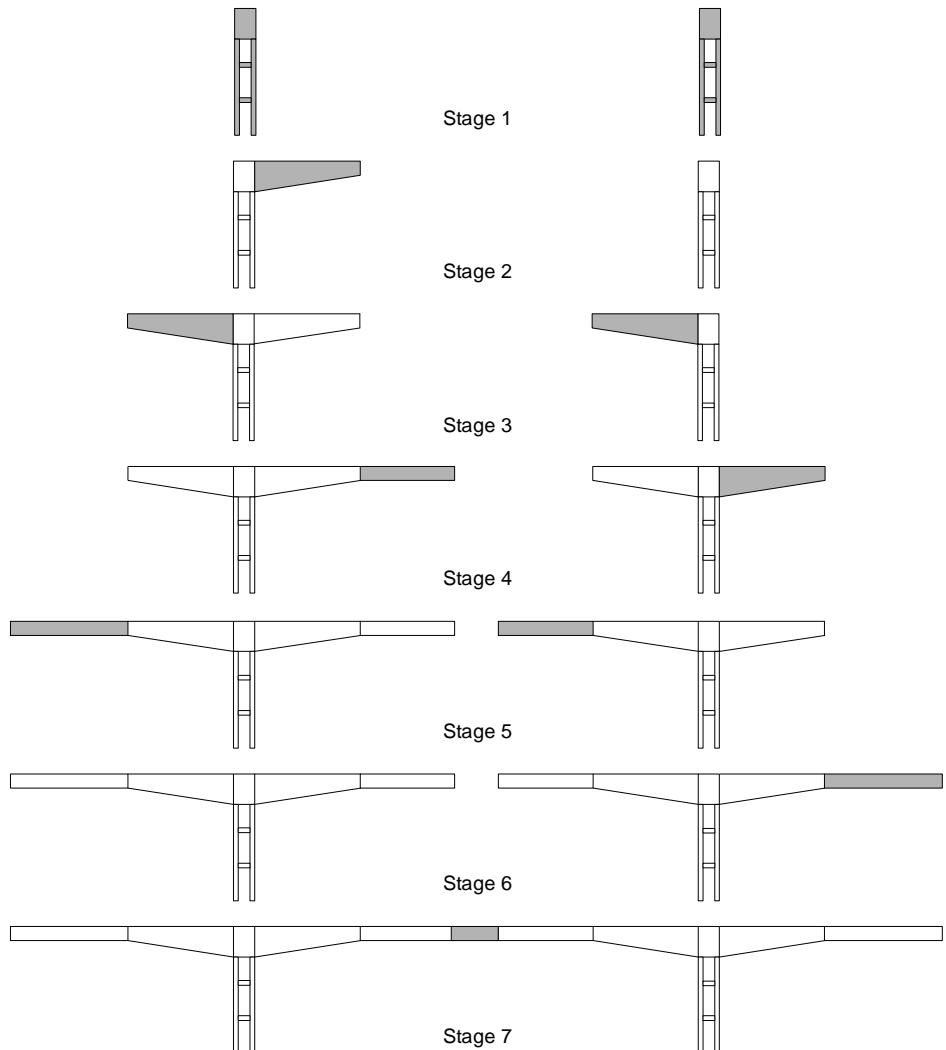


A complete analysis of a large balanced cantilever segmentally constructed bridge is a complex and large analysis to undertake. For this example the bridge geometry has been simplified and the number of segments used in the bridge has been reduced, resulting in longer segment lengths than would otherwise be present. This has been done in order to keep the number of construction stages to a minimum, and to concentrate on the required definition and assignment of tendon profiles, properties and loading.

Traveller loads/formwork and the support they provide before post-tensioning is added have been excluded from this example for simplicity.

Construction stages

The bridge will be constructed outwards from two piers, reaching abutments at the ends and with a final closing segment to join the two constructions together. The construction stages to be modelled are shown in the following diagram.

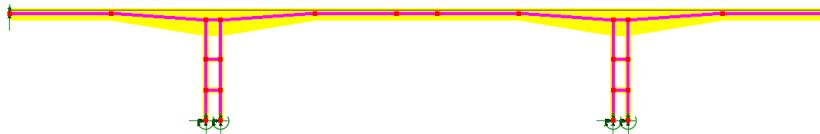


The example incorporates staged construction and the assignment of prestress loading.

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

Modelling methodology

To model this bridge the lines representing deck segments are defined to follow the centroid of the sections.



Objectives

The output requirements of the analysis are:

- Maximum moments during construction

Keywords

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

Associated Files



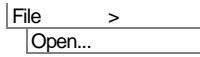
- balanced_cantilever_bridge.mdl** contains the model geometry with all of the attributes assigned. The example will use this model file as a starting point.

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.

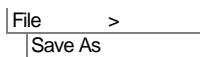
Creating the model

For this example a model file is provided:



- Open... Open the read-only file **balanced_cantilever_bridge.mdl** that is either provided, or located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Depending upon your access rights it may be necessary to copy this file to another folder before it can be opened, and remove the read-only protection before it can be saved to the new folder. Once opened:



- Save the model file to your local Projects directory.

Creating Activation and Deactivation Datasets

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

- 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 elements that are still present in an 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.

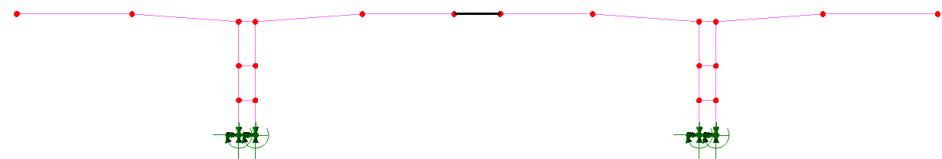
Setting the percentage to redistribute as 100% ensures that the forces in the deactivated elements are fully redistributed to the active elements. See the LUSAS online help for more information.

Modelling of Construction Stage 1

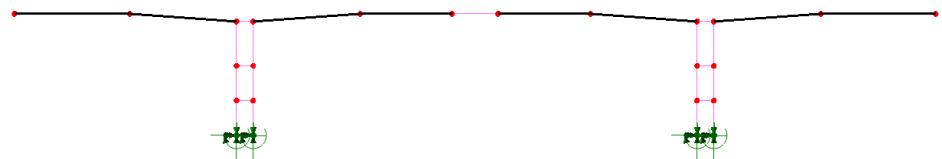
- In the Treeview expand **Analysis 1** then rename **Loadcase 1** to be **Stage 1** by selecting Loadcase 1 with the right-hand mouse button and using the **Rename** option.

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

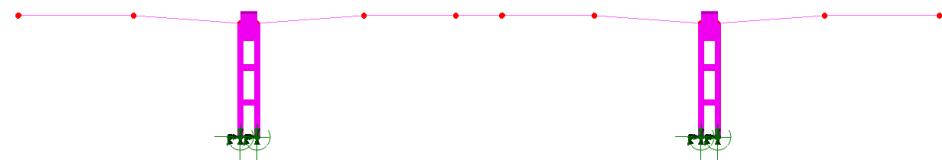
- In the view window select the central closing segment (line 22) as shown below.



- From the Treeview, assign the deactivation attribute **Deactivate closing segment** ensuring that it is assigned to loadcase **Stage 1** and click **OK**
- Now select only the other deck lines, except those which join the two legs of each pier:



- From the Treeview assign the deactivation attribute **Deactivate** ensuring that it is assigned to loadcase **Stage 1** and click **OK**.
- Turn the fleshing on using the **Fleshing on/off** button if it is not already turned on. Only active elements will be fleshed:



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 7

Stages 2 to 7 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 view window select line 20 as shown below:



- 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 dropdown 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.

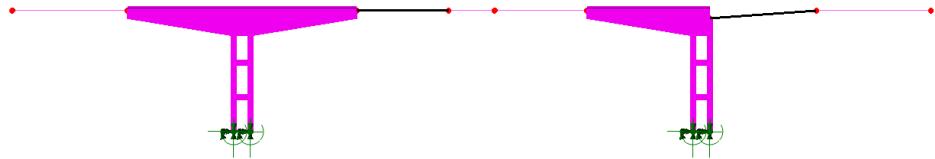
- In the view window select lines 18 and 24:



- 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.

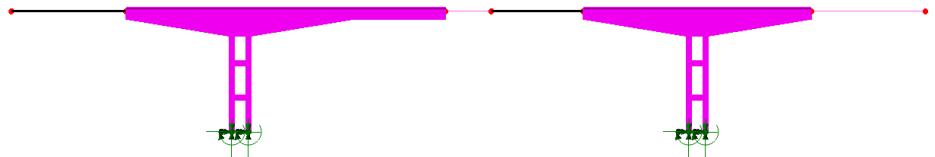
Stage 4

- In the view window select lines 21 and 26. Assign the activation attribute **Activate** and enter **Stage 4** in the loadcase combo box. Click **OK** to finish the activation of the selected section.



Stage 5

- In the view window select lines 17 and 23. Assign the activation attribute **Activate** and enter **Stage 5** in the loadcase combo box. Click **OK** to finish the activation of the selected section.



Stage 6

- In the view window select line 27. Assign the activation attribute **Activate** and enter **Stage 6** in the loadcase combo box. Click **OK** to finish the activation of the selected section.



- In the view window select the far left point; point 17:



- Assign the support attribute **Roller** from the  Treeview. Choose “From loadcase” and ensure the loadcase **Stage 6** is selected. Click **OK** to assign the support.



Note. The support is assigned in the loadcase *after* the adjacent line is activated in order to model the deflection caused by the weight of the segment.

Stage 7

- In the view window select line 22. Assign the activation attribute **Activate** and enter **Stage 7** in the loadcase combo box. Click **OK** to finish the activation of the selected section.



- In the view window select the far right point; point 24:



- Assign the support attribute **Roller** from the  Treeview. Choose “From loadcase” and ensure the loadcase **Stage 7** is selected. Click **OK** to assign the support.

Long term

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

Analyses

Loadcase...

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 loading calculations need to know the age of the concrete at the time it is activated in the model. This is achieved by assigning age attributes.

With the piers and the closing segment being cast in-situ they are taken as being released from their forms at 5 days, hence 5 days is both their curing time (age when shrinkage begins) and their age at activation / placement.

The precast deck segments are released from their forms at 3 days, but not placed on the bridge / loaded until 14 days.

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 all pier elements and the centre deck line (line 22) and drag and drop the **Age 5 days** attribute from the  Treeview onto these members.

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)

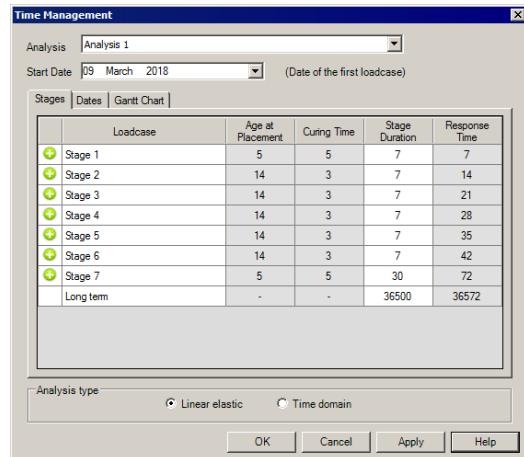
and curing time are displayed based on the age attributes assigned to the activated elements for each stage where available.

The stage durations are required by the prestress facility to calculate prestress losses for each loadcase. The response time is the total analysis time at the end of each loadcase, and 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**.



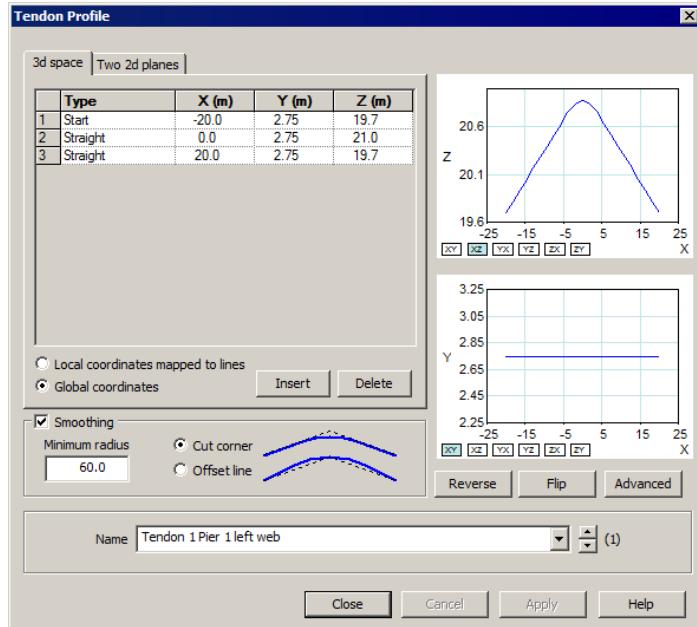
Prestress Loading

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

Utilities
Prestress >
Tendon Profile...



- Enter the attribute name as **Tendon 1 Pier 1 left web**
- Ensure the option **Global coordinates** is selected
- Enter the tendon profile into the grid as

Type	x (m)	y (m)	z (m)
Start	-20	2.75	19.7
Straight	0	2.75	21
Straight	20	2.75	19.7

- Tick **Smoothing** and enter a minimum radius of **60.0** using the **Cut corner** option
- Click **Apply**

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

- To enter the next tendon profile, enter the name as **Tendon 1 Pier 1 right web**

Balanced Cantilever Construction of a Segmental Post Tensioned Bridge

- Enter the tendon profile into the grid as follows:

Type	x (m)	y (m)	z (m)
Start	-20	-2.75	19.7
Straight	0	-2.75	21
Straight	20	-2.75	19.7

- Click **Apply**
- Enter the other tendons as follows:

Tendon 2 Pier 2 left web

Type	x (m)	y (m)	z (m)
Start	60	2.75	19.7
Straight	80	2.75	21
Straight	100	2.75	19.7

Tendon 2 Pier 2 right web

Type	x (m)	y (m)	z (m)
Start	60	-2.75	19.7
Straight	80	-2.75	21
Straight	100	-2.75	19.7

Tendon 3 Pier 1 left web

Type	x (m)	y (m)	z (m)
Start	-40	2.75	19.7
Straight	0	2.75	21.4
Straight	36	2.75	19.9

Tendon 3 Pier 1 right web

Type	x (m)	y (m)	z (m)
Start	-40	-2.75	19.7
Straight	0	-2.75	21.4

Straight	36	-2.75	19.9
----------	----	-------	------

Tendon 4 Pier 2 left web

Type	x (m)	y (m)	z (m)
Start	44	2.75	19.9
Straight	80	2.75	21.4
Straight	120	2.75	19.7

Tendon 4 Pier 2 right web

Type	x (m)	y (m)	z (m)
Start	44	-2.75	19.9
Straight	80	-2.75	21.4
Straight	120	-2.75	19.7

Tendon 5 Continuity left web

Type	x (m)	y (m)	z (m)
Start	20	2.75	19.7
Straight	60	2.75	19.7

Tendon 5 Continuity right web

Type	x (m)	y (m)	z (m)
Start	20	-2.75	19.7
Straight	60	-2.75	19.7

- Click **OK** to exit the dialog

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

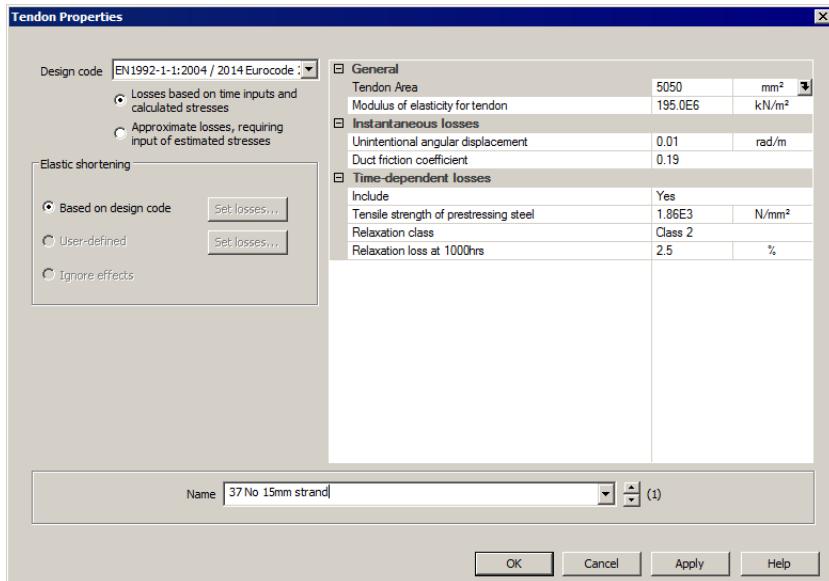


Note. Tendon profile definitions can be reviewed before proceeding further by clicking the up and down buttons adjacent to the tendon profile name.

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

The Multi-strand prestressing system being considered here consists of 37No 15mm diameter strands that give a nominal area of 5050mm².



- Enter the attribute name as **37 No 15mm strand**
- Enter the tendon area as **5050**
- Accept the defaults for the remaining properties
- Click **OK**

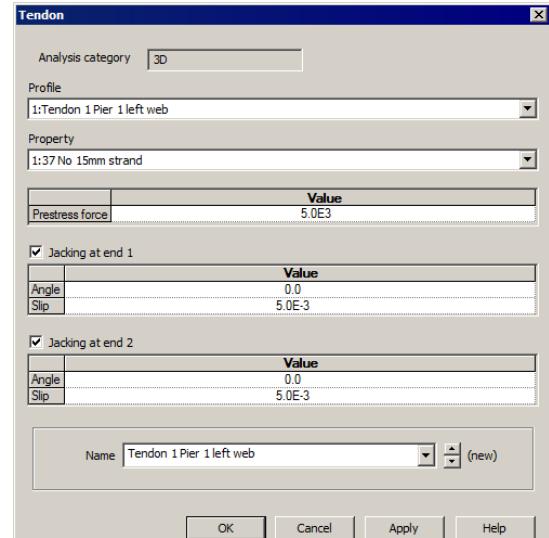
The tendon property is added to the Tendon Properties folder in the Utilities Treeview.

Defining the Tendon Loading

For tendon 1:

Attributes
Loading...

- Select the **Tendon** option and click **Next**
- Ensure the profile combo box is set to **Tendon 1 Pier 1 left web**
- Enter the prestress force as **5000**
- For **Jacking at end 1** enter the slip as **5E-3**.
- Tick for **Jacking at end 2** and also enter a slip of **5E-3**.
- Enter the attribute name as **Tendon 1 Pier 1 left web**
- Click **Apply**.
- Select each other Profile in turn, entering an appropriate **Name** and clicking **Apply** for each.



This completes defining the tendon loading attributes.



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.

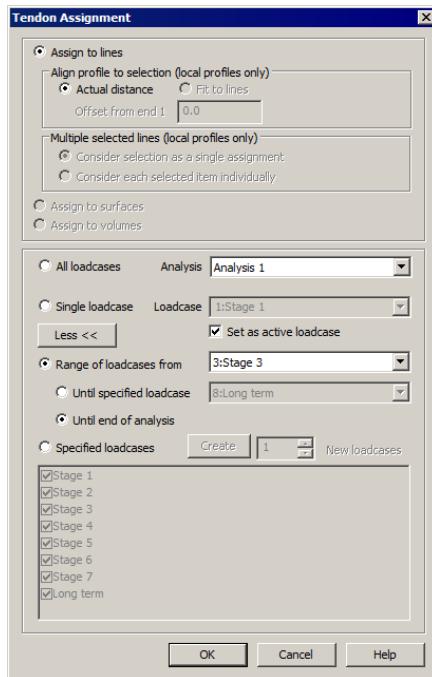
- Click **Finish**

Assigning the Tendon Loading

- Select the three lines tendon 1 acts on.



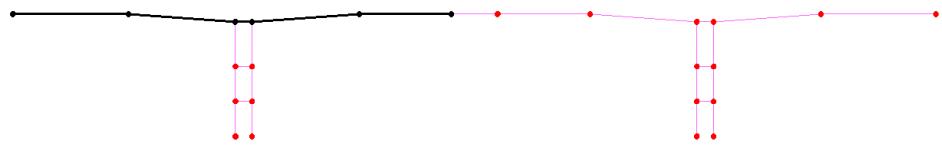
- Drag and drop **Tendon 1 Pier 1 left web** from the  Treeview onto the selected lines.
- On the Tendon Assignment dialog click “**More >>**” to show all loadcase options.



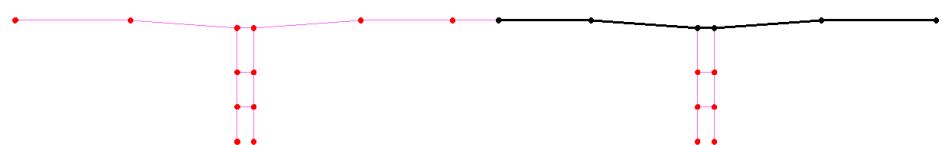
- Select **Range of loadcases from** and **Stage 3** and choose **Until end of analysis**. Click the **OK** button.
- Repeat to drag and drop **Tendon 1 Pier 1 right web** onto the same selected lines for the same loadcase settings.
- Now select the three lines tendon 2 acts on.



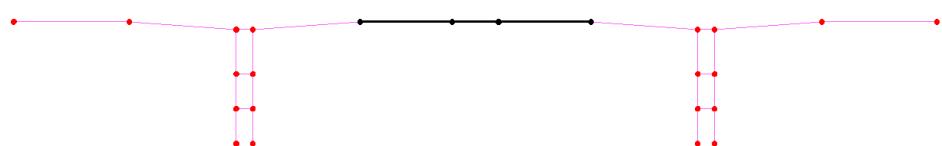
- Drag and drop **Tendon 2 Pier 2 left web** from the  Treeview onto the selected lines.
- Assign it to a **Range of loadcases from** **Stage 4** to the end of the analysis. Click the **OK** button.
- Repeat to assign **Tendon 2 Pier 2 right web** to the same lines.
- Select the five lines tendon 3 acts on.



- Drag and drop **Tendon 3 Pier 1 left web** from the  Treeview onto the selected lines.
- Assign it to a **Range of loadcases** from **Stage 5** to the end of the analysis. Click the **OK** button.
- Repeat to assign **Tendon 3 Pier 1 right web** to the same lines.
- Select the five lines tendon 4 acts on.



- Drag and drop **Tendon 4 Pier 2 left web** from the  Treeview onto the selected line
- Assign it to a **Range of loadcases** from **Stage 6** to the end of the analysis. Click the **OK** button.
- Repeat to assign **Tendon 4 Pier 2 right web** to the same lines.
- Select the three lines tendon 5 acts on.



- Drag and drop **Tendon 5 Continuity left web** from the  Treeview onto the selected line
- Assign it to a **Range of loadcases** from **Stage 7** to the end of the analysis. Click the **OK** button.
- Repeat to assign **Tendon 5 Continuity right web** to the same lines.

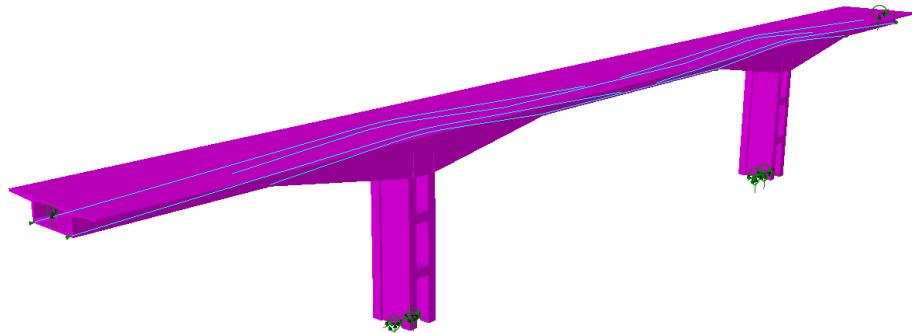


Turn the fleshing on if it is not already turned on.



Turn the loading on if it is not already turned on.

The tendon arrangement should look like this.



 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 Attributes Treeview, right clicking and selecting **Edit Assignments**.

Tendon profile data is stored in the Utilities Treeview.

Editing of tendon profile data is permitted, but should be done with caution.

Save the model



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

If the analysis is successful...

Analysis loadcase results are added to the Analyses  Treeview.

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



- balanced_cantilever_bridge~Analysis 1.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- balanced_cantilever_bridge~Analysis 1.mys** this is the LUSAS results file which is loaded automatically into the Analyses  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, a replacement model file (provided) may be opened in order to solve and view results.

Opening a replacement model

If it proves impossible for you to correct the errors reported, a replacement model file is provided to allow the remainder of the example (showing how to view results) to be completed.



balanced_cantilever_bridge_completed.mdl includes all required construction stages, age, time management values and prestress loading, ready for solving.

 Copy the original model file **balanced_cantilever_bridge_completed.mdl** to your working directory.

 Open the model **balanced_cantilever_bridge_completed.mdl** and click **No** to not load any previous results files.



Open the Solve Now dialog. Ensure **Analysis 1** and **Update prestress loading** are selected and press **OK** to run the analysis.

A LUSAS Datafile and a Prestress Results Datafile will be created from the model information. LUSAS Solver uses this datafile to perform the analysis.

Viewing the Results

Analysis loadcase results are present in the Analyses  Treeview for each stage.

Bending moments of M_y over the segments are to be investigated for each stage of the construction process. A summary of results on each results plot also 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 **Balanced Cantilever Bridge**, change the name to **Prestress Report** and click **OK**
- In the Reports  Treeview right click on **Prestress Report** and choose **Add Chapter**.
- Go to the **Prestress** tab and tick the **Prestress** check box and press **OK**.
- Right click on **Prestress Report** and choose **View Report**.

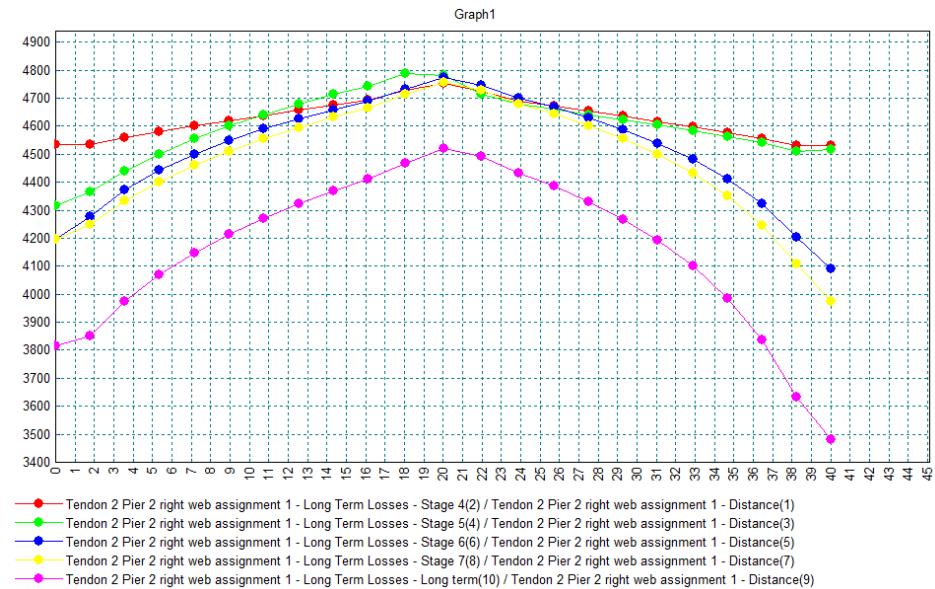
A PDF report is created giving details of the tendon profile, properties and loading assignments.

Graphing of Prestress Force in a Tendon

Utilities
Graph Wizard...

Add the graph of the Force for Tendon 2 Pier 2 right web.

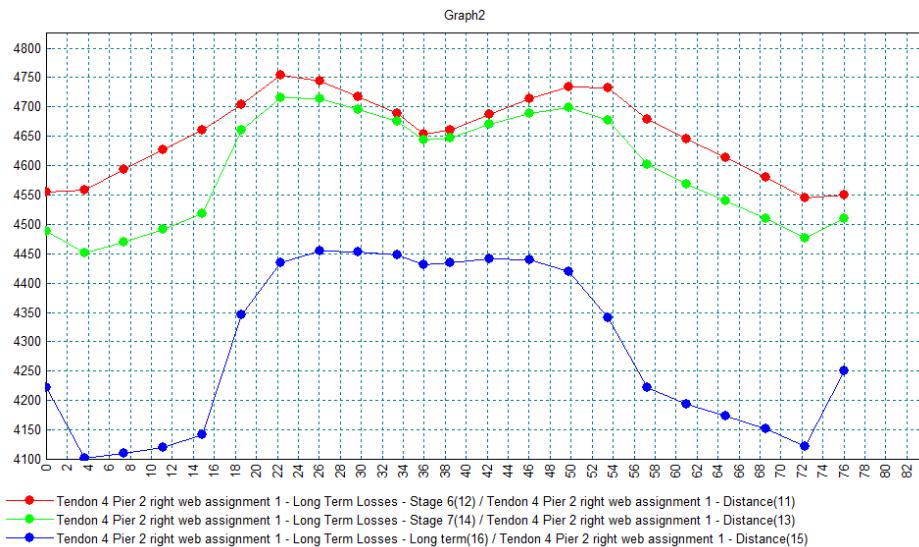
- Select the **Prestress** option and click **Next**
- Select the check box for **Tendon 2 Pier 2 right web assignment 1** and ensure all other check boxes are deselected. Click **Next**
- Click **Finish** to display the graph



Now create a graph of the Force for Tendon 4 Pier 2 right web:

Utilities
Graph Wizard...

- Select the **Prestress** option and click **Next**
- Select the check box for **Tendon 4 Pier 2 right web assignment 1** and ensure all other check boxes are deselected. Click **Next**
- Click **Finish** to display the graph.



Close the graph windows.

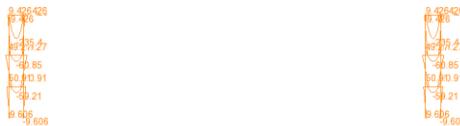
Stage 1 results

- 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 **100**. 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 are selected.
- Click **OK** to display the bending moment diagram for the active elements for Stage 1.

```

Scale: 1. 458.923
Zoom: 100.0
Csys: (0.0, 1.0, 0.0)
Nonlinear analysis
Analysis: Analysis 1
Diagram stage: Stage 1, 1 increment 1
Results file: balanced_cantilever_bridge_completed-Analysis 1.mys
Maximum displacement 0.18979E-3 at node 10
Deformation exaggeration: 100.0
Diagram entity: Force/Moment - Thick 3D Beam
Diagram unit: mm (Units: AN m)
Diagram maximum 50.9097 at node 1 of element 21
Diagram minimum -23.41 at Gauss point 6 of element 35
Diagram scale: 1. 0.042479

```



- If necessary, view the model along the Y-axis.



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.

```

Scale: 1: 448.023
Zoom: 100.0
Eye: (0.0, -1.0, 0.0)
Nonlinear analysis
Analysis: Analysis 1
Loadcase: BLong term, BIncrement B
Results for balanced cantilever bridge, completed-Analysis 1.mys
Maximum displacement 0.127037 at node 49
Deformation exaggeration: 100.0
Diagram entity: Force/Moment - Thick 3D Beam
Diagram component: My (Units: KN.m)
Diagram maximum: 11.67E3 at node 43 of element 45
Diagram minimum: -4.003E3 at node 69 of element 45
Diagram scale: 1: 0.110102E-3

```



Maximum and minimum results

Analyses

Envelope...

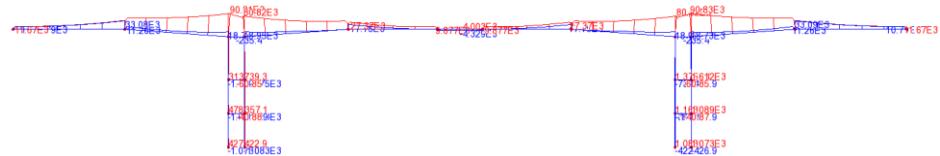
- Select all the loadcases and click the double right arrow to move them into the envelope. Give the name of **Construction Envelope** and click **OK**.
- In the Treeview right-click on **Construction Envelope (Max)** and select the **Set Active** option.
- Ensure that the Entity **Force/Moment - Thick 3D Beam** is selected and component **My**. Click **OK**. If a message pops up stating that the deformed mesh will no longer be available, click **OK**.

The maximum moments which occur throughout the construction stages will be displayed.



To add minimum results to the same plot:

- In the Treeview double-click on the **Diagrams : My (Force/Moment - Thick 3D Beam)** entry and on the Properties dialog select the **Show minimum too** option, then click **OK**.



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 (that can be 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.

Balanced Cantilever Construction of a Segmental Post Tensioned Bridge