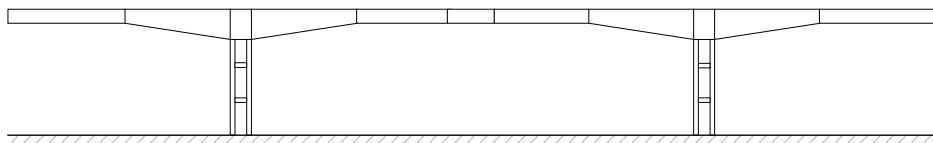


Balanced Cantilever Construction of a Segmental Post Tensioned Bridge

For LUSAS version:	22.0
For software product(s):	LUSAS <i>Civil & Structural Plus</i> and LUSAS <i>Bridge Plus</i>
With product option(s):	Nonlinear

Description

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

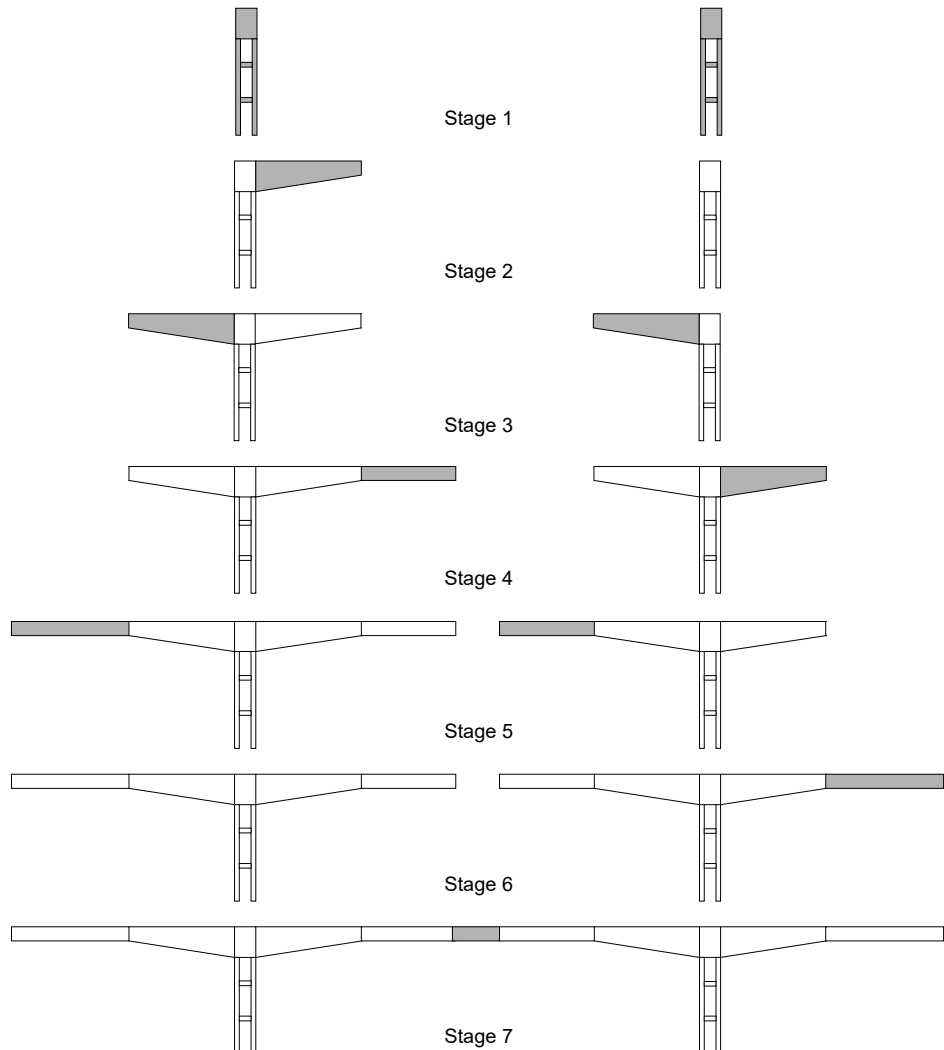


A complete analysis of a large balanced cantilever segmentally constructed bridge is a complex and large analysis to undertake. For this example, the bridge geometry has been simplified and the number of segments used in the bridge has been reduced, resulting in longer segment lengths than would otherwise be present. This has been done 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, Activate, Deactivate, Construction Table, Age, Time Management, Prestress Loading, Tendon Profile, Tendon Properties, Tendon Loading, Post Tensioned.

Associated Files

Associated files can be downloaded from the user area of the LUSAS website.



- ☐ **balanced_cantilever_bridge.lvb** creates the model geometry with all of the attributes assigned. The example will use the model built from this file as a starting point.
- ☐ **balanced_cantilever_bridge_completed.lvb** creates a model that includes all required construction stages, age and prestress loading. Time management values will need to be defined prior to solving.

Modelling



Note. Model and results images included in this example were captured in an earlier version and may differ in appearance from those seen and obtained from this version.

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

- Enter a file name of **balanced_cantilever_bridge**

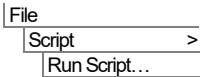
File	
New...	

Balanced Cantilever Construction of a Segmental Post Tensioned Bridge

- Use the default **User-defined** working folder.
- Ensure an Analysis category of **3D** is set.
- Click the **OK** button.



Note. There is no need to enter any other new model details when the intention is to run a script to build an initial model, since the contents of the script will overwrite any other settings made.



To create the model, open the file **balanced_cantilever_bridge.lvb** that was downloaded and placed in a folder of your choosing.

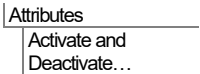


Save the model file.

The basic bridge geometry will be displayed.

Creating Activation and Deactivation Datasets

In order to carry out a staged construction analysis the activation and deactivation 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**
- Select **Follow active mesh**. Enter the attribute name as **Deactivate** and click **Apply**
- Select **Closing part** and change the name to **Deactivate closing segment** and click **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.

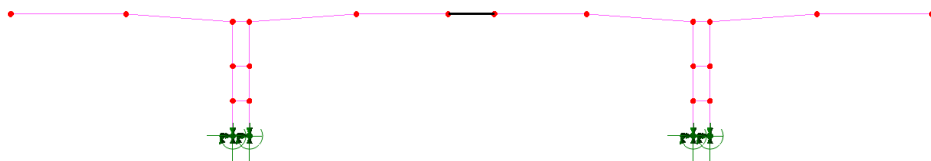
See the LUSAS online help for more information.

Modelling of Construction Stage 1

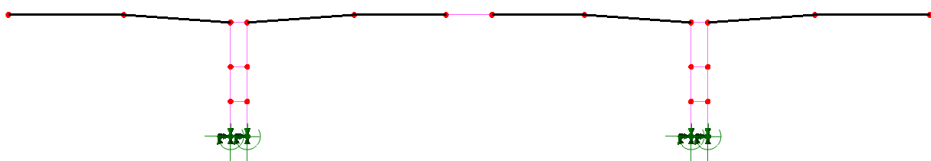
- In the  Treeview expand **Analysis 1** then rename **Loadcase 1** to be **Stage 1** by selecting Loadcase 1 with the right-hand mouse button and using the **Rename** option.

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 activation and deactivation 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.

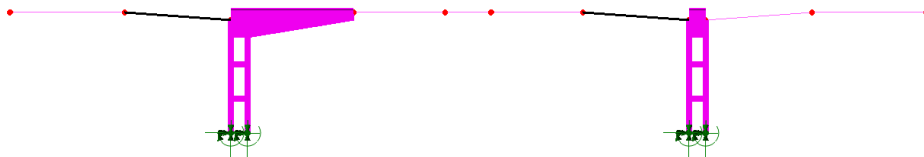


Note. If you need to verify loading assignments such as self-weight or to check when particular lines (and hence elements) become active in an analysis select a line on the model and then, using the right-hand mouse button, select **Properties**. Expand the droplist for **Activate** and select **Details**.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

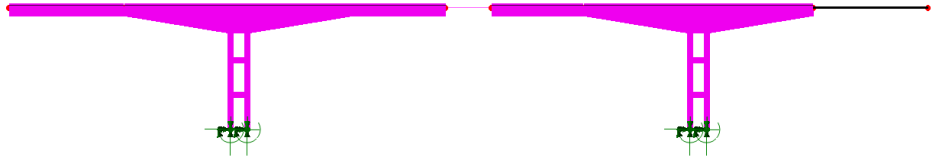
- In the view window select **lines 17 and 23**. Assign the activation attribute **Activate** and enter **Stage 5** in the loadcase combo box. Click **OK** to finish the activation of the selected section.



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.

Balanced Cantilever Construction of a Segmental Post Tensioned Bridge



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



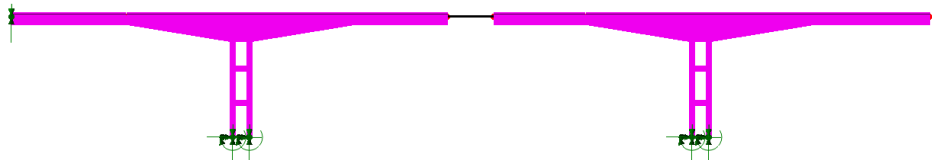
- Assign the support attribute **Roller** from the  Treeview. Choose **From loadcase** and ensure the loadcase **Stage 6** is selected. Click **OK** to assign the support.



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.

- 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 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.

- Ensure Age at activation time is set to **Specified directly** and 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 time for **Specified directly** 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)

Analyses
Loadcase...

Attributes
Age...

Balanced Cantilever Construction of a Segmental Post Tensioned Bridge

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.

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

Bridge
Time
Management...

Time Management

Analysis: Analysis 1

Start Date: 06 October 2022 (Date of the first loadcase)

Stages Dates Gantt Chart

	Loadcase	Age at Placement	Age at shrinkage	Stage Duration	Response Time
+	Stage 1	5	5	7	7
+	Stage 2	14	3	7	14
+	Stage 3	14	3	7	21
+	Stage 4	14	3	7	28
+	Stage 5	14	3	7	35
+	Stage 6	14	3	7	42
+	Stage 7	5	5	30	72
	Long term	-	-	36500	36572

Analysis type: ☒ Linear elastic ☐ Time domain

OK Cancel Apply Help

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...

Tendon Profile

3d space Two 2d planes

	Type	X (m)	Y (m)	Z (m)
1	Start	-20.0	2.75	19.7
2	Straight	0.0	2.75	21.0
3	Straight	20.0	2.75	19.7

☐ Local coordinates mapped to lines
☒ Global coordinates

☒ Smoothing
 Minimum radius: 60.0
☒ Cut corner
☐ Offset line

Name: Tendon 1 Pier 1 left web (new)

OK Cancel Apply Help

- 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**
- Enter the tendon profile into the grid as follows:

Balanced Cantilever Construction of a Segmental Post Tensioned Bridge

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.

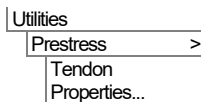
The tendon profiles are added to the Tendon Profile folder in the Utilities  Treeview.



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.



Balanced Cantilever Construction of a Segmental Post Tensioned Bridge

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

Tendon Properties

Design code: **EN1992-1-1:2004 / 2014 Eurocode**

☒ 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	5.05E3	mm ²
Modulus of elasticity for tendon	195.0E6	kN/m ²
Installation losses		
Unintentional angular displacement	0.01	rad/m
Duct friction coefficient	0.19	
Post-installation losses		
Include	Yes	
Tensile strength of prestressing steel	1.86E3	N/mm ²
Relaxation class	Class 2	
Relaxation loss at 1000hrs	2.5	%

Name: **37 No 15mm strand** (new)

OK **Cancel** **Apply** **Help**

- 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 **Prestressing tendons** 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**.

Tendon

Analysis category: 3D

Profile: 1:Tendon 1 Pier 1 left web

Property: 1:37 No 15mm strand

	Value
Prestress force	5.0E3

☒ Jacking at end 1

	Value
Angle	0.0
Slip	5.0E-3

☒ Jacking at end 2

	Value
Angle	0.0
Slip	5.0E-3

Name: Tendon 1 Pier 1 left web (1)

Buttons: Close, Cancel, Apply, Help

- Select each other Profile in turn, entering an appropriate Name and clicking **Apply** for each.

This completes defining the tendon loading attributes.

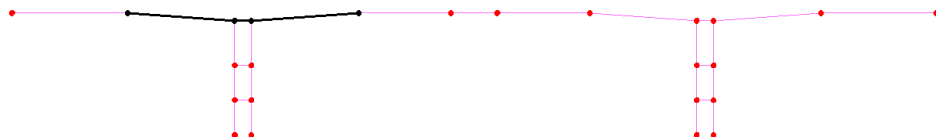


Note. Tendon loading assignments can be reviewed and corrected, if necessary, before proceeding further by clicking the up and down buttons adjacent to the tendon profile name.

- Click **Finish**

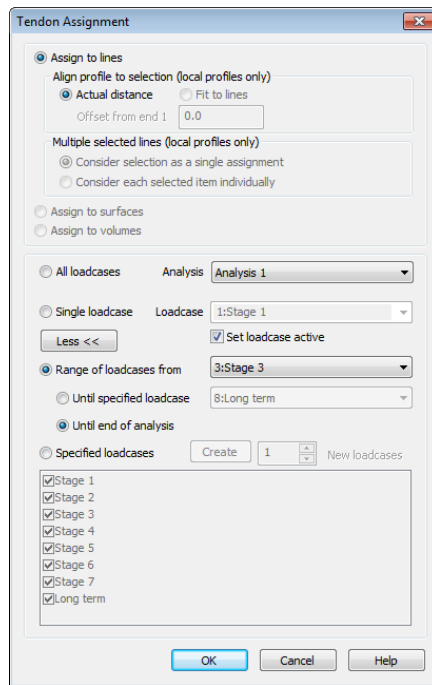
Assigning the Tendon Loading

- Select the three lines tendon 1 acts on.

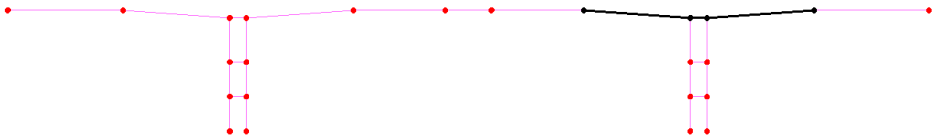


- Drag and drop **Tendon 1 Pier 1 left web** from the Treeview onto the selected lines.
- On the Tendon Assignment dialog click **“More >>”** to show all loadcase options.

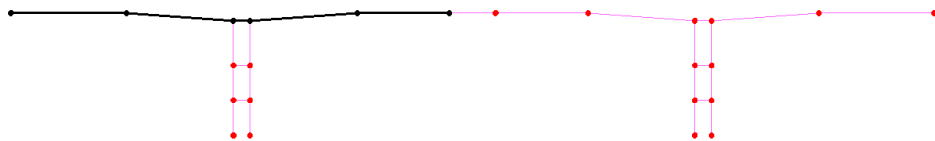
Balanced Cantilever Construction of a Segmental Post Tensioned Bridge



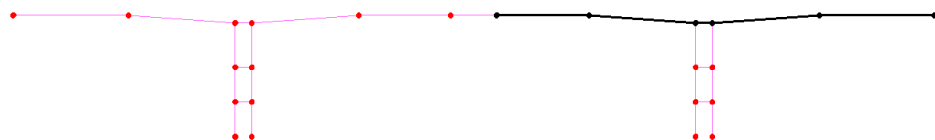
- Select **Range of loadcases from** and **Stage 3** and choose **Until end of analysis**. Click the **OK** button.
- Repeat to drag and drop **Tendon 1 Pier 1 right web** onto the same selected lines for the same loadcase settings.
- Now select the three lines tendon 2 acts on.



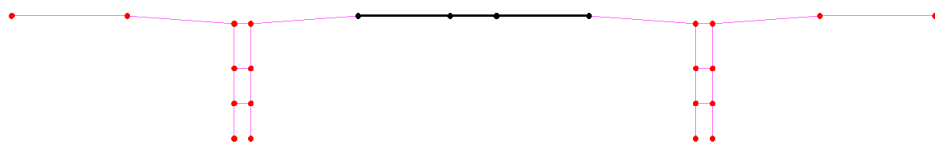
- Drag and drop **Tendon 2 Pier 2 left web** from the  Treeview onto the selected lines.
- Assign it to a **Range of loadcases** from **Stage 4** to the end of the analysis. Click the **OK** button.
- Repeat to assign **Tendon 2 Pier 2 right web** to the same lines.
- Select the five lines tendon 3 acts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

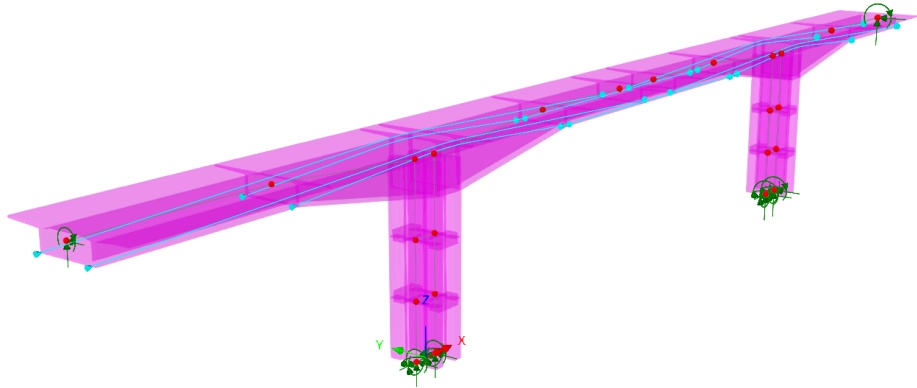


Turn the loading on if it is not already turned on.

Balanced Cantilever Construction of a Segmental Post Tensioned Bridge

- From the toolbar, select the **Fleshing all transparent**  button from the droplist provided.

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 by pressing the **Settings** button to change the visualisation settings.



Caution. Forces calculated in prestress tendons may be incorrect 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 (as in this example) 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.

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.



Caution. Whilst editing of tendon profile data is permitted, it should be done with caution.

Prestress analysis options

- In the  Treeview scroll to the bottom and double click on the  **Prestress analysis options**. Make sure **Provide primary and secondary effects** is ticked.

This enables retrospective calculation by LUSAS of the primary and secondary effects, for which additional tendon loss calculation options are also available but not examined in this example.

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 the nested options of **Compute prestress losses** and **Main analysis** 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.

If the analysis is successful...

Analysis loadcase results are added to the Analyses  Treeview.

In addition, these files will be created in the LUSASFiles\<model_name> folder:



- ☐ **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. Any errors listed in the

Balanced Cantilever Construction of a Segmental Post Tensioned Bridge

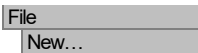
text output window should be corrected in LUSAS Modeller before saving the model and re-running the analysis.

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. You may download this file from the user area of the LUSAS website.



- ☐ **balanced_cantilever_bridge_completed.lvb** This creates a model that includes all required construction stages, age and prestress loading. Time management values will need to be defined prior to solving.

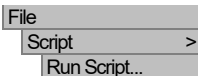


Start a new model file. If an existing model is open Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **balanced_cantilever_bridge**
- Use the default **User-defined** working folder.
- Ensure an Analysis category of **2D Inplane** is set.
- Click the **OK** button.



Note. There is no need to enter any other new model details when a script is run to build a model, since the contents of the script will overwrite any other settings made.

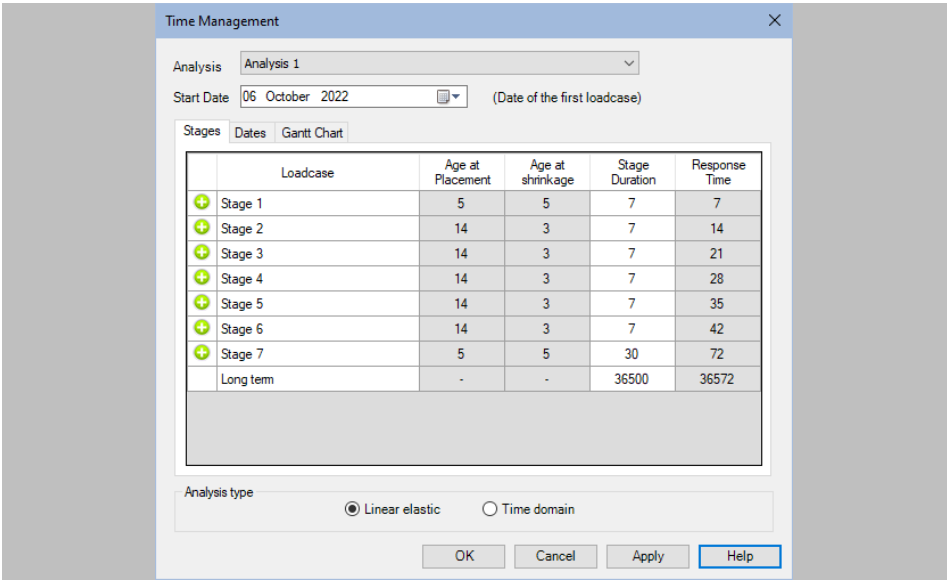


To recreate the model, select the file **balanced_cantilever_bridge_completed.lvb** that was downloaded and placed in a folder of your choosing.

Specify time management values

Time management values (stage durations) are not provided by the model recovery script. Construction 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.

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



Solve the model



Open the Solve Now dialog. Ensure **Analysis 1** 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.

Viewing the Results

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

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 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 loading on/off button.

Generating a report showing Prestress Details

- Enter the Title as **Balanced Cantilever Bridge**, change the name to **Prestress Report** and click OK

Utilities

Reporting >

New Report...

Balanced Cantilever Construction of a Segmental Post Tensioned Bridge

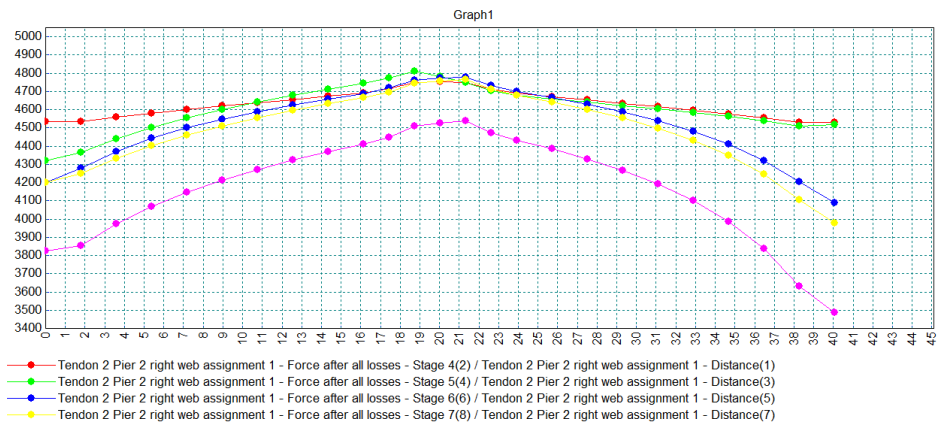
- In the Reports  Treeview right-click on **Prestress Report** and choose **Add Chapter**.
- Select the **Prestress** tab and tick the check boxes for all tendon assignments that are present to include data for these in the report and press **OK**.
- In the Reports  Treeview, right-click on **Prestress 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 a Tendon

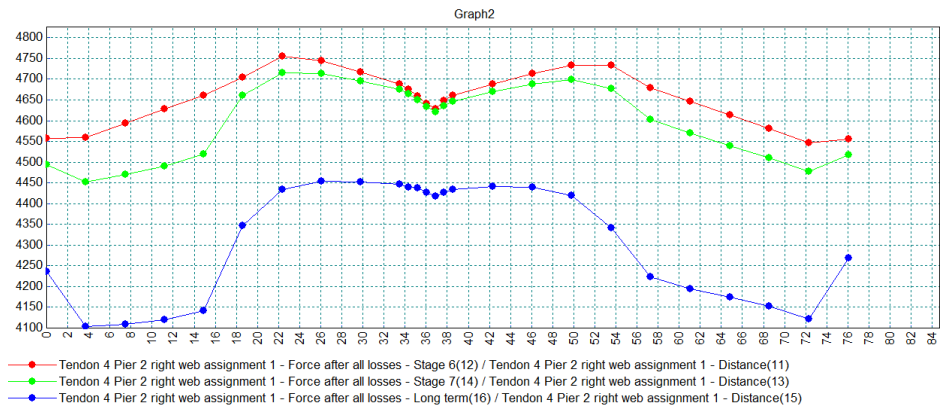
Add the graph of the Force for Tendon 2 Pier 2 right web.

- 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:

- 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.



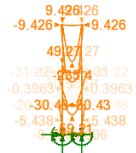
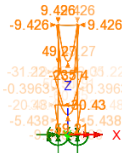
Close the graph windows.

Stage 1 results

- In the Treeview, and within **Stage 1**, right-click on loadcase **1: Total effects** and select the **Set Active** option.
- Turn off the display of the **Mesh**, **Geometry** and **Attributes** layers from the Treeview.
- If not there already, add the **Deformed mesh** layer to the Treeview, and click **OK**.
- Using the **Deformations...** button at the bottom of the Treeview, specify a factor of **100**. Click the **Window summary** option to add a summary to the view window.
- Add the **Diagrams** layer to the Treeview. On the dialog, select **Force/Moment** results from the entity drop-down list and component **My**. Select the **Diagram Display** tab.
- Ensure the **Label values** and the **Peaks only** options are selected.
- Click **OK** to display the bending moment diagram for the active elements for Stage 1.

Balanced Cantilever Construction of a Segmental Post Tensioned Bridge

Scale: 1: 543.755
Zoom: 100.0
Eye: (0.0, -1.0, 0.0)
Nonlinear analysis
Analysis: Analysis 1
Loadcase: 1: Stage 1, 1: Total effects
Results file: balanced_cantilever_pb-Analysis 1.mys
Maximum displacement 0.153985E-3 at Gauss point 6 of element 25
Deformation exaggeration: 16.8445E3
Diagram entity: Force/Moment - Thick 3D Beam
Diagram component: My (Units: kN.m)
Diagram maximum 30.9097 at node 9 of element 9
Diagram minimum -235.41 at internal point 6 of element 25
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.

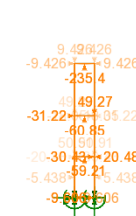
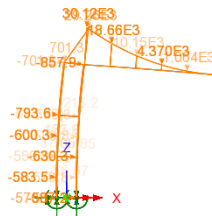


Note. Results for **Primary effects** and **Secondary effects** are also available, as separate results loadsets. These are not investigated here.

Stage 2 results

- In the  Treeview, click the 'Plus' button  at the bottom to advance to **Stage 2Total effects**.

Scale: 1: 543.755
Zoom: 100.0
Eye: (0.0, -1.0, 0.0)
Nonlinear analysis
Analysis: Analysis 1
Loadcase: 2: Stage 2, 2: Total effects
Results file: balanced_cantilever_pb-Analysis 1.mys
Maximum displacement 0.030233 at node 47
Deformation exaggeration: 107.913
Diagram entity: Force/Moment - Thick 3D Beam
Diagram component: My (Units: kN.m)
Diagram maximum 30.1204E3 at node 22 of element 28
Diagram minimum -857.979 at node 22 of element 22
Diagram scale: 1: 0.332001E-3

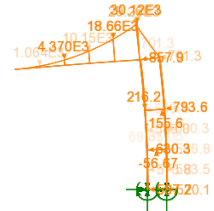
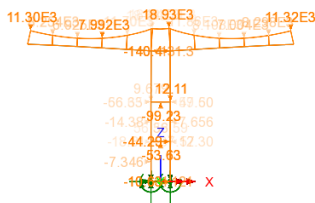


Note. Clicking the 'Plus' button  will advance to the next of the same results type, i.e. as **Total effects** was active in **Stage 1**, it sets active **Total effects** in **Stage 2**, rather than **Primary effects** or **Secondary effects** from **Stage 1**.

Stage 3 results

- In the  Treeview, click the 'Plus' button  at the bottom to advance to **Stage 3 Total effects**.

Scale: 1: 543.755
Zoom: 100.0
Eye: (0.0, -1.0, 0.0)
Nonlinear analysis
Analysis: Analysis 1
Loadcase: 3 Stage 3, 3-Total effects
Results file: balanced_cantilever_pb-Analysis 1.mys
Maximum displacement 0.030233 at node 57
Deformation exaggeration: 107.913
Diagram entity: Force/Moment - Thick 3D Beam
Diagram component: My (Units: kN.m)
Diagram maximum 30.1204E3 at node 23 of element 29
Diagram minimum -155.559 at internal point 10 of element 20
Diagram scale: 1: 0.332001E-3



Long term results

- In the  Treeview right-click on loadcase **8: Total effects** under **Long term** and select the **Set Active** option.

Scale: 1: 543.755
Zoom: 100.0
Eye: (0.0, -1.0, 0.0)
Nonlinear analysis
Analysis: Analysis 1
Loadcase: 8 Long term, 8-Total effects
Results file: balanced_cantilever_pb-Analysis 1.mys
Maximum displacement 0.126995 at node 41
Deformation exaggeration: 25.6902
Diagram entity: Force/Moment - Thick 3D Beam
Diagram component: My (Units: kN.m)
Diagram maximum 90.735E3 at node 21 of element 27
Diagram minimum -4.00378E3 at node 52 of element 56
Diagram scale: 1: 0.110206E-3



Maximum and minimum results

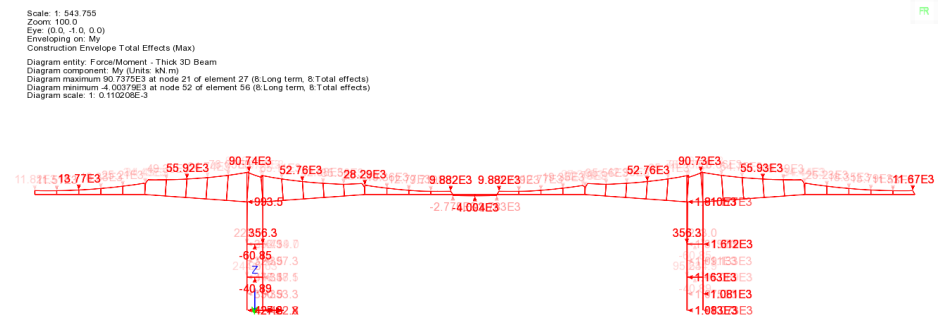
Analyses
Envelope...

- Type 3 in the Step box, then left click **Analysis 1** in the Available treeview, and press the 'Add to'  button to include every third results loadset (i.e. the **Total effects** ones) in the envelope. Give the name of **Construction Envelope Total Effects** and click OK.

Balanced Cantilever Construction of a Segmental Post Tensioned Bridge

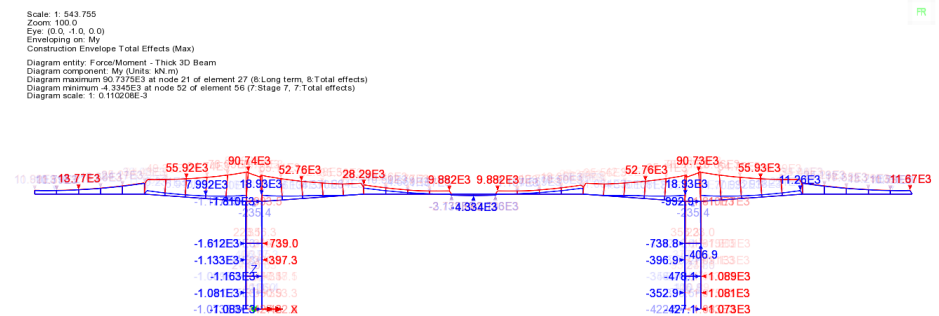
- In the  Treeview right-click on **Construction Envelope Total Effects (Max)** and select the **Set Active** option.
- Ensure that the Entity **Force/Moment** is selected and component **My**. Click **OK**.
- If a message pops up stating that the deformed mesh will no longer be available, click **OK**.

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**.



Note. Viewing of contour results is best done using a fleshed view of the model.

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 > Concrete (Nonlinear)** menu item). In addition, the selection of 'Time Domain' from the Time management dialog (that can be accessed from the context menu for an analysis entry in the Analyses Treeview) would also be required. This will apply the required nonlinear controls to the staged construction loadcases to carry out the transient analysis.

