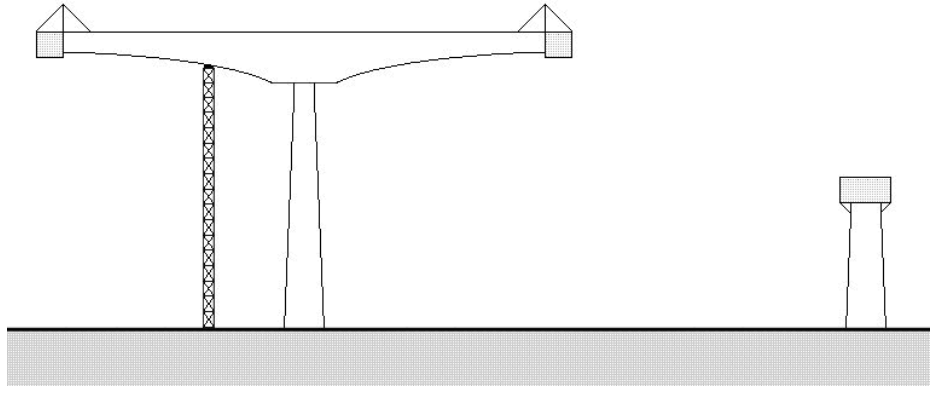
Segmental Construction of a Post Tensioned Bridge

For LUSAS version:	16.0
For software product(s):	LUSAS *Civil & Structural Plus* and LUSAS *Bridge Plus*
With product option(s):	Nonlinear

Description

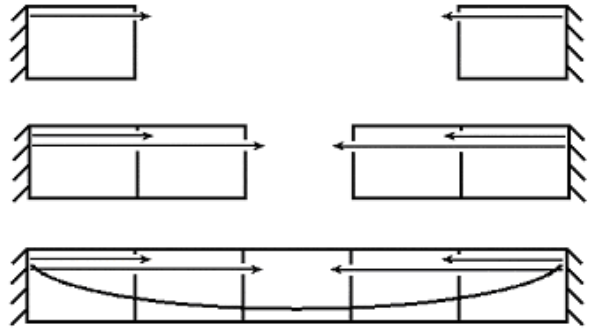
The construction of a balanced cantilever segmentally constructed bridge is to be modelled using a beam analysis.



A complete analysis of a 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 properties.

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)

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

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

Objectives

The output requirements of the analysis are:

- Maximum moments during construction

Keywords

3D, Beam, Staged Construction, Age, Time Management, Multiple Tendon Prestress Wizard, Post tensioned.

Associated Files

- **segmental_bridge_modelling.vbs** carries out the modelling of the structure up to the section titled **Prestess 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…

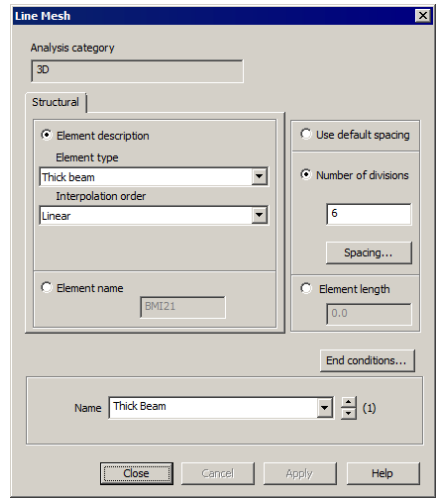

Enter a translation in the **X** direction of **6**

- 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.



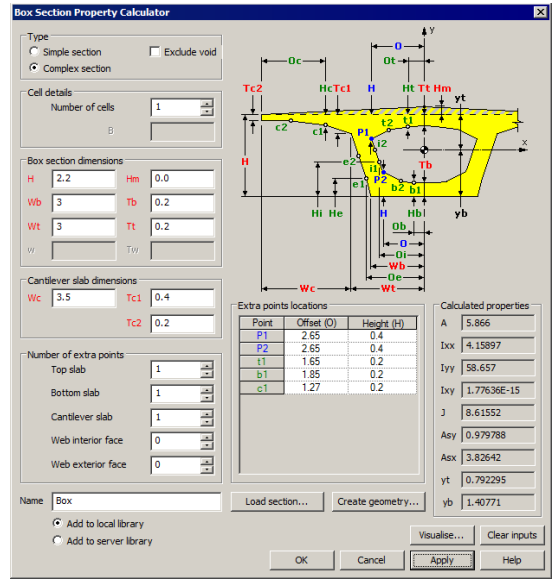
Defining and Assigning Geometric Properties

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…

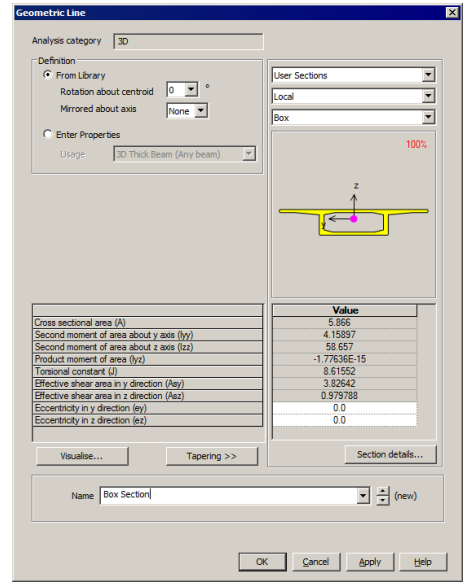
The box section property calculator will be used to define the box section required.

- 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. Where the prestress wizard is used to consider time dependent effects (as this example does) 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 **EN1992-1-1:2014** 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.

- Select all lines on the model and 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 wizard to calculate time dependent effects. Since we will consider these effects in this example we must use 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 analysis by right clicking the analysis and selecting the Add gravity menu item. This latter method when all loadcases are defined.

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

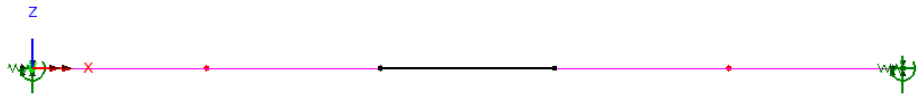
- In the Treeview expand **Analysis 1** then rename **Loadcase 1** to be **Stage 1** by selecting Loadcase 1 with the right-hand mouse button and using the **Rename** option.

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.

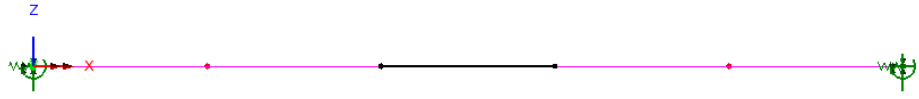
- Assign the activation attribute **Activate** from the Treeview. Enter **Stage 2** in the loadcase combo box and ensure that **Set as the active loadcase** is checked. Click **OK** to finish activation of the selected sections.

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. Select the **Activate Elements** tab. Selecting the **Activate** entry in the right-hand table of Assigned properties will show the loadcase to which the activation is assigned.

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 activated only** 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.

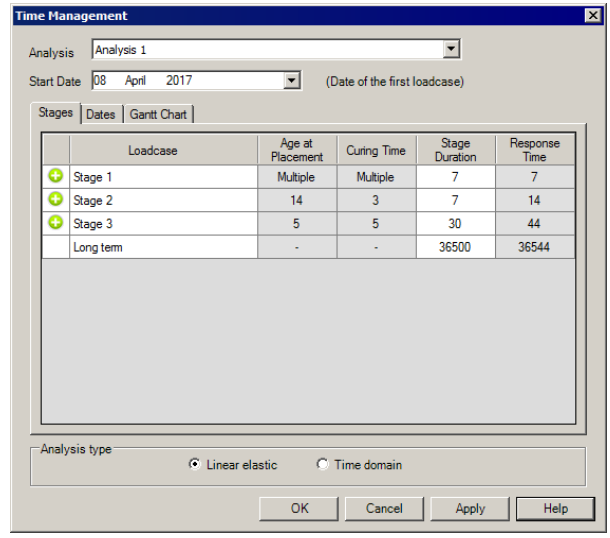
Time management

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

The calculation and assignment of the equivalent prestress force to the loadcases already created is carried out by using the multiple tendon prestress wizard. It requires the following to be defined:

1. **Design code**
2. **Tendon profiles**
3. **Tendon properties**
4. **Tendon loadings**
5. **Tendon loading assignments**

Once entered, and once the multiple tendon prestress wizard has been run, values for each of these entries are retained and can be viewed and subsequently modified by re-running the wizard. Previously calculated equivalent loadings are automatically re-calculated.

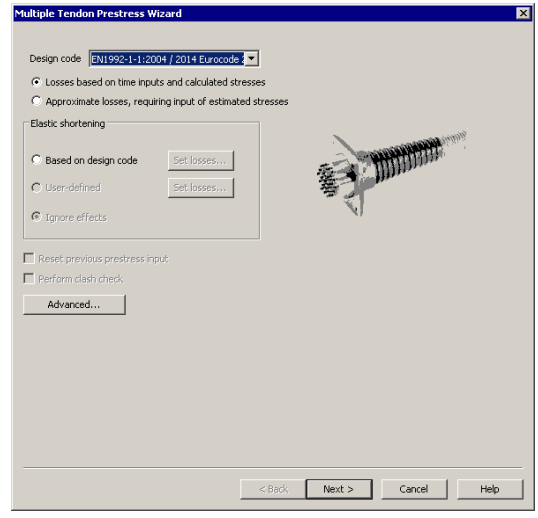
Using the Multiple Tendon Prestress Wizard

Lines on the model will be selected during the use of the wizard, so:

- Ensure that the **Geometry** layer is present in the Treeview.

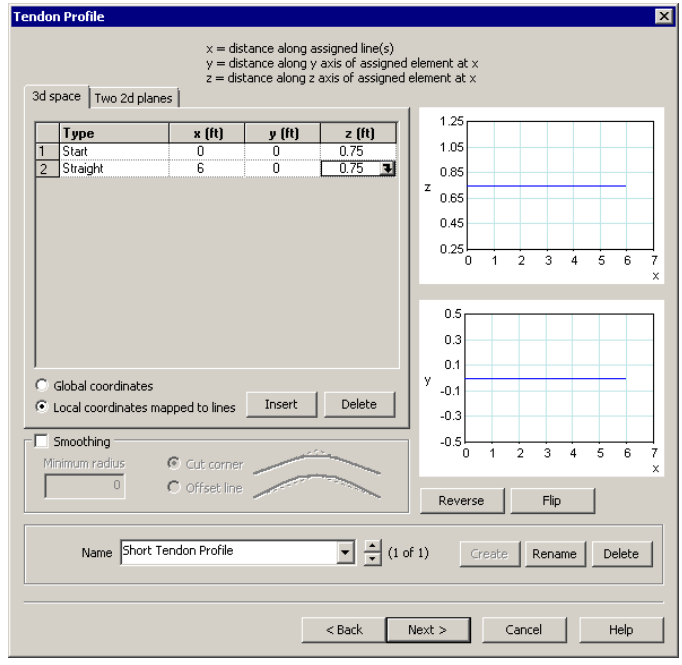
Bridge
Prestress Wizard >
Multiple Tendon...

- 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 and click **Next**



Defining the Tendon Profiles

Any number of tendon profiles can be defined on the tendon profile page prior to moving on to the next page of the Multiple Tendon Wizard. By entering a name the Create button becomes active and tendon details can be defined. Entering another name stores the previously defined profile and enables another set of data to be entered, and so on.



- Enter the attribute name as **Short Tendon Profile** and click the **Create** button.
- Ensure the option **Local coordinates mapped to lines** is selected
- Enter the tendon profile into the grid as

Type	X (m)	Y (m)	Z (m)
Start	0	0	0.75
Straight	6	0	0.75

This completes the short tendon profile definition.

- Now, to start entering another tendon profile, enter the name as **Long Tendon Profile** and click the **Create** button. In doing so, this stores the previous short tendon profile values and allows existing values to be edited as appropriate for the next profile to be defined.

- 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

This completes the long tendon profile definition.

- Lastly, enter the attribute name as **Full Tendon Profile Left** and click the **Create** button.
- 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.
- Enter the attribute name as **Full Tendon Profile Right** and click the **Create** button.
- 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

This completes the long tendon profile definitions

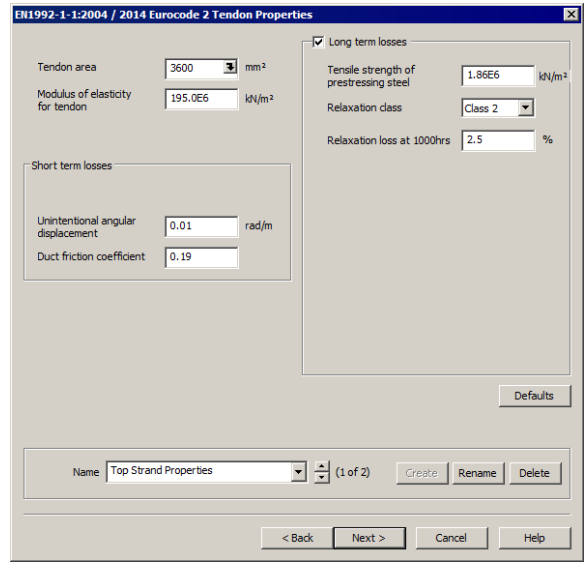
Note. Tendon profile definitions can be reviewed before proceeding further by clicking the up and down buttons adjacent to the tendon profile name.

- Click **Next** to proceed to the Tendon Properties dialog.

Defining the 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** and click the **Create** button
- Enter the tendon area as **3600**
- Change the name to **Web Strand Properties** and press create
- Enter the tendon area as **1800**
- Accept the defaults for the remaining properties by clicking the **Next** button.

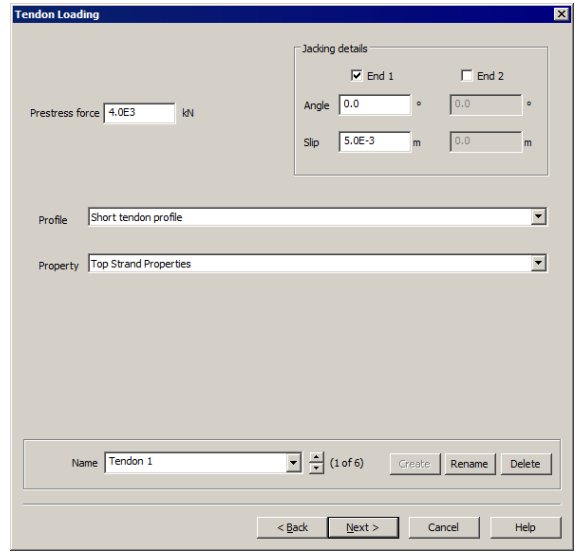


Defining the Tendon Loading

This page allows a tendon load to be associated with previously entered profiles and properties.

For tendon 1:

- Enter the attribute name as **Tendon 1** and click the **Create** button.
- Enter the prestress force as **4E3**
- Ensure that jacking from **End 1** is the only jacking detail selected and enter the slip as **5E-3**
- Ensure the profile combo box is set to **Short tendon profile** and the property combo box is set to **Top Strand Properties**



This completes the tendon loading for tendon 1.

For tendon 2:

- Enter the attribute name as **Tendon 2** and click the **Create** button. In doing so, this stores the previous short tendon profile values and allows existing values to be edited as appropriate for the next profile to be defined.
- Enter the prestress force as **4E3**
- Ensure that jacking from **End 2** is the only jacking detail selected and enter the slip as **5E-3**
- Ensure the profile combo box is set to **Short tendon profile** and the property combo box is set to **Top Strand Properties**

This completes the tendon loading for tendon 2.

For tendon 3:

- Enter the attribute name as **Tendon 3** and click the **Create** button.
- Enter the prestress force as **4E3**
- Ensure that the jacking from **End 1** is only selected and enter the slip as **5E-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.

This completes the tendon loading for tendon 3.

For tendon 4:

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

This completes the tendon loading for tendon 4.

For tendon 5:

- Enter the attribute name as **Tendon 5** and click the **Create** button.
- 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.
- Change the profile combo box so that **Full Tendon Profile Left** is displayed and ensure the property combo box is set to **Web Strand Properties**

This completes the tendon loading for tendon 5.

For tendon 6:

- Enter the attribute name as **Tendon 6** and click the **Create** button.
- 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.
- Change the profile combo box so that **Full Tendon Profile Right** and the property combo box is set to **Web Strand Properties**

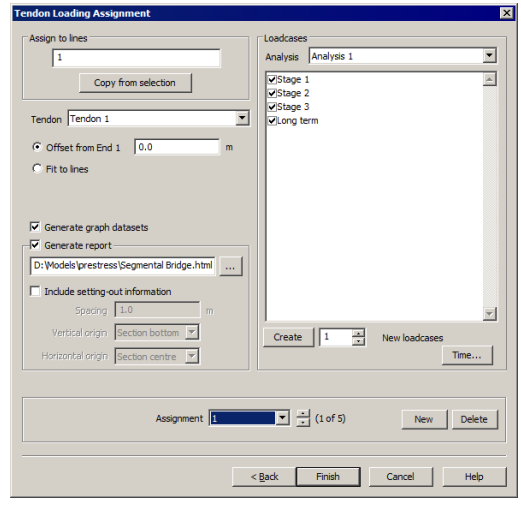
This completes the tendon loading for tendon 6.

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 **Next** to proceed to the Tendon Loading Assignment dialog.

Defining the Tendon Loading Assignment(s)

The page of the wizard is used to assign the tendon loads that have been defined in the previous dialog to selected lines on the model.



- Click the **New** button and the Assignment will change to 1
- Drag the dialog so that the model can be clearly seen in the view window

- Select the line tendon 1 acts on.

- On the Tendon Loading Assignment dialog click the **Copy from selection** button and the selected line number will be entered in the text box.
- Ensure the tendon combo box has **Tendon 1** displayed.
- In the loadcases section of the dialog, ensure that the check boxes for **Stage 1, Stage 2, Stage 3** and **Long term** are selected.
- Ensure that the **Generate graph datasets** and **Generate report** options are selected.

This completes the tendon assignment for line 1.

Note. Clicking on a loadcase higher than another will select or deselect all loadcases beneath it.

- Click the **New** button and the Assignment will change to 2.
- Select the line tendon 2 acts on.

- Click the **Copy from selection** button and the selected line number will be entered in the text box.
- Change the tendon combo box so that **Tendon 2** is displayed.
- In the loadcases section of the dialog, ensure that **Stage 1, Stage 2, Stage 3** and **Long term** are selected
- Click the **New** button and the Assignment will change to 3.
- Select the two lines tendon 3 acts on.

- Click the **Copy from selection** button and the selected line numbers will be entered in the text box.
- Change the tendon combo box so that **Tendon 3** is displayed.
- In the loadcases section of the dialog, deselect Stage 1 and ensure that only the check boxes for **Stage 2, Stage 3** and **Long term** are selected
- Click **New** and the Assignment will change to 4.
- Select the two lines tendon 4 acts on.

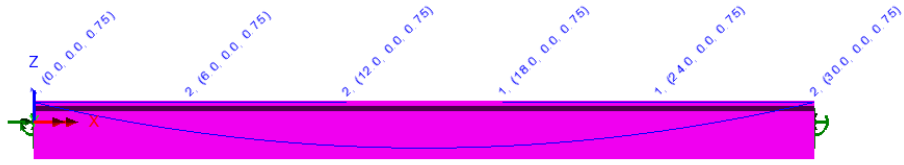
- Click the **Copy from selection** button and the selected line numbers will be displayed in the text box.
- Change the tendon combo box so that **Tendon 4** is displayed.
- In the loadcases section of the dialog, ensure that only **Stage 2, Stage 3** and **Long term** are selected
- Click **New** and the Assignment will change to 5.

- Select the lines tendon 5 acts on. (All the lines in the model.)
- Click the **Copy from selection** button and the selected line numbers will be displayed in the text box.
- Change the tendon combo box so that **Tendon 5** is displayed.
- In the loadcases section of the dialog, deselect Stage 2 and ensure that only the check boxes for **Stage 3** and **Long term** are selected.
- Click **New** and the Assignment will change to 6.
- Ensure all lines in the model are still selected
- Click the **Copy from selection** button and the selected line numbers will be displayed in the text box.
- Change the tendon combo box so that **Tendon 6** is displayed.
- In the loadcases section of the dialog, ensure that only the check boxes for **Stage 3** and **Long term** are selected.
- Select the **Finish** button to complete the wizard. A message indicates that the prestress loading will be calculated when the model is next solved.

Note. Graph datasets will be created in the Treeview for graphing if required. A report will be generated in the directory that you are currently working in but this can be changed if required.

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 and the display of start and finish coordinates can be controlled by accessing the **Utilities** layer Properties dialog and on the Tendon Profile tab pressing the **Settings** button to change the visualisation setting to **Show definition coordinates**. Visualisations will then be identified by their start and finish identifiers (1 or 2) and by the coordinates of these points. Because all tendons are active in loadcase 3 the visualisations will overlap.



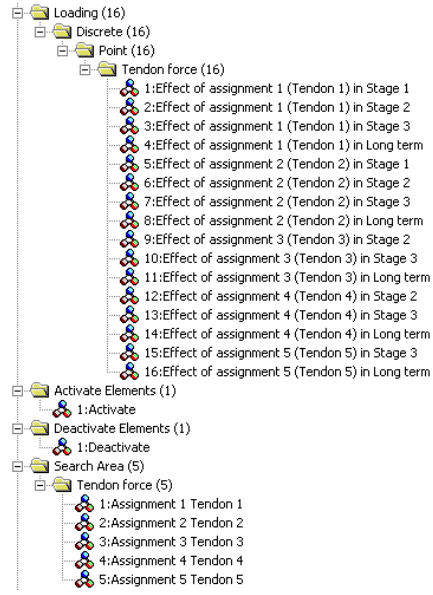
Notes on Prestress Wizard-created data

In using the Prestress Wizard, prestress loading is added to the analysis model as equivalent discrete point loading. Attributes containing this data can be seen in the Treeview in the Tendon force folder.

Since each loadcase represents a stage of the construction and therefore a period of time, prestress loading is different for each loadcase.

Search areas are created and used automatically by the prestress wizard to define the target geometry (in this case, the lines) to be loaded. These can be seen in the Treeview.

Graphing datasets (which allow graphing of prestress losses etc) are added to the Treeview.



Notes on editing Prestress Wizard-created data

If any tendon assignments are incorrect, the Multiple tendon prestress wizard must be re-run (previously entered values will be retained) and the Next button can be clicked to get to the tendon loading assignment page. Incorrect assignments can be deleted or simply corrected as required before clicking the Finish button to update the prestress loading on the next solve.

Tendon profile data defined within the Prestress Wizard is also stored in the Treeview.

- Editing of tendon profile data in the Treeview (that is, outside of the wizard) is permitted but all attributes that require re-calculating by re-running the prestress wizard will be marked with the symbol. Re-running the prestress wizard will update all values accordingly.
- Editing of any attribute data calculated and generated by the prestress wizard is not permitted.

Save the model

File
Save

 Saving the model file also saves all the multiple tendon prestress input. This allows the prestress definition and assignment data to be modified by re-entering the multiple tendon prestress wizard.

Running the Analysis

With the model loaded:

 Open the **Solve now** dialog. Ensure **Analysis 1** is selected and press **OK** to run the analysis.

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

If the analysis is successful...

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 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 editing the tendon profile properties of the Utilities layer in the Layers Treeview.

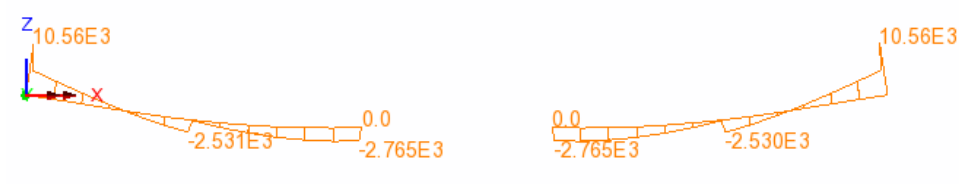
Stage 1 results

- 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 are 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.

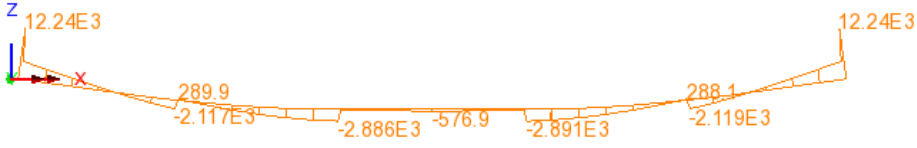
Stage 2 results

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



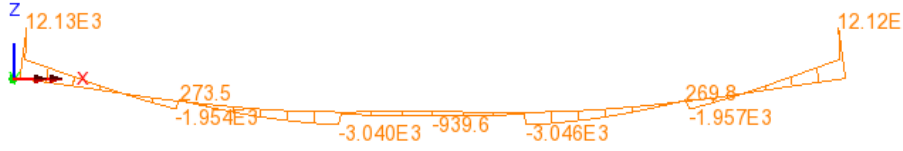
Stage 3 results

- 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. Examination of the load attribute **Effect of assignment 1 (Tendon 1) in Stage 1** from the  Treeview shows that the tendon loading starts at 3e-3m from the element end, hence the difference between the support moment and the beam moment.

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.

In this example a linear analysis was carried in which the time dependent effects are only considered 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, in the Time management dialog, selecting “Time Domain” from the analysis type options. would be required. This will apply the required nonlinear controls to the staged construction loadcases to carry out the transient analysis.