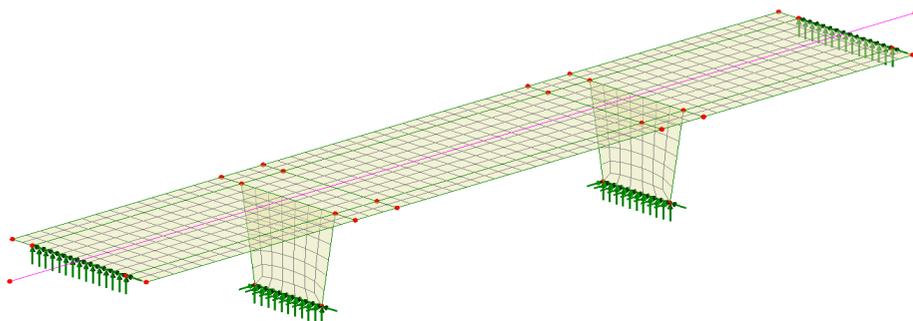


Joint Modelling for an RC Bridge with a Drop-in Span

For LUSAS version:	21.0
For software product(s):	LUSAS <i>Civil & Structural</i> or LUSAS <i>Bridge</i> .
With product option(s):	None

Description

A 3-span concrete bridge deck with a middle drop-in span is supported via bearings on tapered piers. The example shows how to define and introduce joints to represent the bearing support for the drop-in section, and the bearings required at the top of the piers.



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

Objective

- To carry out design checks to a selected design code and confirm suitability for steel member sizes.

Keywords

3D, Deck, Pier, Bearings, Joints

Associated Files

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



- ❑ **drop_in_span_bridge_preliminary.lvb** creates the basic model geometry. The example uses the model created by the file as a starting point.

Part 1: Modelling a joint-free model

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

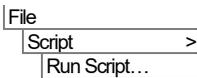
- Enter a file name of **drop_in_span_bridge**
- Use the default **User-defined** working folder.
- Ensure an Analysis type of **3D** is set.
- Click the **OK** button.

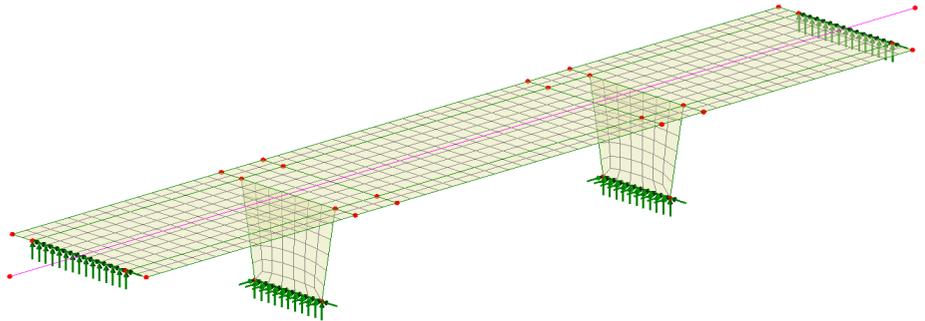


Note. There is no need to enter any other new model details when the intention is to run a script to build an initial model, since the contents of the script will overwrite any other settings made.



To create the model required for this example, open the script file **drop_in_span_bridge_preliminary.lvb** that was downloaded and placed in a folder of your choosing.





Groups have been defined for the features in the deck and the piers. These are required to restrict the extent of data used to create graphs of results along the deck. The line along the centre of the bridge is used when graphing results.

Save the model



Save the model file.

Running the analysis for the joint-free model

With the model loaded:



Select the **Solve Now** button from the toolbar and click **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

Addition files will be created in the directory where the model file resides, including:



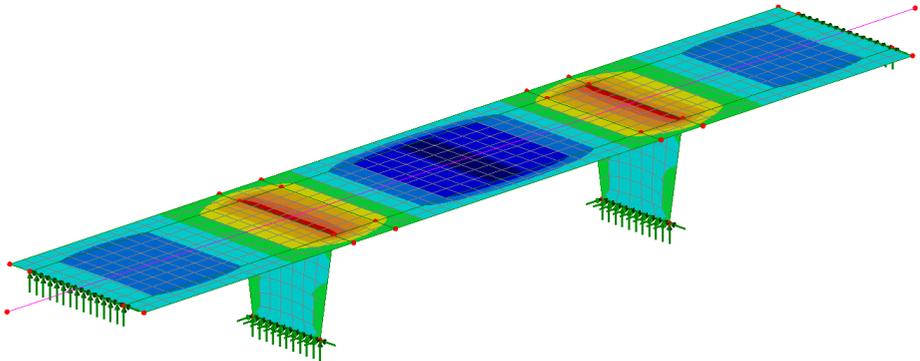
- Drop_in_span_bridge.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- Drop_in_span_bridge.mys** this is the LUSAS results file which is loaded automatically into the  Treeview to allow results processing to take place.

Viewing the results for the joint-free model

Analysis loadcase results are present in the  Treeview, and loadcase **Dead load** will be automatically set active.

Joint Modelling for an RC Bridge with a Drop-in Span

- With no features selected, click the right-hand mouse button in a blank part of the view window and select the **Contours** option to add the contours layer to the  Treeview.
- Select entity **Force/Moment** and component **Mx** and click **OK**.



This shows the moments present in the deck and piers when no joints have been assigned to the model.

Graphing of moment along deck

A graph of bending moment M_x along the deck is to be produced. Graphing of data along a selected line is to be used in this example. Other methods are available.

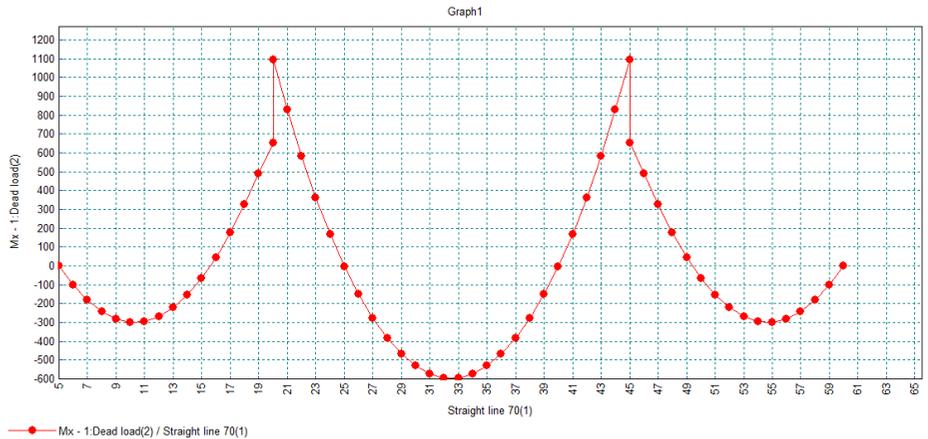
- First select the button at the bottom of the interface to view the model along the z-axis (a plan view)
- Then **select the line** running along the centre of the bridge.
- Select the option **By selected line** and click **OK**.
- Define a graph for the **Active** loadcase and select **Deck** from the Extent droplist to limit the graphed data to that set of elements and click **Next**.



Note. The option ‘Visible model’ cannot be chosen here because the slices that are taken along the line running along the centre of the bridge would also include elements that were in the piers, and an incorrect diagram would result.

- Select entity **Force/Moment** and component **Mx** and press **Next**.
- Enter a title of **Mx along centreline** and click **Finish**.

Utilities
Graph Through
2D...

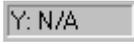


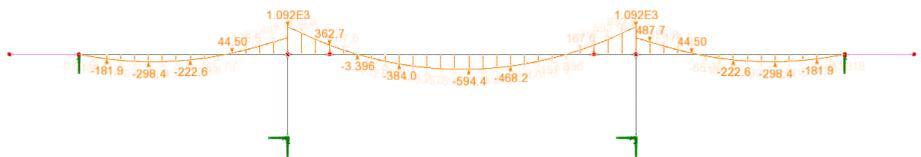
By hovering the cursor over graph points of interest it can be seen that the peak moment at the internal supports is 1092kNm and the lesser moment is 654kNm – meaning that the pier is currently attracting a moment of $1092 - 654 = 438$ kNm.

Once a graph has been created, diagrams of graph locations can also be produced. This has the benefit of seeing the results overlaid on the model.

- Close the graph window.

Viewing a bending moment diagram

- With no features selected, click the right-hand mouse button in a blank part of the view window and select the **Diagrams** option to add the diagrams layer to the  Treeview.
- Select entity **Force/Moment** and component **Mx** and click **OK**.
- Select the  button at the bottom of the interface to view the model along the y-axis (from the side).



-  Use the dynamic zoom button to enlarge the left-hand pier and the ends of the adjacent spans.

Joint Modelling for an RC Bridge with a Drop-in Span



From the updated diagram the previously described bending moments at the support can be seen, and the moment at the intended drop-in slab bearing location is in the order of 363kNm.

- Turn off the display of the **Contours** and **Diagrams** layers.
- Return the model to an appropriate isometric style of view.

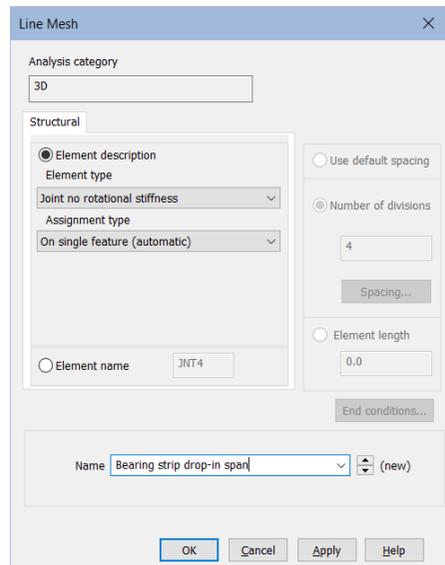
Part 2: Modelling the bearings (joints) for a drop-in span

Defining the joints for the drop-in span

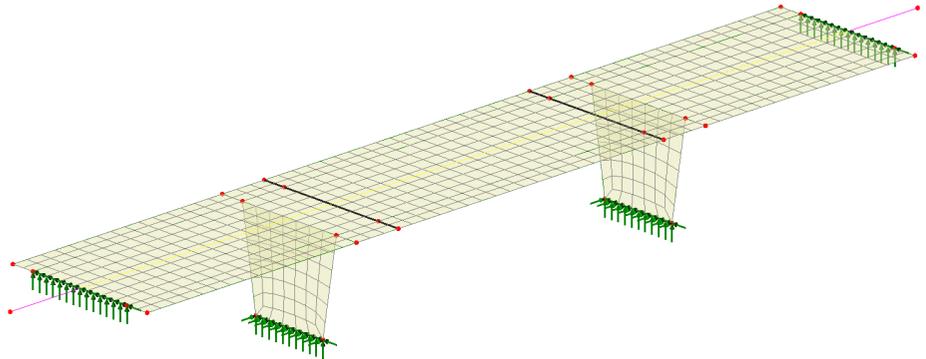
Define line mesh

To model the bearing strips for the drop-in span, a joint with no rotational stiffness is used.

- Select element type **Joint no rotational stiffness**
- Ensure assignment type is **On single feature (automatic)**
- Enter a name of **Bearing strip drop-in span** and click **OK**.



- In the model view window select the **6 lines** that define the extent of the drop-in span.



Assign mesh to model

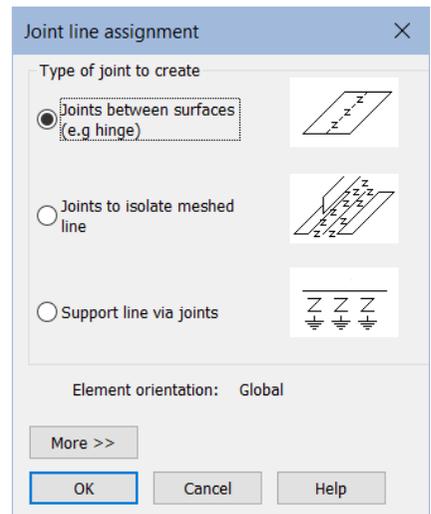
- From the  Treeview drag and drop **Bearing strip drop-in span** onto the selected features.
- Leave the type of joint to create as the default **Joints between surfaces (e.g. hinge)** and click **OK**.



Note. Symbols representing the inserted joints will not be seen immediately because the mesh is locked. This will be explained after defining and assigning joint material to the currently selected lines.



Note. Because results were previously obtained and the model has now been edited, the View window will carry the wording “Modification Mode” and “Results are out of date” until the model is solved again.



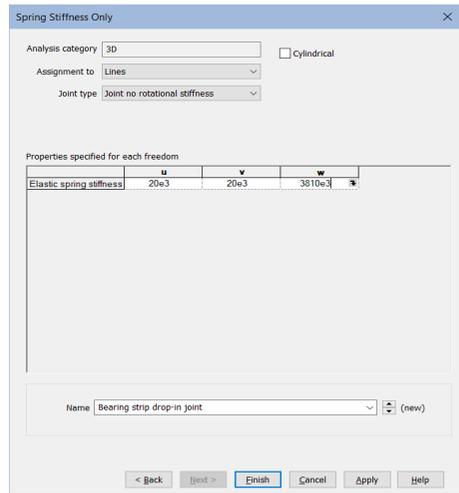
Define drop-in joint material

- On the Joint material dialog select **Spring Stiffness Only** and click **Next**.

Attributes
Material
Joint...

Joint Modelling for an RC Bridge with a Drop-in Span

- On the Spring stiffness dialog define the assignment as being for **Lines**
- Select joint type **Joint no rotational stiffness**
- Enter a value of **20e3** for the u and v freedoms and **3810e3** for the w freedom.
- Enter the name as **Bearing strip drop-in joint** and click **Finish**.



Assign material to model

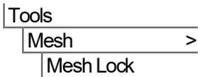
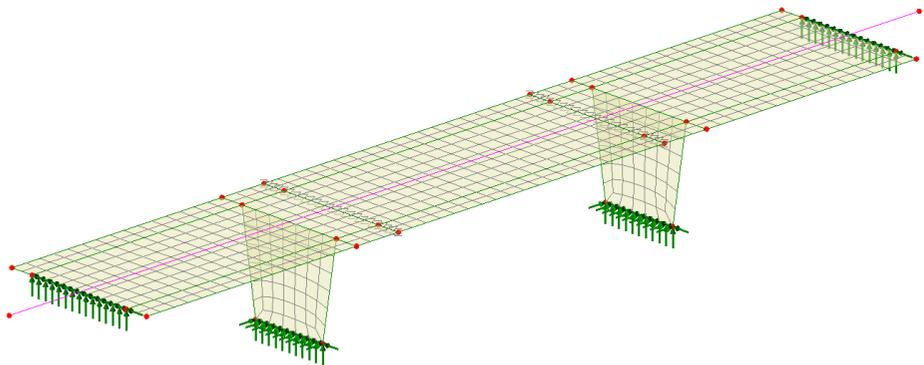
- With the lines representing the joint locations still selected, and from the  Treeview, drag and drop the joint material **Bearing strip drop-in joint** onto the selected features, clicking **OK** to assign to lines.



Note. Because results are currently available for the model prior to these joints being applied the model's mesh has been locked, so this needs to be unlocked to cause the mesh to be re-created to include the joints. In doing so, the results file for the previous model will need to be closed.

This unlocks the currently locked mesh and forces a re-mesh of the model, showing the joints inserted at the bearing locations.

- Click **OK** to close the results files.



To see the effect of this change the model will be re-solved.

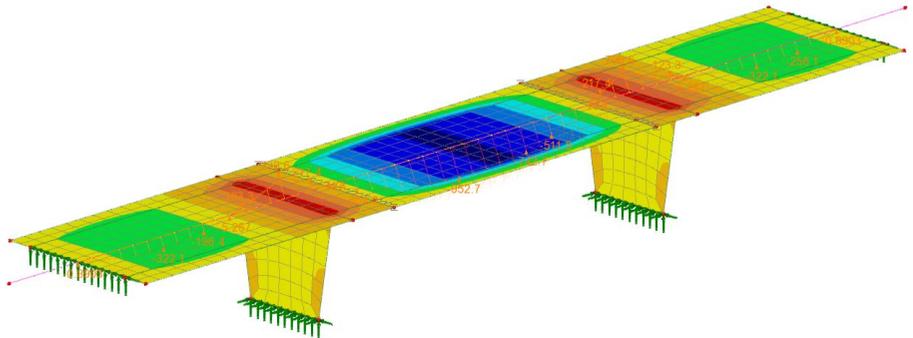
Running the analysis with drop-in span joints inserted



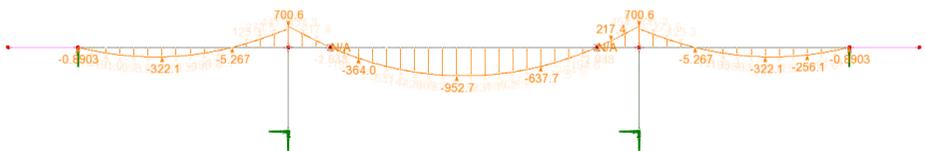
Select the **Solve Now** button from the toolbar and click **OK** to run the analysis.

Viewing the results with drop-in span joints inserted

- In the Treeview turn on the display of the **Contours** and **Diagrams** layers to see the results for entity **Force/Moment Thick Shell** and results component **Mx**.



- Select the button at the bottom of the interface to view the model along the y-axis.

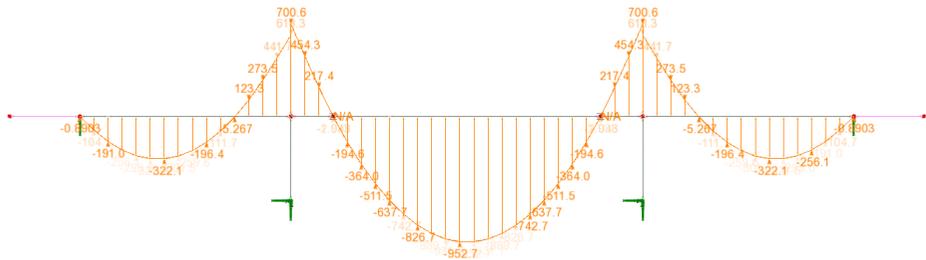


From the updated diagram it can be seen that the bending moment at the drop-in slab bearing location is effectively zero as the joints don't have any rotational stiffness. There is now a reduced amount of moment in the piers which will be investigated next.



Note. Zooming-in on a model will generally show more value data, but in this case the diagram values obtained at the supports will be more easily read by increasing the scale of the diagram.

- In the Treeview double click the **Diagrams** layer, select the **Scale** tab and use a **Local scale** magnitude of **40 mm** and click **OK**.



From the updated diagram it can be seen that the pier itself is now attracting in the order of $700 - 618 = 82\text{kNm}$.

- Turn off the display of the **Contours** and **Diagrams** layers.
- Return the model to an appropriate isometric style of view.

Part 3: Modelling joints at the top of the piers

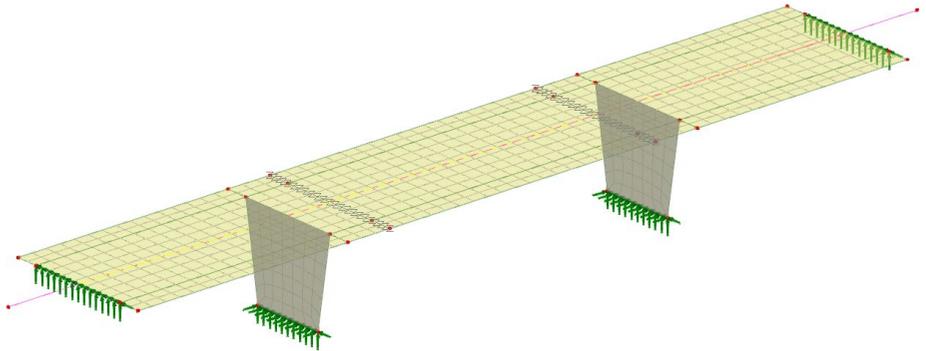
Defining the bearings (joints) for the top of the piers

Define line mesh

To model the bearing strips at the top of the piers, a joint with no rotational stiffness is also to be used.

- Select element type **Joint no rotational stiffness**
- Ensure assignment type is **On single feature (automatic)**
- Enter a name of **Bearing strip for piers** and click **OK**.
- Select the **2 surfaces** representing the piers as shown below.

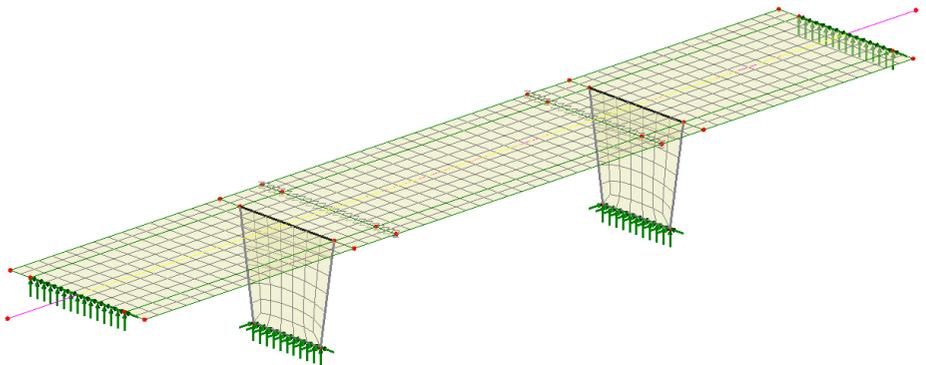
Attributes
Mesh
Line...



Clarify the features to be connected

Because the lines where the joints are to be inserted are shared between surfaces representing the deck and the piers, selection memory must be used. Its use clarifies between which higher order features (the surfaces) the line joints need to be added.

- With the surfaces selected, right-click in the view window and choose **Selection Memory > Set**
- Click in an empty area of the view window to deselect the previously selected surfaces.
- Now select only the **two lines** representing the top of the piers.

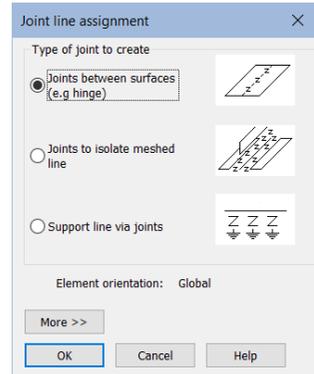


Assign mesh to model

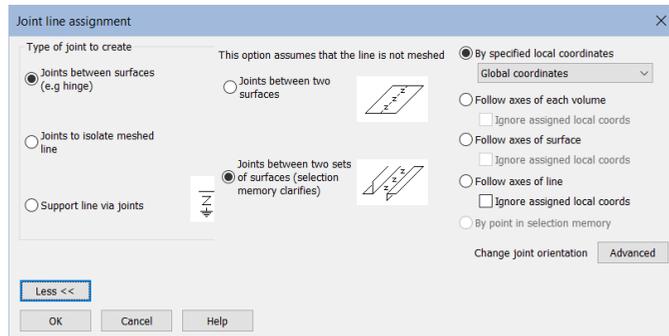
- From the  Treeview drag and drop **Bearing strip for piers** onto the selected features.
- Leave the type of joint to create as the default **Joints between surfaces (e.g. hinge)** and click **OK**.

The View window will carry the wording “Modification Mode” and “Results are out of date” until the model is solved again.

As explained previously, symbols representing the inserted joints will not be seen immediately because the mesh is currently locked. It will be unlocked after the assignment of joint material.



Note. Whilst not required here, the use of the More >> button on the joint assignment dialog provides other options to clarify how joint insertion should be done.



Clear selection memory

Note that deck/pier joint material cannot be assigned when features are in selection memory.

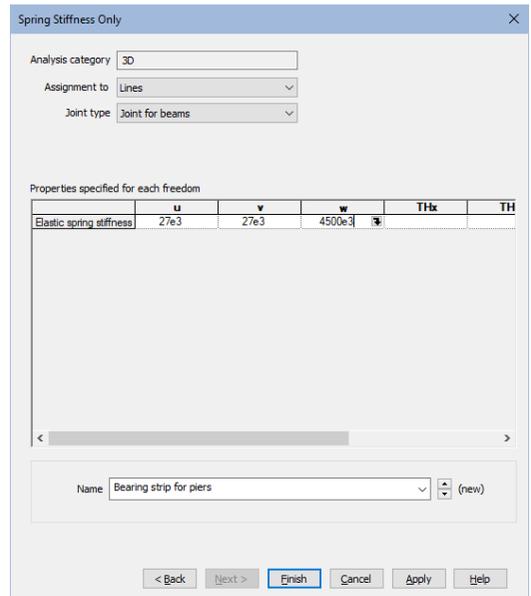
- Right-click in the view window and choose **Selection memory > Clear**

Define deck/pier joint material

- On the Joint material dialog select **Spring Stiffness Only** and click **Next**.

Attributes
Material
Joint..

- On the Spring stiffness dialog define the assignment as being for **Lines**
- Ensure the joint type is set as for **Joint no rotational stiffness**
- Enter a value of **27e3** for the u and v freedoms.
- Enter a value of **4500e3** for the w freedom.
- Enter the name as **Bearing strip for piers** and click **Finish**.



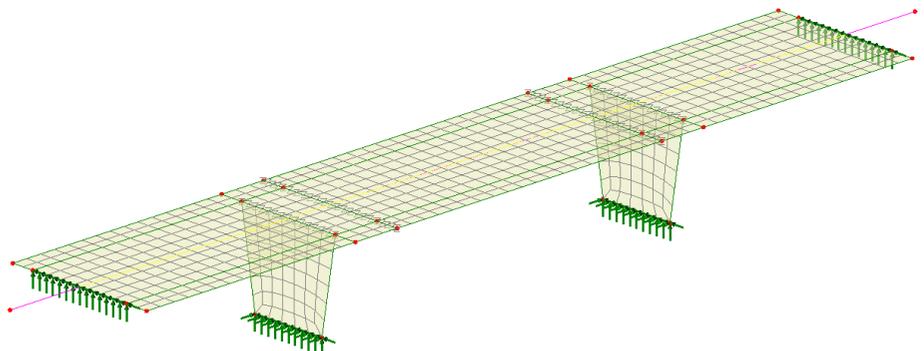
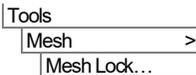
Assign material to model

- Select the two lines representing the top of the piers.
- Drag and drop the joint material **Bearing for piers** from the  Treeview onto the selected features, assigning to lines and pressing **OK**.

Remove the mesh lock

This forces a re-mesh of the model, showing the joints inserted at the bearing locations.

- Click **OK** to close the results files.



Running the analysis with top of pier joints inserted

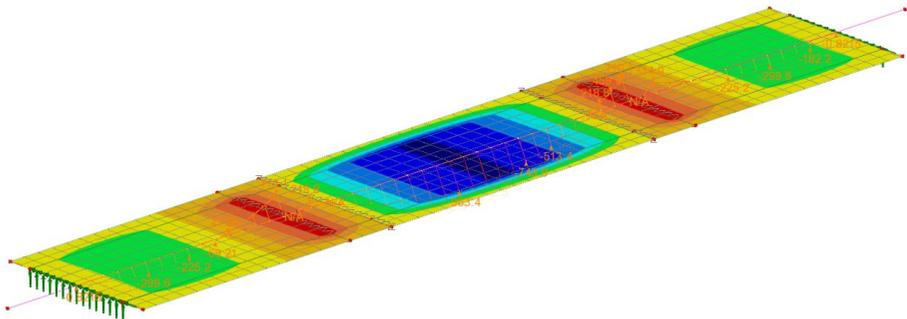
With the model loaded:



Select the **Solve Now** button from the toolbar and click **OK** to run the analysis.

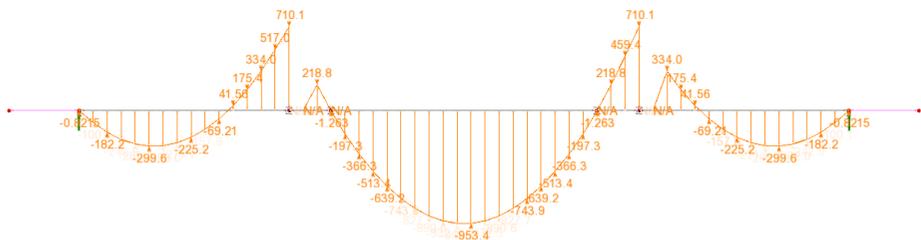
Viewing the results with top of pier joints inserted

- To see just the results for the deck, in the Treeview, right-click on the **Deck** entry and choose **Set as Only Visible**
- In the Treeview turn on the display of the **Contours** layer to see the results for entity **Force/Moment Thick Shell** and results component **Mx**.
- In the Treeview turn on the display of the **Diagrams** layer. Select the **Scale** tab and use a **Local scale** magnitude of **40 mm** and click **OK** to see the results for entity **Force/Moment Thick Shell** and results component **Mx**



This shows that the moment along the lines of the bearings (joints) of the drop-in span is zero as the joints don't have any rotational stiffness.

- Select the button at the bottom of the interface to view the model along the y-axis.



Viewing the results with top of pier joints inserted

From the updated diagram it can be seen that the bending moment at the top of the piers is now 710 and the moment at the drop-in slab bearing locations is effectively zero

This completes the example.

