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

For LUSAS version:	20.0
For software product(s):	LUSAS *Civil & Structural* or LUSAS *Bridge.*
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.
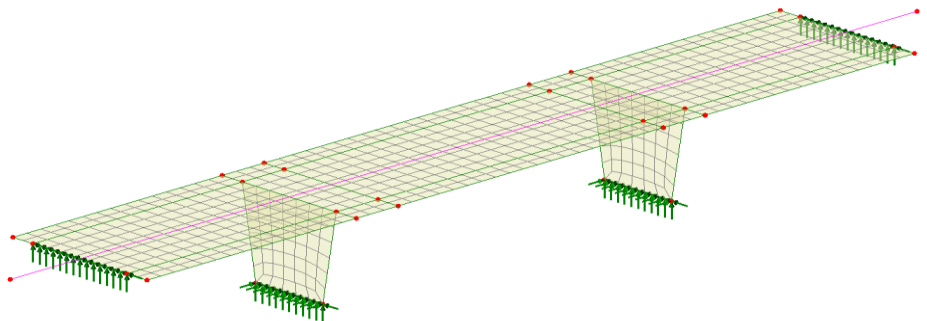
Units used are kN, m, t, s, C throughout.

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

File
New…

- 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.

File
Script   >
Run Script…

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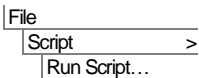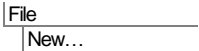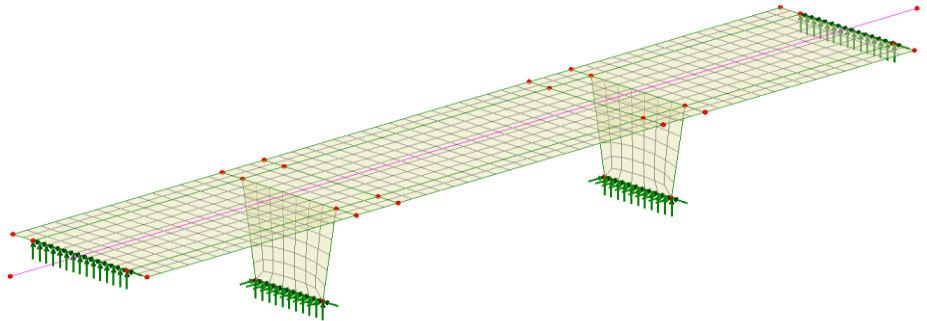
Groups have been defined for the features in the deck and the piers. These are used to restrict the extent of data used to create graphs of results along the deck. The line along the centre of the bridge is used when graphing results.

Save the model

File
   Save

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.

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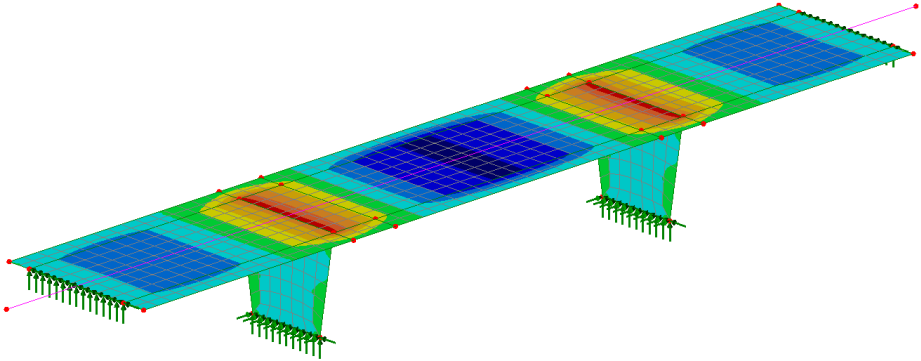
This shows the moments present in the deck and piers when no joints have been assigned to the model.

Graphing of moment along deck

A graph of bending moment Mx 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 Z: N/A 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 middle of the bridge.

Utilities
Graph Through
2D…

- Select the option **By selected line** and click **OK**.

- Define a graph for the **Active** loadcase and select **Visible model** from the Extent droplist to limit the graphed data to that set of elements and click **Next.**

- Select entity **Force/Moment** and component **Mx** and press **Next**.

- Enter a title of **Mx along centreline** and click **Finish**.
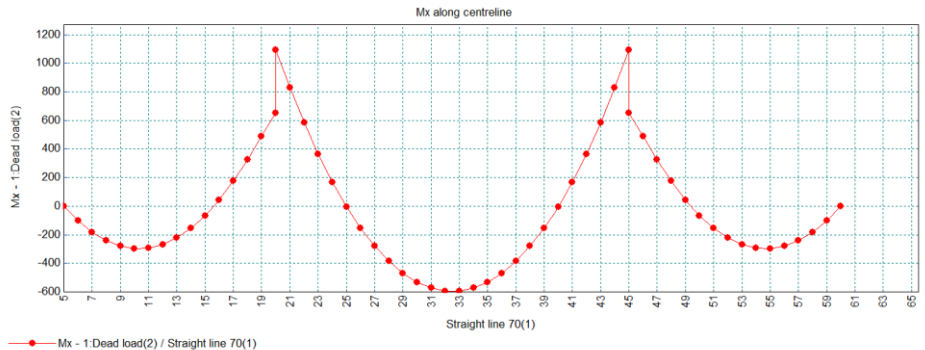
By hovering the cursor over graph points of interest it can be seen that the peak moment at the internal supports in 1092kNm and the lesser moment is 654kNm – meaning that the pier is currently attracting a moment of 1092 – 654 = 438kNm.

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 Y: N/A button at the bottom of the interface to view the model along the y-axis (from the side).
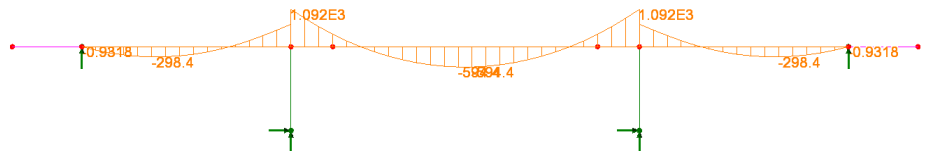
- In the Treeview double click the **Diagrams** layer, select the Diagram Display tab and turn off the **Peaks only** option and click **OK**.

- Use the dynamic zoom button to enlarge the left-hand pier and the ends of the adjacent spans.
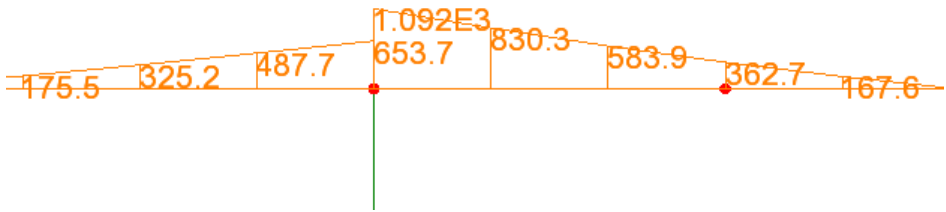
From the updated diagram the previously described bending moments at the support can be seen, and the moment at the intended drop-in slab bearing location is in the order of 363kNm.

- In the ⬚ Treeview double click the **Diagrams** layer, select the Diagram Display tab and turn back on the **Peaks only** option and click **OK**.

- 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
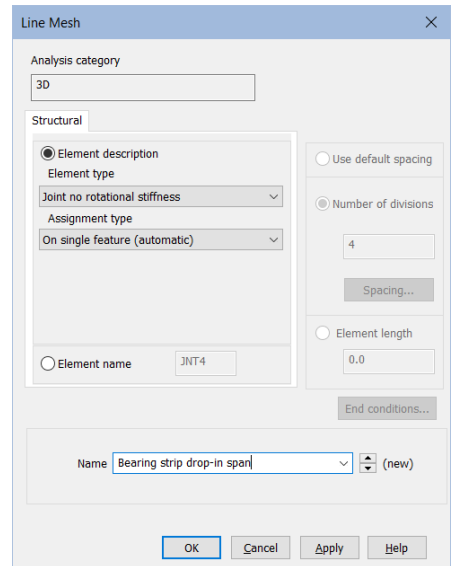
Define line mesh

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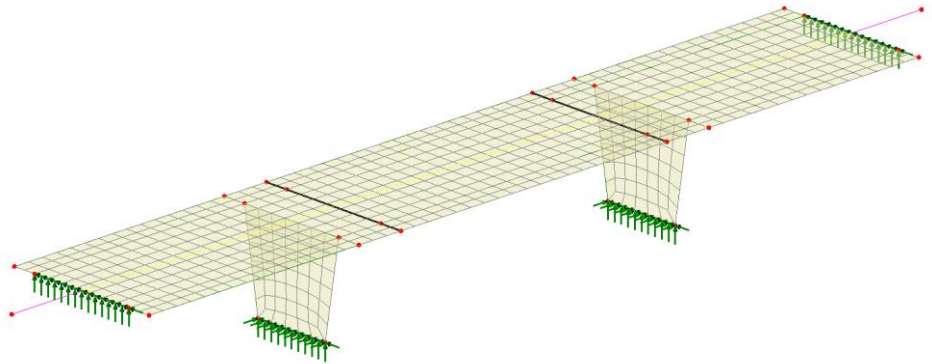
Attributes
  Mesh
    Line...

- 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.
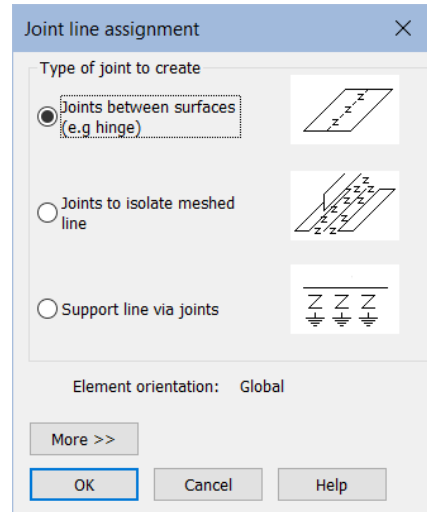
Define drop-in joint material

Attributes
  Material
    Joint...

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

- 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.**
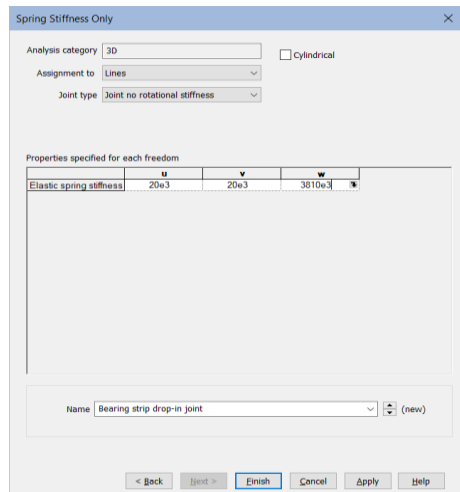
Assign material to model

- 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.
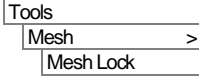
```
Tools
  Mesh          >
    Mesh Lock
```

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

- Click **OK** to close the results files.
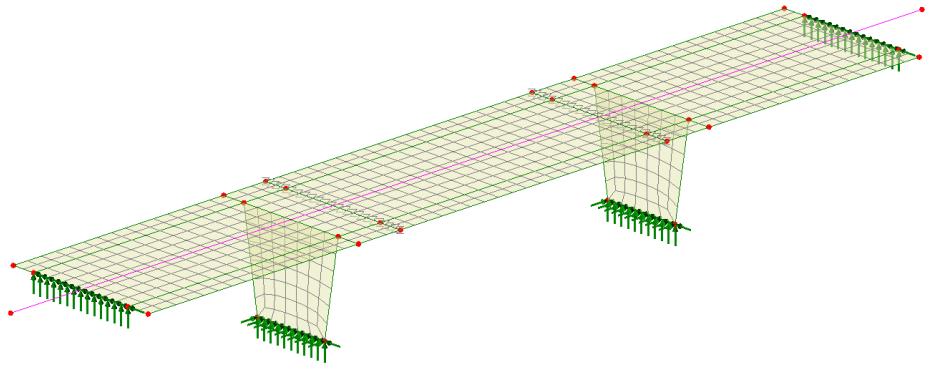
To see the effect of this change the model will be re-solved.

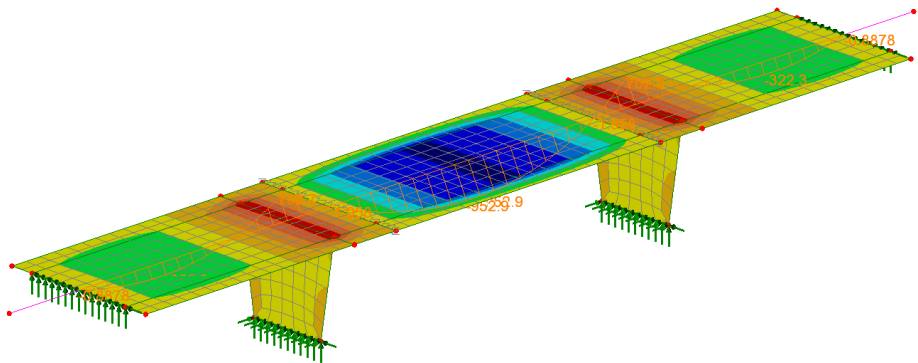# Running the analysis with drop-in span joints inserted

▤ 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.**

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

- Select the Y: N/A button at the bottom of the interface to view the model along the y-axis. The diagrams layer was previously set to show 'Peaks only'
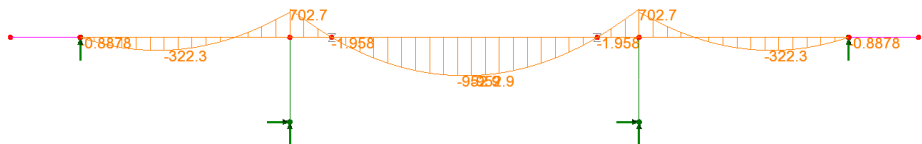
From the updated diagram it can be seen that the bending moment at the drop-in slab bearing location is effectively zero. There is a now a reduced amount of moment in the piers which will be investigated next.

- In the ▣ Treeview double click the **Diagrams** layer, select the Diagrams Display tab and turn off the **Peaks only** option and click **OK**.

- ▶ Use the dynamic zoom button to enlarge the left-hand pier and the ends of the adjacent spans.
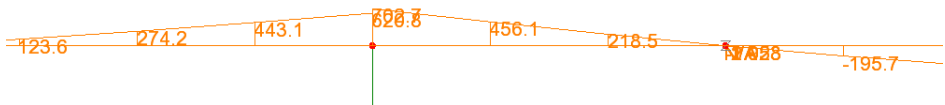
From the updated diagram it can be seen that the pier itself now attracting in the order of 702 – 620 = 82kNm.

- 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.

| Attributes |
| --- |
| Mesh |
| Line... |

- 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.
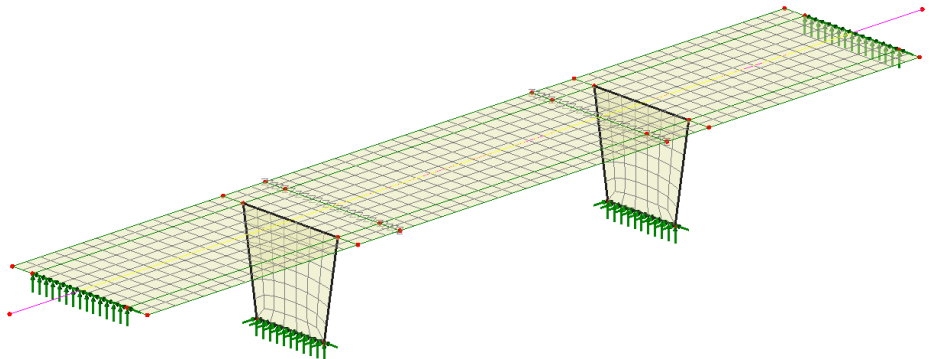
Clarify the features to be connected

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 (the features previously selected and put into selection memory will appear as grey lines).

- Now select only the **two lines** representing the top of the piers.
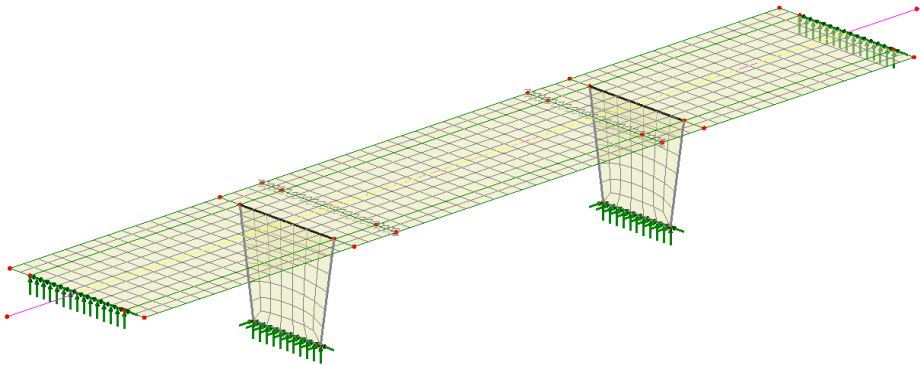
Assign mesh to model

- 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.**

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.
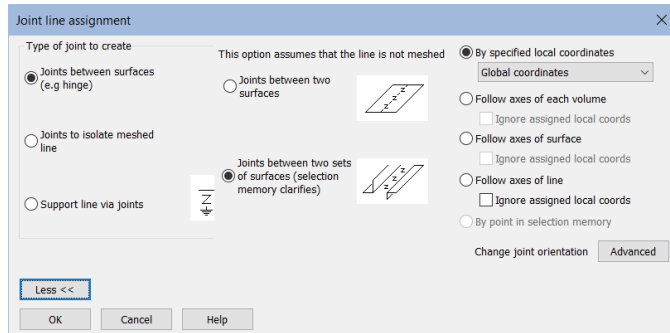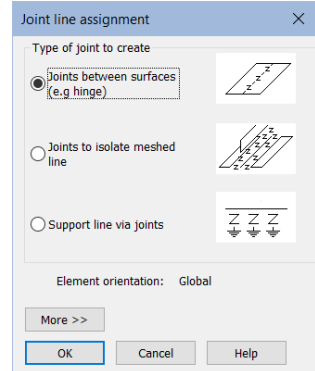
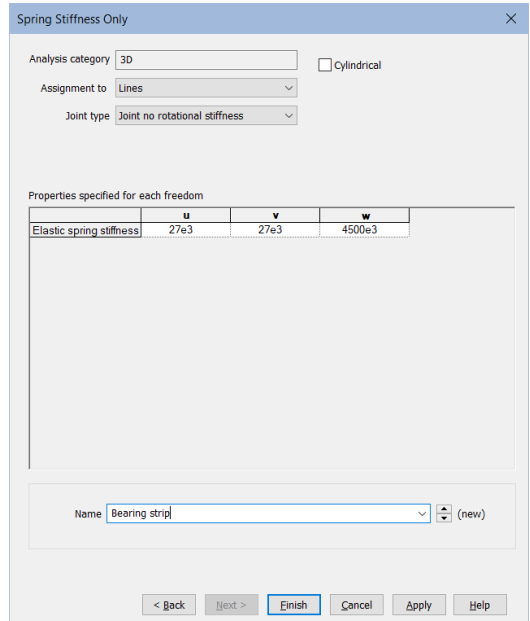

Clear selection memory

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.**

- 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.**

Spring Stiffness Only

Analysis category	3D
Assignment to	Lines
Joint type	Joint no rotational stiffness

☐ Cylindrical

Properties specified for each freedom

	u	v	w
Elastic spring stiffness	27e3	27e3	4500e3

Name Bearing strip (new)

[< Back] [Next >] [Finish] [Cancel] [Apply] [Help]

Assign material to model

- With the lines representing the top of the piers still selected, 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.
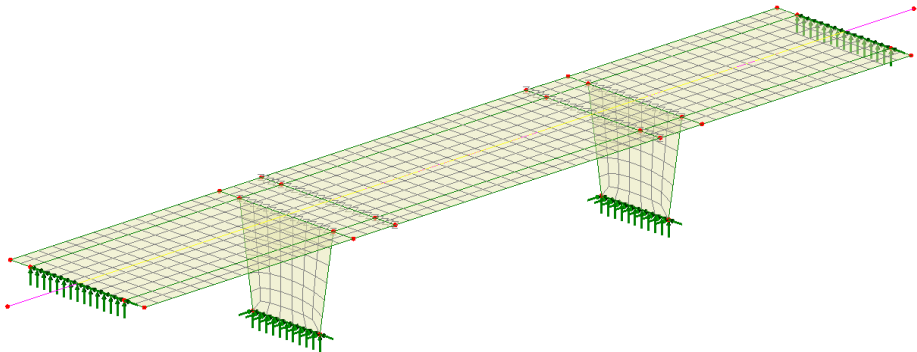
## Running the analysis with top of pier joints inserted

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** to see the results for entity **Force/Moment Thick Shell** and results component **Mx.**

- In the ▣ Treeview double click the **Diagrams** layer, select the Diagram Display tab and turn on the **Peaks only** option and click **OK**.
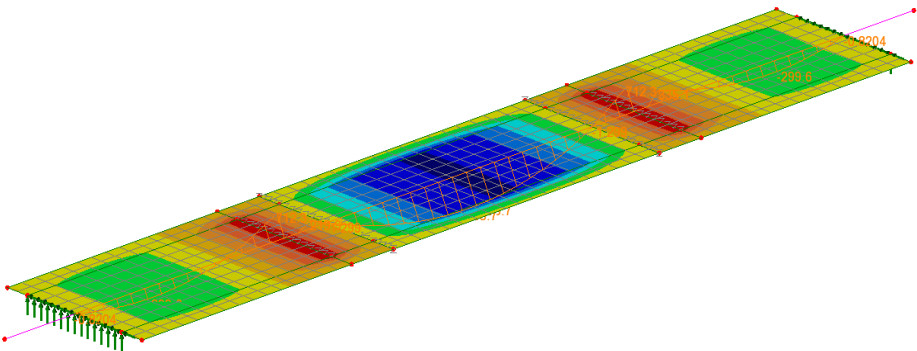
This shows that the moment along the lines of the bearings (joints) of the drop-in span is zero as the joints don't have any rotational stiffness.

- Select the Y: N/A button at the bottom of the interface to view the model along the y-axis.
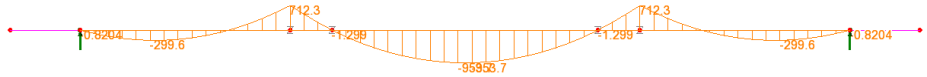
From the updated diagram it can be seen that the bending moment at the top of the piers is now 712kN and the moment at the drop-in slab bearing locations is effectively zero

This completes the example.