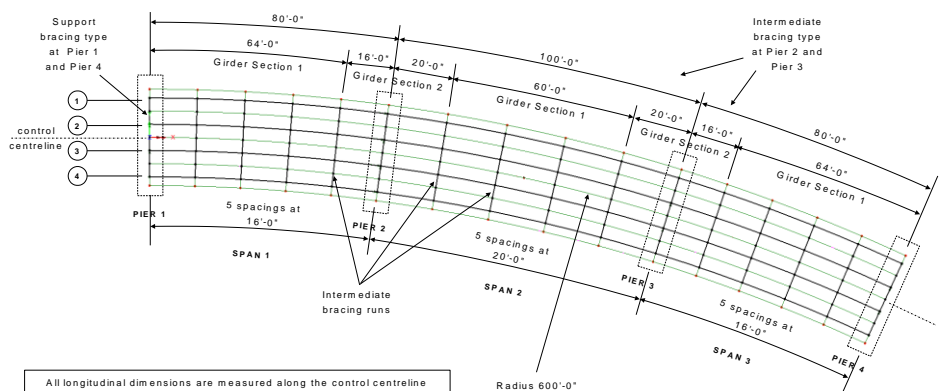


# Staged Construction Modelling of a 3-Span Bridge Deck

For LUSAS version:	19.0
For software product(s):	LUSAS <i>Civil &amp; Structural</i> or LUSAS <i>Bridge</i> .
With product option(s):	Nonlinear
Note: This example exceeds the limits of the LUSAS Teaching and Training Version.	

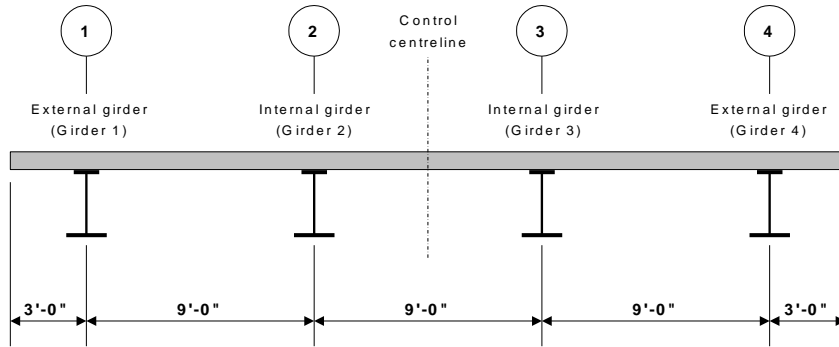
## Description

This example shows how to model the staged construction of a 3-span curved bridge deck. It uses a supplied model that was created by the Steel Composite Bridge Wizard.



The bridge consists of three spans of length 80 feet, 100 feet and 80 feet. The bridge follows a radius of 600 feet along the control centreline and all supports are along radials. There are four girders across the width of the deck with stiffeners and bracing provided at supports and fifth point intervals along the span lengths. The deck cross-section is shown below.

## Staged Construction Modelling of a 3-Span Bridge Deck



Girder and slab thickness are as follows:

	External girder section 1	Internal girder section 1	External girder section 2	Internal girder section 2
Slab width	7'6"	9'0"	7'6"	9'0"
Slab thickness	8"	8"	8"	8"
Slab height of haunch	1.5"	1.5"	1.5"	1.5"
Slab offset to web	-9"	0"	-9"	0"
Top flange width	18"	18"	18"	18"
Top flange thickness	0.75"	0.75"	1.0"	1.0"
Web depth	4'0"	4'0"	4'0"	4'0"
Web thickness	0.5"	0.5"	0.5"	0.5"
Bottom flange width	24"	24"	24"	24"
Bottom flange thickness	1"	1"	1.5"	1.5"

### Loading

For reasons of brevity and simplicity, in the in-service condition this example is limited to consideration of dead loads and traffic loads only and in the case of limit states is limited to Strength I, Service II and Fatigue I. Traffic loads for a 30'-0" wide carriageway are generated using the Vehicle Load Optimisation facility. Construction stages are considered, with a uniform construction load applied to the wet and hardened areas of concrete.

The following loads are considered in addition to the self-weight of the steel beams:

Item	Loading
Wet concrete slab	0.10 kip/ft <sup>2</sup>
Construction load	0.02 kip/ft <sup>2</sup>
Parapet	0.225 kip/ft
Surfacing	0.023 kip/ft <sup>2</sup>

Note due to the modelling method used, the wet concrete slab and construction load are assigned to the model as line loads of an appropriate value per unit length, and not as a surface load per unit area.

#### Load combinations considered:

- Constructability      1.4 (DC + C)
- Strength I              1.25 (DC) + 1.5 (DW) + 1.75 (LL)
- Service II              DC + DW + 1.3 (LL)
- Fatigue I                1.75 (LL)

where DC = dead load: components and attachments

DW = dead load: wearing surface and utilities

C = construction load

LL = vehicle live load

Units used are kip,ft,kslug,s,F throughout.

#### Objective

To show the modelling processes required to carry out a staged construction analysis of a 3-span curved bridge deck.

#### Keywords

3D, Steel, Concrete, Composite, Bridge, Structural, AASHTO 8<sup>th</sup>, Bridge Deck Grillage, VLO, Staged Construction, Load Combinations, Branched Analysis.

#### Associated Files



- composite\_bridge\_deck\_design\_preliminary.mdl** contains the basic model geometry. The example uses this model as a starting point.

- ❑ **composite\_bridge\_deck\_design\_sections.csv** This file contains the geometric sections referenced by the supplied preliminary model file. It should be copied to the same folder where the supplied model is saved and renamed to be **sections.csv**.
- ❑ **composite\_bridge\_deck\_design\_ready.mdl** includes the definition of design members, creation of the construction stages and Vehicle Load Optimisation settings. To be used where stated in case of user errors in developing the staged construction model.

### Associated Examples

- ❑ The worked example ‘Steel Composite Bridge Wizard’ illustrates how to create the initial model of the steel composite bridge using the Steel Composite Bridge Wizard.
- ❑ The worked example ‘Composite Bridge Deck Design’ shows how to carry out a design check of a staged construction model of a 3-span bridge deck.

## Modelling

### 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.



### Loading the supplied starting model

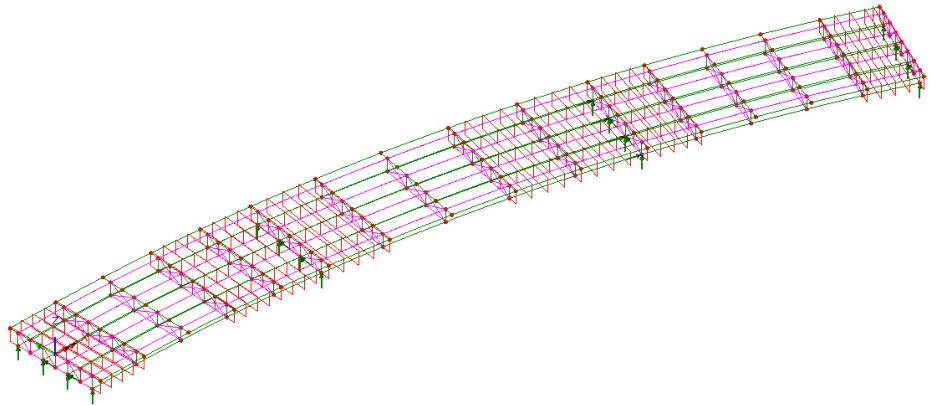
The supplied starting model is based on the geometry used in the worked example ‘Steel Composite Bridge Wizard’. It includes the design attributes described during and at the end of that example. It also includes the lines that define the extent of the 30’-0” wide carriageway in a group named ‘Kerblines’ and the series of lines forming the extent of the deck to which parapet loads are assigned in a group called ‘Parapet’.

- To load the model, open the read-only file **composite\_bridge\_deck\_design\_preliminary.mdl** located in the <LUSAS Installation Folder>\Examples\Modeller directory.

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

-  If necessary, select the isometric button.


-  Toggling the Fleshing button on and off will show the steelwork and slab arrangement.
- In the  treeview turn off the display of the **Mesh** layer.



**Note.** The kerb lines seen in the model were created by selecting three points on the start, middle and end of each outer extent of the deck prior to selecting the **Geometry > Line> Arc/Circle > from Coords/Points** menu item. Then, each newly created arc was selected, and using the **Geometry > Line > By Offsetting** menu item each kerb line was created 18” away from the selected arc, prior to adding each line representing a kerb to the ‘Kerblines’ group. Lines forming the inner and outer extents of the deck were also selected and saved as a group named ‘Parapets’.

### Save the model

File  
Save As

-  Save the model file to a new working folder on the local PC where you have read and write permission, and save the model with the new name of **composite\_bridge\_deck\_staged.mdl**

### Copying the supplied geometric sections file

A supplied file containing the geometric sections that are referenced by the supplied preliminary model file should be copied to the same folder where the model has been saved and then be renamed.

- From the <Lusas installation folder> Examples > Modeller > Examples folder, copy the file **composite\_bridge\_deck\_design\_sections.csv** to the same folder where the supplied model was saved. Rename the file to be **sections.csv**

### Construction and loading phases to be considered

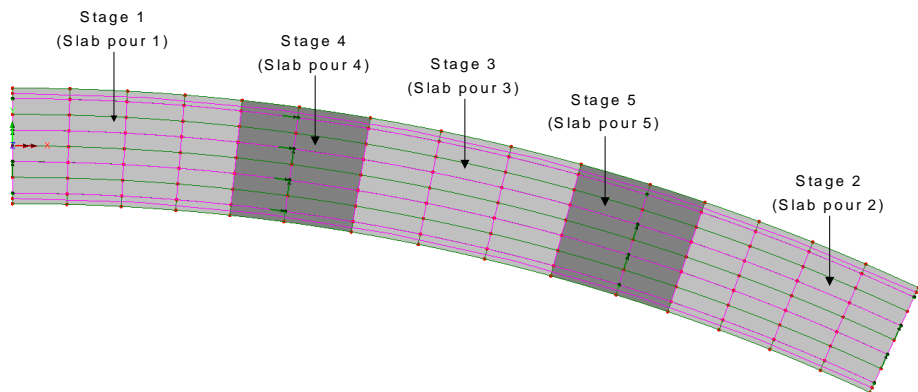
The following modelling and loading phases are to be considered:

- Slab construction stages
- Phase 1 - Steel only
- Phase 2 - Permanent and long term loading (Parapet and surfacing loads)
- Phase 3 - Variable or short-term loading (Vehicle loading)

### Slab Construction Stages

Five stages of slab construction will be considered in this model, with each representing a slab pour, and with each defined as a separate loadcase. They are:

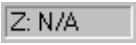


- Stage 1: Pour concrete slab over the extent of Girder Section 1 in Span 1.
- Stage 2: Pour concrete slab over the extent of Girder Section 1 in Span 3.
- Stage 3: Pour concrete slab over the extent of Girder Section 1 in Span 2.
- Stage 4: Pour concrete slab over the extent of Girder Section 2 in Spans 1-2.
- Stage 5: Pour concrete slab over the extent of Girder Section 2 in Spans 2-3.



**Note.** In a slab pour the weight of the concrete for that pour is to be allowed for in that stage, but the slab itself is not activated in the model until the next stage, when the concrete has hardened and the slab has become ‘active’ in the structure.

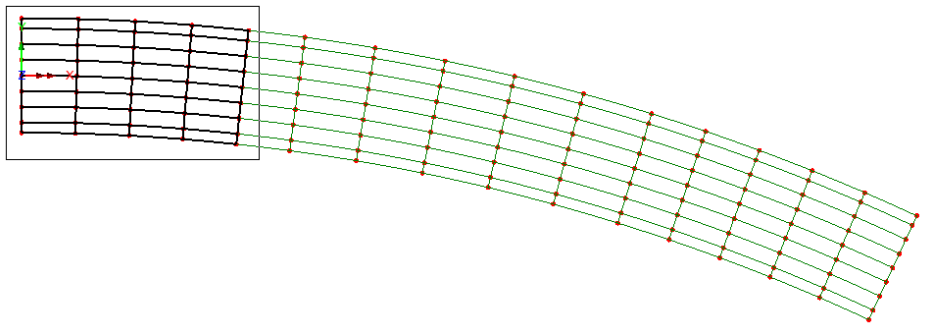
## Creating groups to represent slab pours


To aid with the modelling of the staged construction process, features representing the different areas of slab will be put into named groups.

-  Select the **View from Z-axis** cell at the bottom of the interface to make selection of features easier. Resize the view to show the whole model if necessary.
-  In the treeview right-click on the **Concrete** material attribute and choose **Set as Only Visible**.
-  Change the selection cursor to 'Select Surfaces' only.

### Pour 1

- Box-select the surfaces in the first four braced bays in span 1 that cover the extent of the geometric attributes of type 'Section 1'.



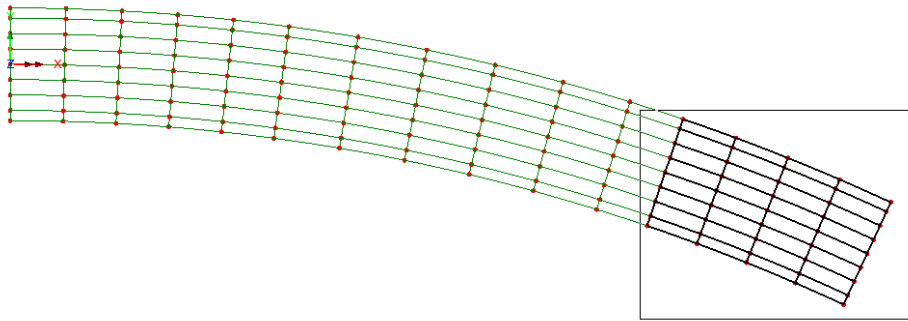
-  Create a new group and change the name to **Slab stage 1** and click **OK**.


### Pour 2

- Box-select the surfaces in the last four braced bays in span 3 that cover the extent of the geometric attributes of type 'Section 1'.

## Staged Construction Modelling of a 3-Span Bridge Deck

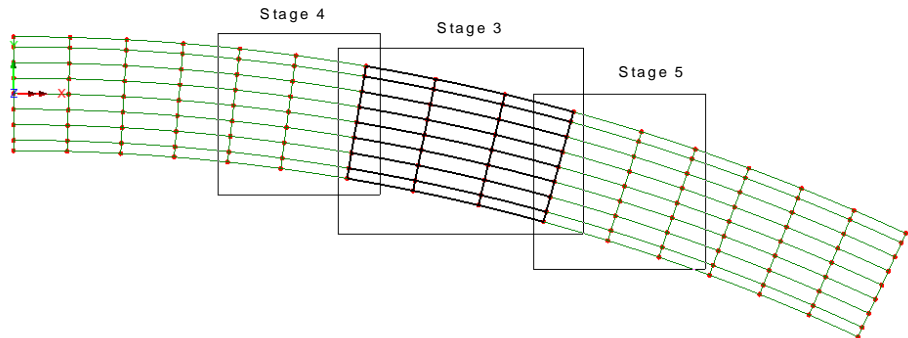
---




-  Create a new group and change the name to **Slab stage 2** and click **OK**.

### Pours 3, 4 and 5

- Repeat the process by box-selecting the appropriate features, and creating named groups of **Slab stage 3**, **Slab stage 4** and **Slab stage 5**, as shown on the following image:



**Note.** The members in a group can be checked by right-clicking on a group name in the  Groups treeview and selecting **Select Members**.

### Defining loading

Slab pour loading is to be represented by global distributed loads applied to the top flange of each composite design member as a per unit length load. The same is to be done for slab construction loading. Parapet loading is also defined per unit length. This modelling method requires the loading for each girder to be calculated manually according to the width of slab that each girder supports. Surfacing loading is to be defined per unit area for assignment to the relevant surfaces of the model that represent the deck. Construction loading at the wet concrete stage is considered.



Attributes

Loading

### Slab loading on external beams

- Select the **Global Distributed** radio button and click **Next**.
- Choose **Per unit length** and enter a value of **-0.75** in the Z direction cell.
- Change the name to **Slab external** and press **Apply**.

### Slab loading on internal beams

- Change the load in the **Z** direction to **-0.90**
- Change the name to **Slab internal** and press **Apply**.

### Construction loading on external beams

- Change the load in the **Z** direction to **-0.15**
- Change the name to **Construction external** and press **Apply**.

### Construction loading on internal beams

- Change the load in the **Z** direction to **-0.18**
- Change the name to **Construction internal** and press **Apply**.


### Parapet loading

- Change the load in the **Z** direction to **-0.225**.
- Change the name to **Parapet** and press **Apply**.

### Surfacing

- Change the load type to **Per unit area**.
- Change the load in the **Z** direction to **-0.023**
- Change the name to **Surfacing** and press **OK**.

### Creating the first construction stage

- Accept the default settings, change the analysis name to **Construction Stage** and click **OK**.
- In the  treeview, change the name of Loadcase 2 in the Construction stage analysis to be **Slab Pour 1**.

Analyses

Structural  
Analysis...

### Creating activate and deactivate attributes

Activate and Deactivate attributes are used to model the staged construction process.

Attributes

Activate and  
Deactivate

- With the **Activate** radio button selected click **Next**.
- Accept the default name of **Act1** and click **Apply**.
- Click the **Back** button.
- Change the selection to the **Deactivate** radio button and click **Next**.
- Accept the default settings and name of **Deact1** and click **Finish**.

### Modelling the slab construction stages

The Activate and Deactivate attributes need to be assigned to features in the model.








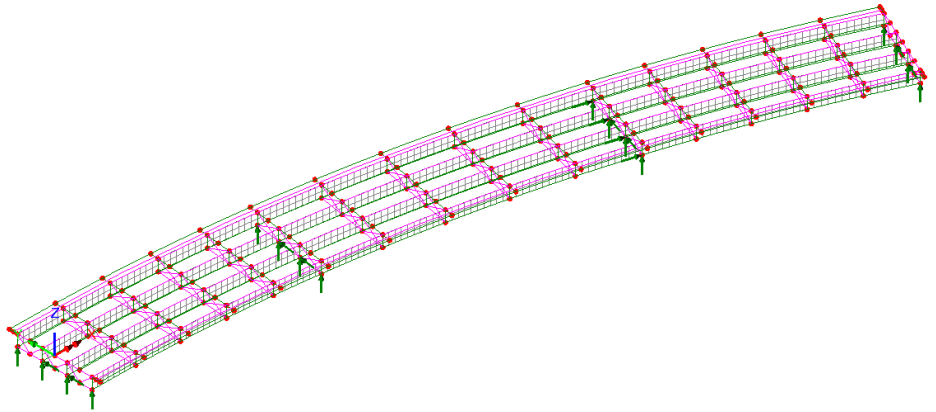
**Note.** To model a staged construction process all the features and elements required to model that process must be present in the model prior to solving the model. Selected features may be de-activated in readiness for activation in stated analyses and loadcases as required.

#### Construction stage: Slab pour 1

The elements that have been assigned concrete material now need to be deactivated because the concrete slab will not be present in the initial construction stage.

Note that in a slab pour the weight of the concrete for that pour is allowed for in that stage but the slab is not activated until the next stage when the concrete has hardened and the slab has become ‘active’ in the structure.



-  Select the ‘**Any Cursor**’ option.
- Click in an empty region of the view window to deselect any selected members.
- Right-click in a blank part of the view window and select **All Visible**.
- In the  treeview turn on the display of the **Mesh** layer.
- In the  treeview right-click on the **Concrete** material attribute and choose **Select Assignments**.
- On the Options dialog select **Clear selection** (if available – otherwise accept the default settings) and click **OK**.
- On the Clarify Assignment(s) to Act On dialog select **Analysis 1** and click **OK**.
- From the  treeview, drag and drop the attribute **Deact1** onto the selection.
- On the Assign/Deactivate dialog for the analysis choose **Construction Stage** and for the loadcase choose **Slab Pour 1** . Click **OK**.
-  Select the Isometric button.

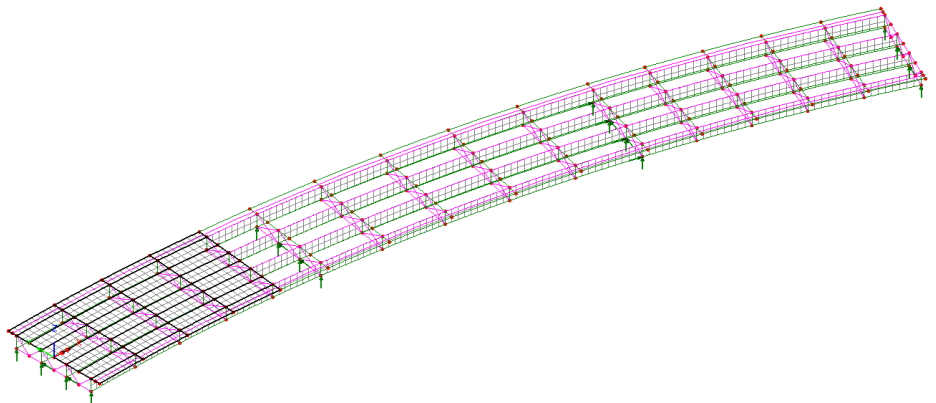


So, prior to slab pour 1 loading being applied, only the steelwork will be present.

### **Construction stage: Slab pour 2**



Now the elements representing the initial concrete slab pour can be activated.

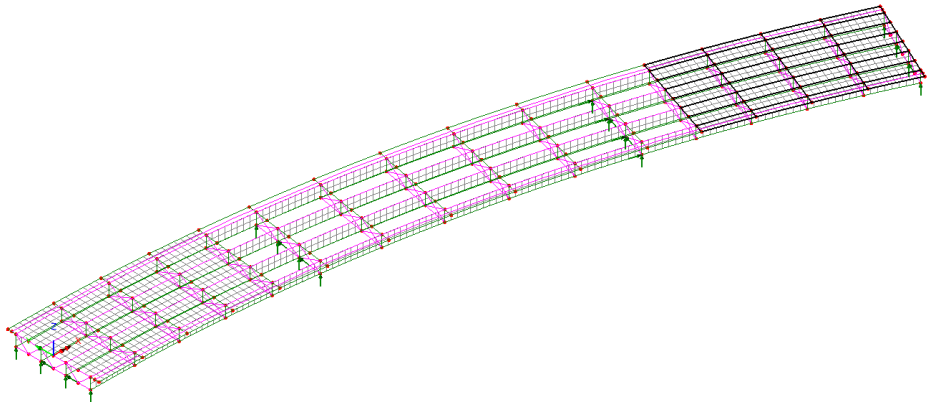
- In the  treeview right-click on **Slab stage 1** and choose **Select members**. On the subsequent dialog select **Yes** to deselect the objects already selected.
- From the  treeview drag and drop the attribute **Act1** onto the selection.
- On the assignment dialog change the analysis to **Construction stage** and in the loadcase textbox type **Slab pour 2** (the loadcase in which this slab will become active) and click **OK**.



So, prior to slab pour 2 loading being applied, slab pour 1 concrete will become active.

### Construction stage: Slab pour 3

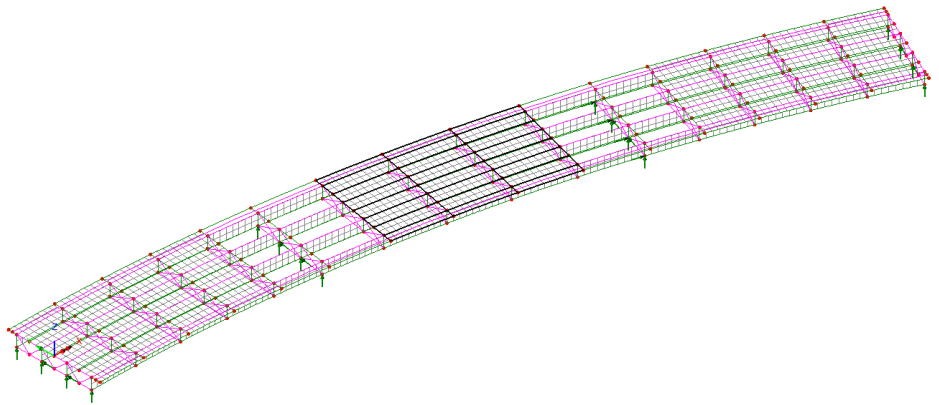
- In the  treeview right-click on **Slab Stage 2** and choose **Select members**. On the subsequent dialog select **Yes** to deselect the objects already selected.
- From the  treeview drag and drop the attribute **Act1** onto the selection.
- On the assignment dialog ensure the analysis is set to **Construction stage** and in the loadcase textbox change the name, by typing, to **Slab pour 3** (the loadcase in which this slab will become active) and click **OK**.



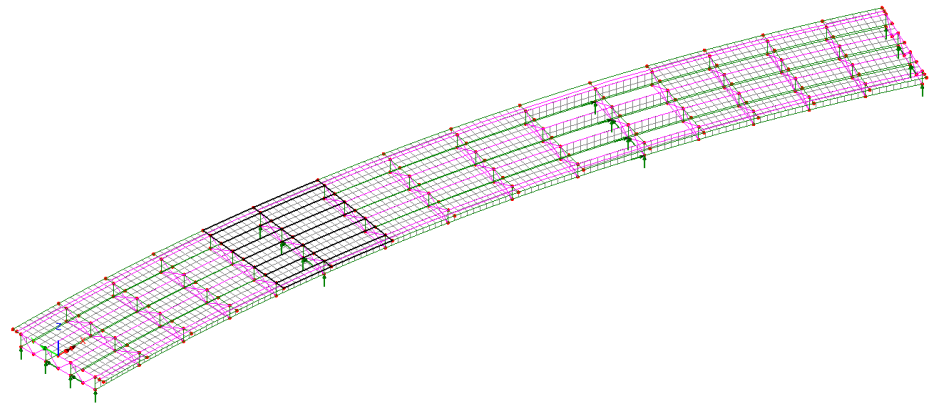
So, prior to slab pour 3 loading being applied, slab pour 2 concrete will become active.

### Construction stages: Slab pour 4, 5 and Full self weight

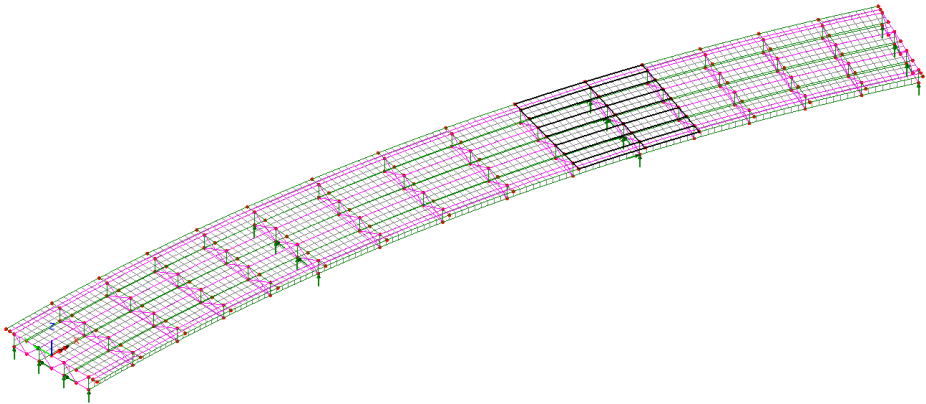
- Repeat the process by right-clicking on the groups **Slab stage 3**, **Slab stage 4** and **Slab stage 5** in turn and choosing **Select members**.
- For each selection, activate each set of members for the **Construction** stage analysis, making loadcase assignments of **Slab pour 4**, **Slab pour 5** and **Full self weight** respectively for the three remaining areas of deck to be cast.



So, prior to slab pour 4 loading being applied, slab pour 3 concrete will become active.





So, prior to slab pour 5 loading being applied, slab pour 4 concrete will become active.

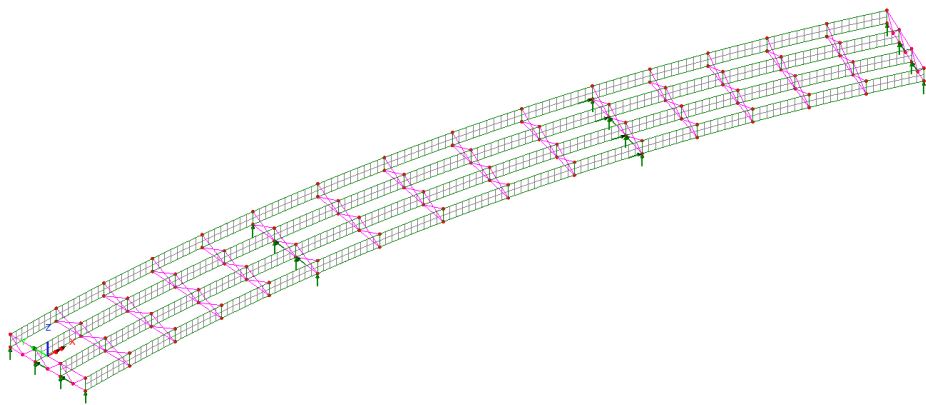


So, for the full self weight loadcase, slab pour 5 concrete will become active.


### Assigning the steel loads

- Click in an empty region of the view window to deselect any previous selection.
- In the  treeview, right-click on the loadcase **Slab pour 1** and select **Set active**
- In the  treeview, right-click on the **Steel** material attribute and choose **Set as Only Visible**.
- On the Clarify Assignment(s) to Act On dialog select **Analysis 1** and click **OK**.



This updates the view window to show only those features to which the steel material has been applied.





## Assigning gravity for the staged construction loadcases

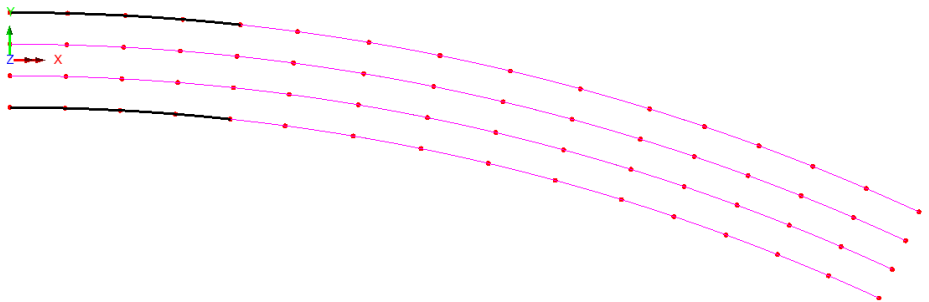
- In the  treeview, right-click on the **Construction Stage** analysis and select **Add Gravity**.


## Assigning the Concrete pour loading

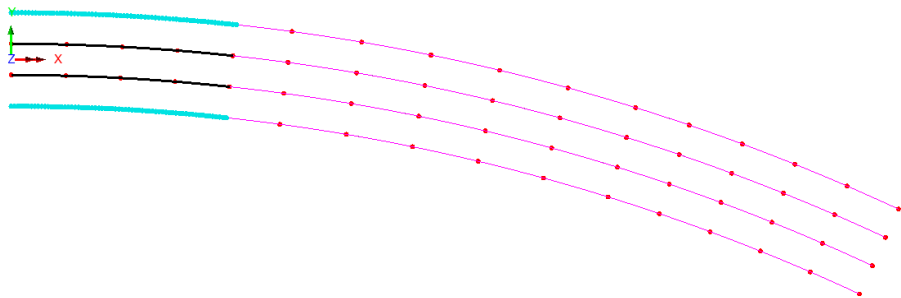
- In the  treeview right-click on **Girder 1 – Top flange** and choose **Set as Only Visible**.
- In the  treeview, click on **Girder 2 – Top flange**, then hold-down the **Ctrl** key and add **Girder 3 – Top flange** and **Girder 4 – Top flange** to the initial selection. Right-click and choose **Visible**.


## Concrete pour 1 loading

-  Click the ‘View along Z-axis’ button at the bottom of the user interface to obtain a plan view of the model.
-  Change the selection cursor to ‘Select lines’.
- By holding-down the **Ctrl** key, select the eight lines that represent the extent of the geometric attribute ‘Section 1 external’ in Span 1.

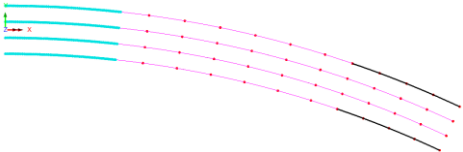



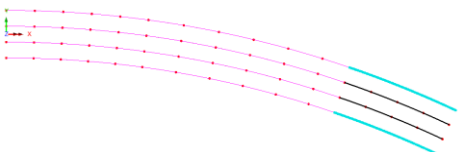

- From the  treeview drag and drop the **Slab external** loading attribute onto the selected features.
- On the Loading Assignment dialog ensure that the Analysis is set to **Construction stage** and Loadcase to **Slab pour 1** and click **OK**.
- Select the eight lines that represent the extent of the geometric attribute ‘Section 1 internal’ in Span 1.



- From the  treeview drag and drop the **Slab internal** loading attribute onto the selected features and click **OK**.

### Concrete pour 2 loading

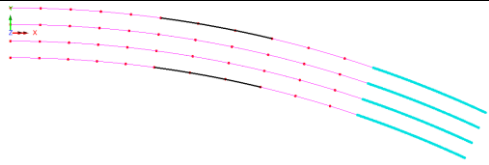
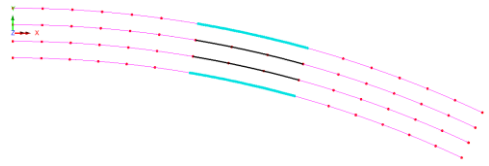
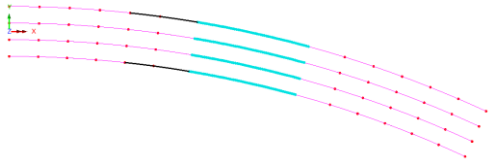
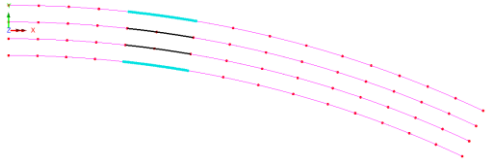
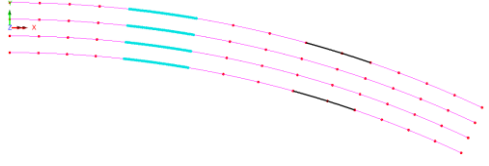
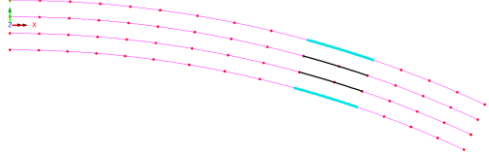
- Select the eight lines that represent the extent of the geometric attribute 'Section 1 external' in Span 3.
- 
- From the  treeview drag and drop the **Slab external** loading attribute onto the selected features.
  - On the Loading Assignment dialog ensure that the Analysis is set to **Construction stage** and Loadcase is set to **Slab pour 2** and click **OK**

- Select the eight lines representing the extent of the geometric attribute 'Section 1 internal' in Span 3.
- 
- From the  treeview drag and drop the **Slab internal** loading attribute onto the selected features and click **OK**.

### Concrete pour 3 to 5 loading


- Repeat for the remaining concrete pours, selecting the appropriate lines representing the extent of the geometric attribute 'Section 1 external / internal' and 'Section 2 external / internal' ensuring that they are assigned to the Single loadcases and range of loadcases shown in the following table.



Select these lines	Assign loading	Analysis	Loadcase
	Slab external	Construction stage	Slab pour 3
	Slab internal	Construction stage	Slab pour 3
	Slab external	Construction stage	Slab pour 4
	Slab internal	Construction stage	Slab pour 4
	Slab external	Construction stage	Slab pour 5
	Slab internal	Construction stage	Slab pour 5

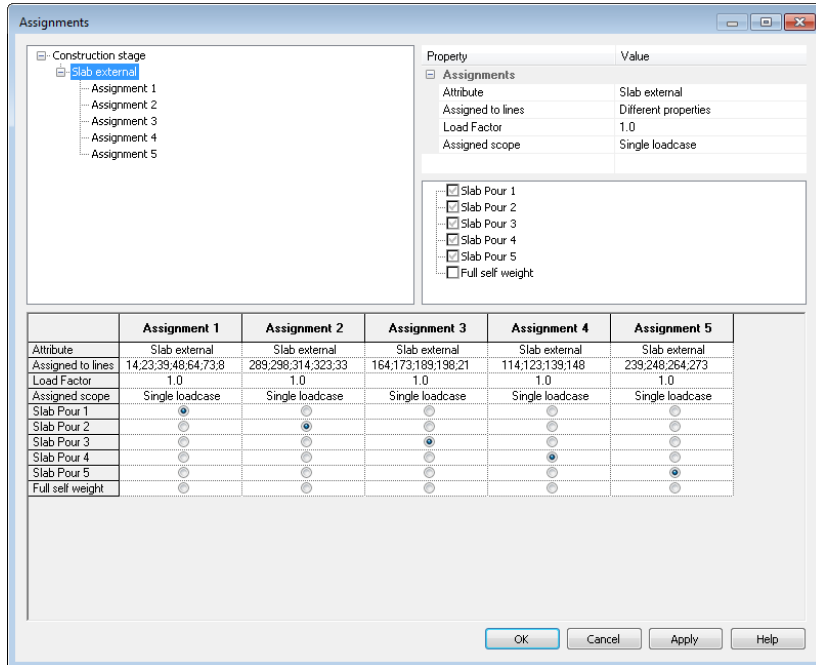
### Checking slab pour loading assignments

The loading assignments made can be checked in a tabular / grid form:

- In the  treeview right-click on **Slab external** and choose **Edit Assignments**.

## Staged Construction Modelling of a 3-Span Bridge Deck



The assignments made for the 'Slab external' loading should look as shown in the following image. If any corrections are required they can be made directly on this Assignments dialog.



- A similar check should be made for the slab internal load assignments.

### Creating analysis branches to assess construction loads


Analysis branches allow the running of one or more sub-analyses during the solution of a main analysis to investigate the state of the model at a particular loadcase to either investigate the movement of construction equipment, or carry out a frequency or a buckling analysis, for instance. In this example analysis, general construction loading is modelled. Analysis branches require (or can inherit) a nonlinear or transient control. The nonlinear control will be defined after assigning the construction loads.

- In the  treeview right-click on the loadcase **Slab pour 1** and choose **New** followed by **Branch**
- Accept the default settings and change the name to **Temporary load 1** and click **OK**
- In the  treeview right-click on the loadcase **Slab pour 2** and choose **New** followed by **Branch**

- Accept the default settings and change the name to **Temporary load 2** and click **OK**
- Repeat, creating an analysis branch of the name **Temporary load <n>** for the remaining loadcases **Slab pour 3 to Full self weight**
- Rename the loadcases in each of the analysis branches to be **Construction load 1**, **Construction load 2** etc.

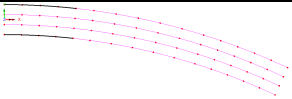
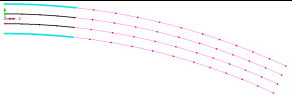
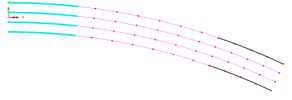
### **Assigning the construction loads**

The construction loads need to be assigned to the top flanges of the external and internal beams for each construction stage.

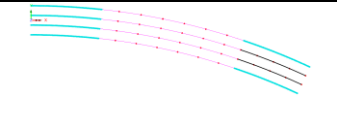
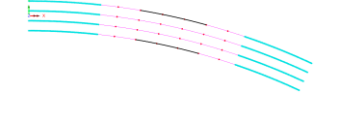
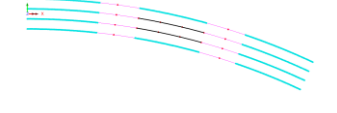
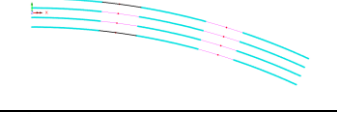
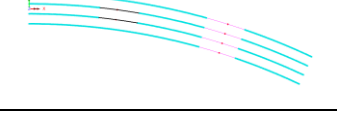
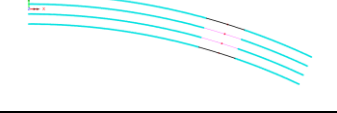
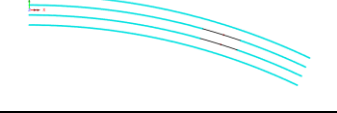
For an advanced user, one way this could be done would be to copy and paste the assigned ‘Slab external’ and ‘Slab internal’ loadings for each Slab Pour in the  treeview and paste them into the Temporary load analysis branches for each Construction load loadcase, and then edit the Assignments for those added loadings to reference the ‘Construction external’ and ‘Construction internal’ loadings instead. But great care would be required to ensure that the correct loadings (referencing the appropriate lines in the model) were assigned to the correct loadcases for all relevant stages of construction.


A more visual way, is to select lines and assign the loadings to the model in a similar manner to that done previously for concrete slab external and internal loading. This is the method adopted in this example.

- With a plan view showing just the lines representing the top flanges, select the appropriate external beams and internal beams for each stage and assign construction external loading / construction internal loading, as shown in the following table:

Select these lines	Assign loading	Individually to these analyses
	<b>Construction external</b>	<b>Temporary load 1, 2, 3, 4, 5, 6</b> (the construction load will automatically change each time to suit)
	<b>Construction internal</b>	<b>Temporary load 1, 2, 3, 4, 5, 6</b>
	<b>Construction external</b>	<b>Temporary load 2, 3, 4, 5, 6</b>


## Staged Construction Modelling of a 3-Span Bridge Deck

	Construction internal	Temporary load 2, 3, 4, 5, 6
	Construction external	Temporary load 3, 4, 5, 6
	Construction internal	Temporary load 3, 4, 5, 6
	Construction external	Temporary load 4, 5, 6
	Construction internal	Temporary load 4, 5, 6
	Construction external	Temporary load 5, 6
	Construction internal	Temporary load 5, 6

-  On completion, change the selection cursor back to 'Select Any'.

### Setting Nonlinear Controls

Since the model includes Activation and Deactivation of features, nonlinear controls must be set before the model can be solved.


- In the  treeview right-click on the **Slab pour 1** loadcase and from the **Controls** menu item, select **Nonlinear and Transient**.
- In the Incrementation panel, select **Nonlinear** and choose **Manual** incrementation.
- In the Solution strategy panel change the Incremental displacement norm to **0**.
- Accept the remaining default values and click **OK**.

## Phase 1 – Steel only

### Steel only loads



A new analysis is required to consider the Phase 1 steel only loading.

Analyses
Structural Analysis...

- Change the analysis name to **Phase 1 (Steel only)** and click **OK**.
- In the  treeview right-click on **Loadcase 2**, select **Rename** and change the name to **Steel SW**.



### Deactivating the concrete

The surfaces that have concrete assigned to them need to be de-activated for this analysis and loadcase.



- Right-click in the view window and select **All Visible**.
- In the  treeview right-click on the material attribute **Concrete** and choose **Select Assignments**.
- On the Options dialog choose **Add to selection** and click **OK**.
- On the Clarify Assignment(s) to Act on dialog choose **Analysis 1** and click **OK**.
- From the  treeview drag and drop the attribute **Deact1** onto the selected features.
- On the attribute assignment dialog ensure **Assign to surfaces** is selected and choose the Analysis **Phase 1 (Steel only)** from the droplist and click **OK**.

The elements in the mesh layer that represent concrete are now no longer visible or active in this loadcase.

### Assigning gravity to the steelwork



- In the  treeview, right-click on **Steel** material and select **Set as Only Visible**.
- On the 'Clarify Assignment(s) to Act on' dialog choose **Analysis 1** and click **OK**.
- In the  treeview, right-click on loadcase **Steel SW** and select **Gravity**.

### Assigning the wet concrete loads

- In the  treeview, click on **Girder 1 – Top flange**, then hold-down the **Ctrl** key and add **Girder 4 – Top flange** to the initial selection. Right-click and choose **Select Members**.
- From the  treeview, drag and drop the **Slab external** load onto the model. Ensure analysis **Phase 1 (Steel only)** is selected and click the radio button for **Single loadcase**. From the Loadcase drop-list choose **New...**

## Staged Construction Modelling of a 3-Span Bridge Deck


---

- Ensure that ‘Automatically add gravity to this loadcase’ is not selected, and enter a loadcase name of **Wet slab**. Click **OK**.
- In the  treeview, click on **Girder 2– Top flange**, then hold-down the **Ctrl** key and add **Girder 3 – Top flange** to the initial selection. Right-click and choose **Select Members**. Click **Yes** to de-select previously selected members.
- From the  treeview, drag and drop the **Slab internal** load onto the model. Ensure analysis **Phase 1 (Steel only)** and loadcase **Wet slab** are selected and click **OK**.

### Phase 2 – Permanent and long-term loads

#### Permanent loads

A new analysis is required to consider long-term loading in the composite state. This analysis uses a different concrete material attribute than that used in the construction stage.



- Change the analysis name to **Phase 2** and click **OK**.
- In the  treeview right-click on **Loadcase 3**, select **Rename** and change the name to **Parapet**.
- In the view window, right-click and select **All Visible**

#### Defining a material for long term effects

The material attribute generated by the Steel Composite Bridge Wizard uses a concrete material with a Young’s modulus suitable for short term loading. A material suitable for long term loading must also be defined.


- Enter the value of Young’s modulus as **191E3**
- Enter the value of Poisson’s ratio as **0.2**
- Enter the value of Density as **4.662E-3**
- Change the name to **Concrete Long Term** and click **OK**.

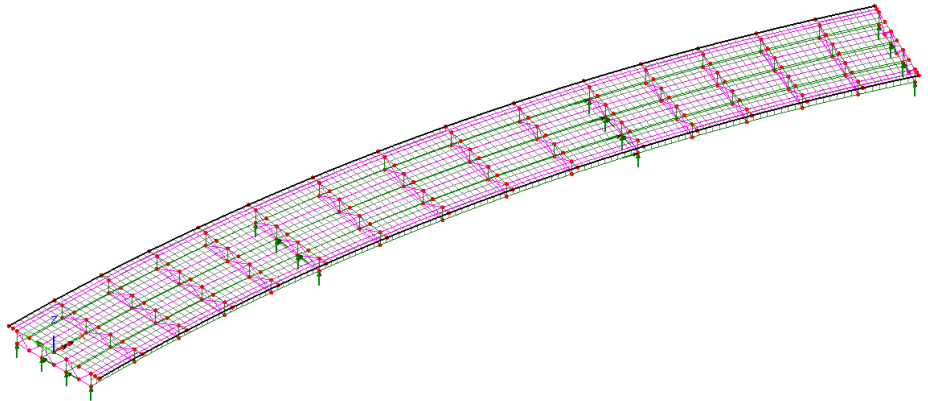
#### Assigning the long term material attribute


- In the  treeview right-click on the material attribute **Concrete** and choose **Select Assignments**.
- On the Options dialog choose **Clear selection** and click **OK**.
- On the Clarify Assignment(s) to Act on dialog choose **Analysis 1** and click **OK**.
- From the  treeview drag and drop the material attribute **Concrete long term** onto the selected features.

- On the attribute assignment dialog ensure **Assign to surfaces** is selected and choose the Analysis **Phase 2** from the droplist and click **OK**.




### Assigning Parapet loading

-  If necessary, turn-on the display of any loading.
- In the Groups treeview right-click on the **Parapets** group and choose **Select members**. Select **Yes** on the next dialog to deselect previously selected members.



- From the  treeview drag and drop the **Parapet** load attribute onto the selection.
- On the Loading Assignment dialog ensure that the analysis is set to **Phase 2** and select the **Single loadcase** radio button ensure the loadcase is set to **Parapet**. Click **OK**.

### Assigning Surfacing loading



- In the  treeview right-click on the **Concrete** material attribute and choose **Select Assignments**. If displayed, on the ‘Options’ dialog choose **Clear selection** and **Direct assignments** and click **OK**. (This selects only the surface features of the slab.)
- On the ‘Clarify assignment(s) to act on’ dialog choose **Analysis 1** and click **OK**.
- From the  treeview drag and drop the **Surfacing** load attribute onto the selection.
- Ensure **Phase 2** is selected and select the **Single loadcase** radio button and change (by over-typing) the loadcase name to **Surfacing** and click **OK**.
-  Turn-off the display of the surface loading.

### Phase 3 – Variable or short-term loading

#### Influence analysis

A Direct Method Influence analysis will be used to create influence surfaces for use with vehicle loading assessment. First, a search area needs to be defined to restrict the analysis to the deck.

#### Defining a search area for the deck

- Change the name to **Deck** and click **OK**.
- In the  treeview right-click on the **Concrete long term** material attribute and choose **Select Assignments**.
- On the Options dialog select **Clear selection** (if available – otherwise accept the default settings) and click **OK**.
- From the  treeview drag and drop the newly defined **Deck** search area onto the model to assign the attribute.
- Click anywhere in the view window to deselect the previous selection.

#### Define the Direct Method influence analysis.

- Choose **Deck** as the search area.
- Ensure that **Grid** is selected.
- Ensure that **Bridge centreline** is selected as the centreline.
- Set the transverse width to **30**.
- Change the name to **DMI\_1** and click **OK**.

#### Define Direct Method influence attributes

- Set the entity type to be **Beam/Shell Slice Resultants**.
- Select **My** as the Component of interest.
- Change the attribute name to **Moment** and click **Apply**.
- Change the Component to **Fz** and the name to **Shear** and click **OK**.

#### Assign the influence attributes


Influence attributes are generally assigned to selected node or points in a model, but in this example they will be assigned to a named composite design member, such as Girder 2. In doing so, influence locations will be generated at all the design locations specified on the composite bridge deck member ‘Girder 2’.


Attributes  
Search area


Analyses  
Direct Method  
Influence Analysis  
General...

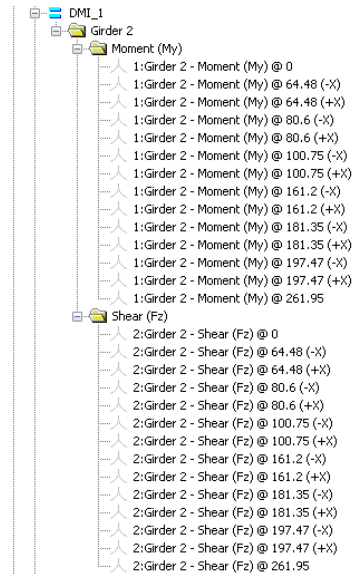
Attributes  
Influence  
Direct Method...



- With nothing selected in the view window, from the  treeview drag and drop the influence attribute **Moment** onto the model.
- On the Influence Assignment dialog choose **Girder 2**.
- Ensure that the Analysis is set to **DMI\_1**. Click **OK**.

The  treeview will be populated with the influence locations created.

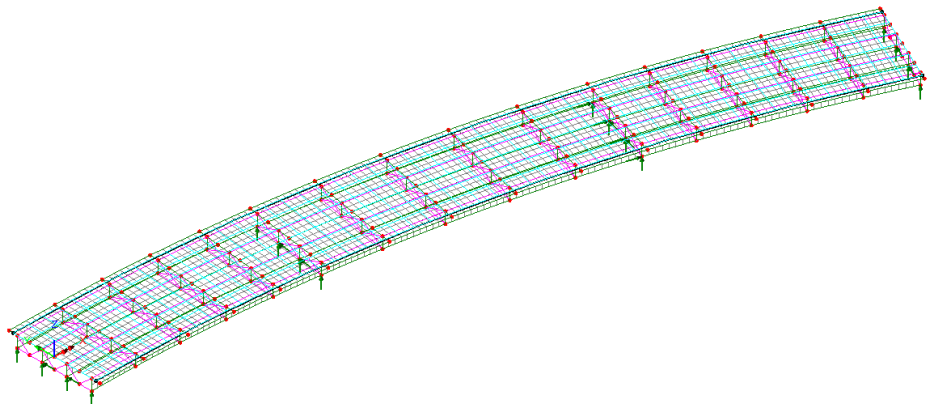
- From the  treeview drag and drop the influence attribute **Shear** onto the model.
- On the Influence Assignment dialog choose **Girder 2**.
- Ensure that the Analysis is set to **DMI\_1**. Click **OK**.



## Vehicle load optimisation

Traffic loading to AASHTO LRFD 7<sup>th</sup> Ed will be assessed using the LUSAS Vehicle Load Optimisation facility.

- First, right-click on the **Kerblines** group and **Select Members** to select the lines representing the extent of the carriageway.



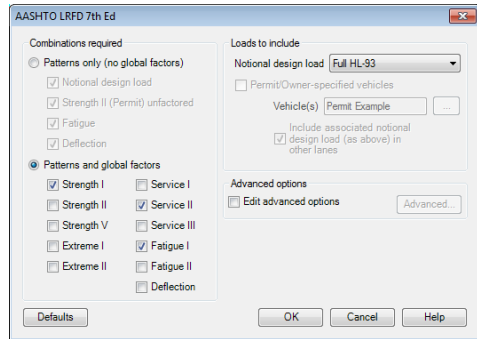
Bridge  
Vehicle Load  
Optimisation...

- Select **United States of America** from the Country drop-down list and choose **AASHTO LRFD 7th Ed** as the design code.

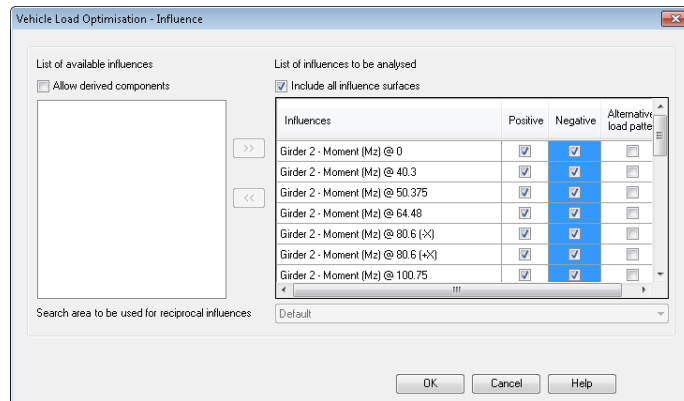
## Staged Construction Modelling of a 3-Span Bridge Deck

For simplicity, only Strength I, Service II and Fatigue I will be considered in this example.

- Click the **Optional code settings** button. On the dialog presented, select the **Patterns and global factors** radio button and ensure that only the **Strength I, Service II** and **Fatigue I** check boxes are selected.
- Click **OK** to exit the dialog.
- On the main VLO dialog, click the **Set influence surfaces** button

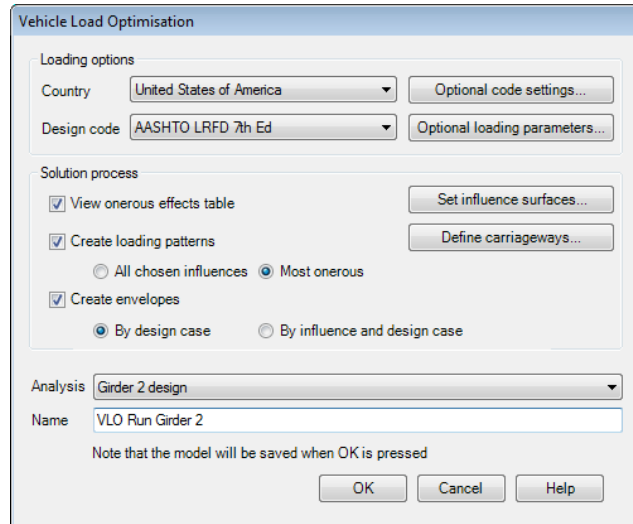


- On the Influence dialog select **Include all influence surfaces**. Ensure that the **Positive** and **Negative** checkboxes are selected for all included influences. Note that clicking in the header cell for a column will select all entries and allow all rows to be checked (ticked) with one click.



- Click **OK** to return to the main VLO dialog.
- On the main VLO dialog, click the **Define carriageways** button. Ensure **Kerbs from selection** is selected and click **Apply** to exit the dialog.
- On the main VLO dialog, select the **Create loading patterns** check box and select **Most onerous**.
- Select the **Create envelopes** check box and select **By design case**.
- For the analysis name, click the **New...** option and enter the name **Girder 2 Design** and click **OK**.

- Change the VLO run name to be **VLO Run Girder 2**



- Click **OK**. The model will be saved automatically, and then solved. The solution will take several minutes due to using slice resultants as the influence component.
- After the VLO analysis has run, re-select the model view.

## Running the Analysis


With the model loaded:



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

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



- ❑ **composite\_bridge\_deck\_staged~<analysis type>.out** these output files contains details of model data, assigned attributes and selected statistics of the analysis.

- **composite\_bridge\_deck\_staged~<analysis type>.mys** these are the LUSAS results files, which are 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, a replacement model file (provided) may be opened in order to solve and view results.

### Using a supplied model

If it proves impossible for you to correct the errors reported, or you wish to use a supplied model to allow the remainder of the example (showing how to use the design facility and obtain results) to be completed, see the instructions that follow.




- **composite\_bridge\_deck\_design\_ready.mdl** includes the definition of design members, creation of the construction stages and Vehicle Load Optimisation settings.



-  Copy the supplied read-only model file **composite\_bridge\_deck\_design\_ready.mdl** located in the `\<LUSAS Installation Folder>\Examples\Modeller` directory to your working directory.
-  Open the model **composite\_bridge\_deck\_design\_ready.mdl**
-  Open the Solve Now dialog. Ensure all items in the Solve treeview are selected (ticked) and press **OK** to run the analysis.

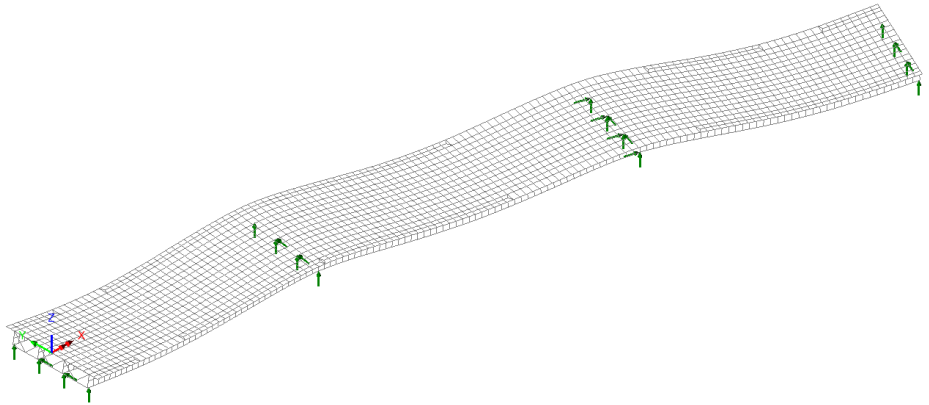
A LUSAS Datafile will be created from the model information. LUSAS Solver uses this datafile to perform the analysis.

## Viewing the Results


Analysis loadcase results are present in the  treeview. The last solved loadcase (Surfacing) will be set active by default.

An initial check will verify the deformed shape.

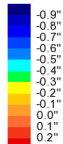
- In the  treeview turn off the display of the **Utilities**, **Geometry** and **Mesh** layers.
- In the  treeview right-click on the **DMI\_1** analysis names and deselect the Show grid option. This removes the blue loading grid from the display.
- With no features selected, right-click in a blank region of the view window and select **Deformed mesh**. Click **OK** to accept the default settings.



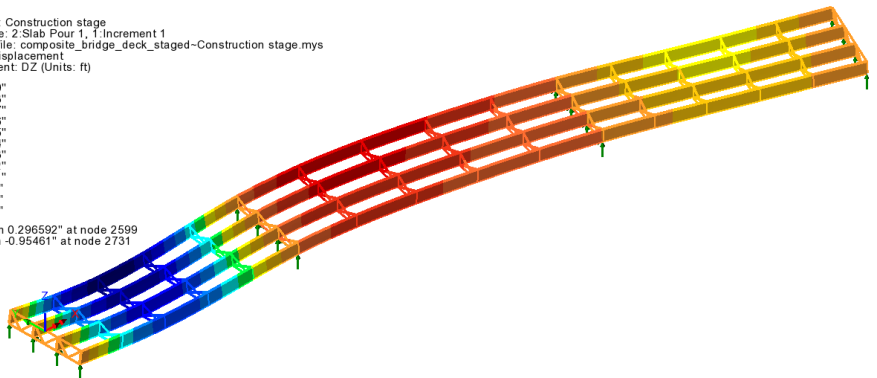
## Slab Pour 1

- In the  treeview expand the **Construction stage** analysis, right-click on loadcase **Slab Pour 1** and select **Set Active**
- In the view window, right-click and select the **Contours...** option. Select entity **Displacement** and component **DZ**. Select the **Contour Range** tab and set a contour **Interval** of **0.1/12** and click **OK**.

Analysis: Construction stage  
 Loadcase: 2.Slab Pour 1, Increment 1  
 Results file: composite\_bridge\_deck\_staged-Construction stage.mys  
 Entity: Displacement  
 Component: DZ (Units: ft)



Maximum 0.296592" at node 2599  
 Minimum -0.95461" at node 2731




## Staged Construction Modelling of a 3-Span Bridge Deck

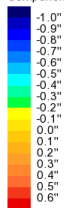


**Note.** Because model units are in feet the required contour interval of 0.1 must be divided by 12 to give inches.

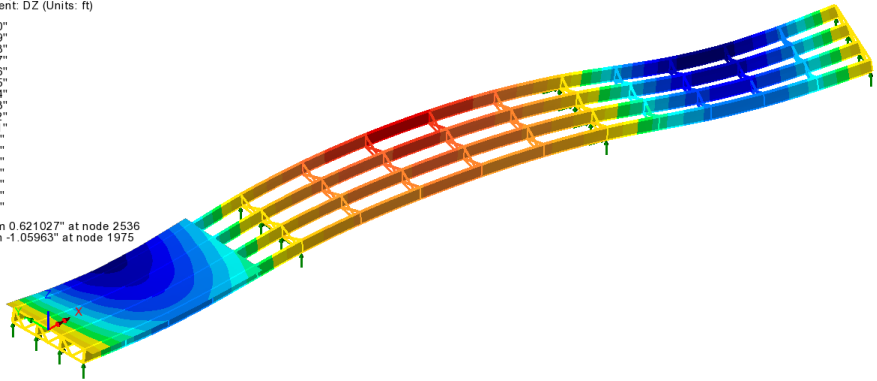
### Slab Pour 2

- In the  treeview expand the **Construction stage** analysis, right-click on loadcase **Slab Pour 2** and select **Set Active**

Analysis: Construction stage  
Loadcase: 3 Slab Pour 2, 2 Increment 2  
Results file: composite\_bridge\_deck\_staged-Construction stage.mys  
Entity: Displacement  
Component: DZ (Units: ft)



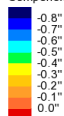
Maximum 0.621027" at node 2536  
Minimum -1.05963" at node 1975



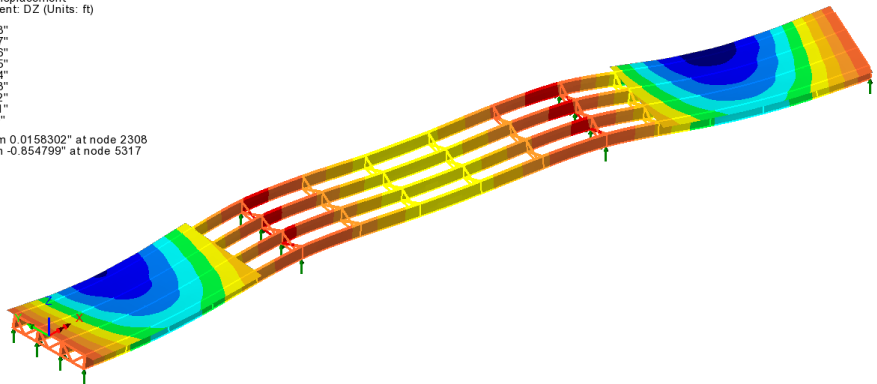
In this way, the results for the remaining slab pours can be viewed.

### Slab Pour 3

Analysis: Construction stage  
Loadcase: 4 Slab Pour 3, 3 Increment 3  
Results file: composite\_bridge\_deck\_staged-Construction stage.mys  
Entity: Displacement  
Component: DZ (Units: ft)

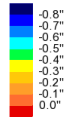


Maximum 0.0158302" at node 2308  
Minimum -0.854799" at node 5317

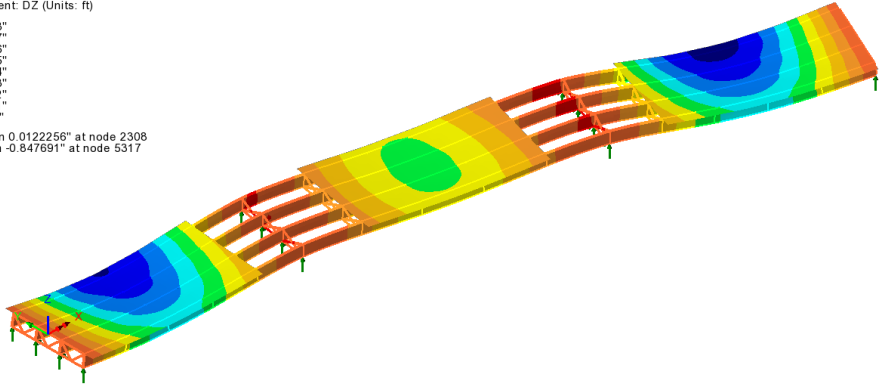


## Slab Pour 4

Analysis: Construction stage  
Loadcase: 5 Slab Pour 4, 4 Increment 4  
Results file: composite\_bridge\_deck\_staged-Construction stage.mys  
Entity: Displacement  
Component: DZ (Units: ft)

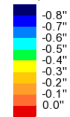


Maximum 0.0122256" at node 2308  
Minimum -0.847691" at node 5317

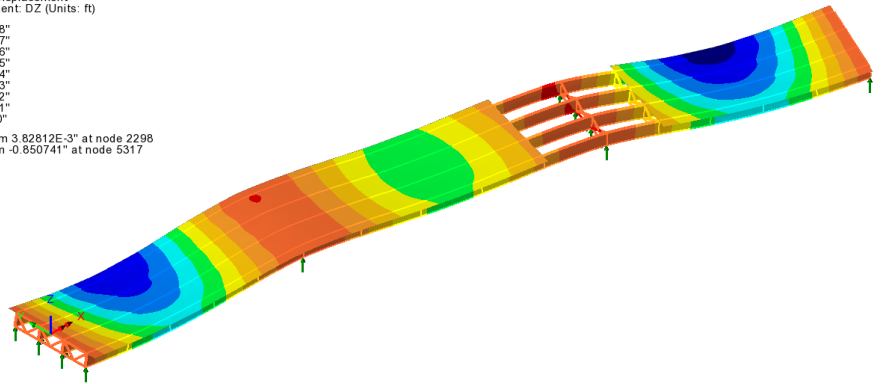


## Slab Pour 5

Analysis: Construction stage  
Loadcase: 6 Slab Pour 5, 5 Increment 5  
Results file: composite\_bridge\_deck\_staged-Construction stage.mys  
Entity: Displacement  
Component: DZ (Units: ft)

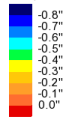


Maximum 3.82812E-3" at node 2298  
Minimum -0.850741" at node 5317

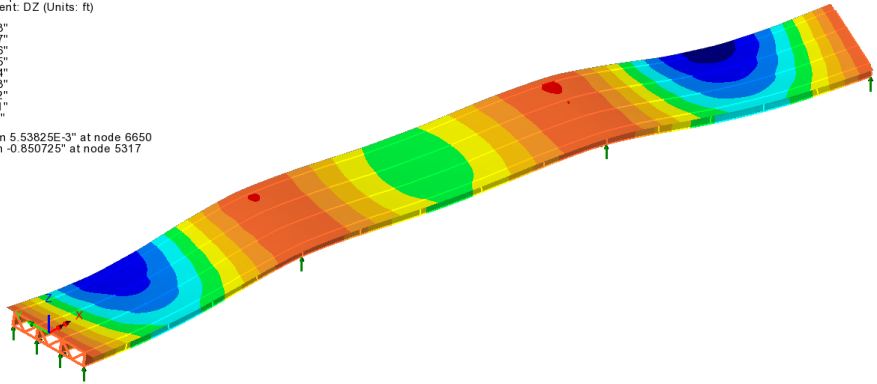


## Slab Full Self Weight

Analysis: Construction stage  
Loadcase: 7: Full self weight, 6: Increment 6  
Results file: composite\_bridge\_deck\_staged-Construction stage.mys  
Entity: Displacement  
Component: DZ (Units: ft)



Maximum 5.53825E-3" at node 6650  
Minimum -0.850725" at node 5317



Other loadcases can also be investigated in the same way.

This completes the example.