Composite Highway Bridge Design

In accordance with the Eurocode and the UK National Annex

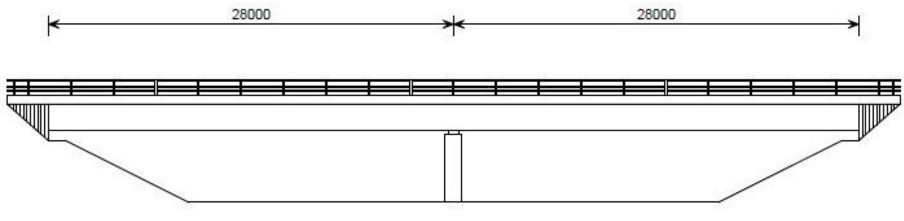
For LUSAS version:	21.1 or later
For software product(s):	LUSAS Bridge or LUSAS Bridge plus
With product option(s):	Steel and Composite Deck Designer (PontiEC4 – V3.5.2 or later) and Vehicle Load Optimisation
With additional software	Microsoft Excel installed

Description

The global analysis and design of a simple two span continuous bridge is to be carried out using the Steel and Composite Deck Designer (PontiEC4) and LUSAS Bridge.

The primary aim of this particular example is to show the LUSAS model building requirements to allow a design check with PontiEC4 to be carried out. A separate example 'Assessment of a Composite Bridge Deck to Eurocodes' shows the use of the PontiEC4 Composite Deck Designer and the interaction between it and a supplied LUSAS model.

The bridge carries a 2-lane highway over another road.



A four-girder arrangement has been chosen, and a deck slab thickness of 250 mm has been assumed. The girders are spliced at a distance of 6.3m either side of the central pier, giving overall girder lengths of 21.7, 12.6 and 21.7m respectively.

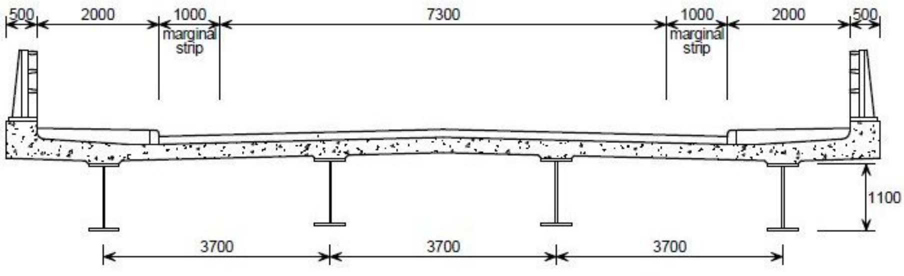
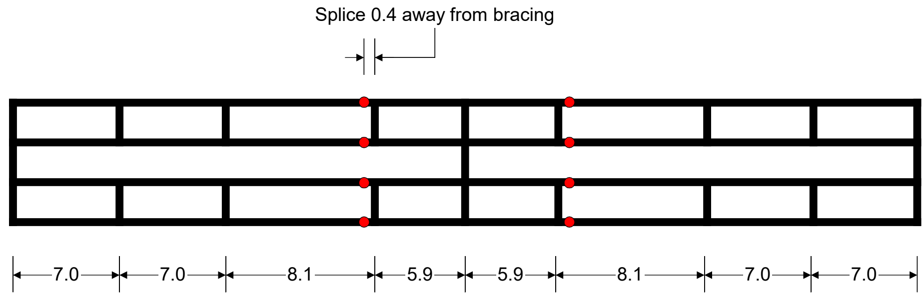


Table of main girder properties

	21.7 m span girder	**12.6 m pier girder**	**21.7 m span girder**
Top flange	500 x 25	500 x 40	500 x 25
Web	12	18	12
Bottom flange	500 x 50	650 x 60	500 x 50
Top rebars	B16 @ 150mm	B25 @ 150mm	B16 @ 150mm
Bottom rebars	B16 @ 150mm	B25 @ 150mm	B16 @ 150mm

Bracing arrangement



Objectives

This shows how the Steel and Composite Deck Designer (PontiEC4) and LUSAS Bridge can be used to carry out the analysis and design of the bridge shown previously.

Keywords

Steel, Composite, Deck, Designer, Vehicle Load Optimisation, PontiEC4, Design, Checking, Import, Export, ULS, SLS, Detailed design.

Associated files

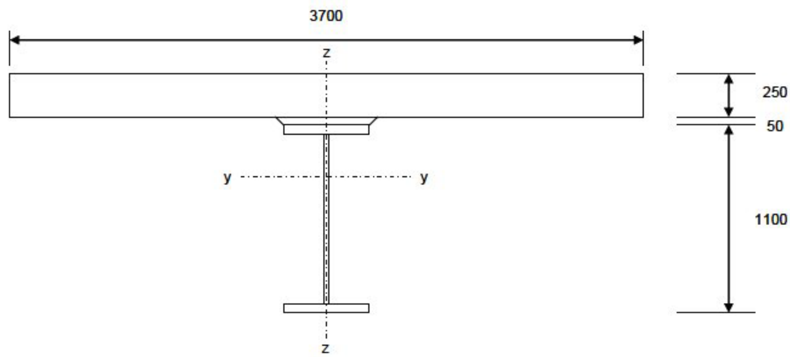
These are provided as part of a PONTI EC4 installation

- **LUSAS_Comp_Bridge.csv** - Contains initial material and geometric data / definitions for use with PontiEC4
- **Sections_Rev0.vbs** - A file created by PontiEC4 containing geometric section data to import into LUSAS. A supplied file is only for use in case of difficulties in manually creating the appropriate data.
- **LUSAS_Shrinkage.vbs** - A file created by PontiEC4. A supplied file is only for use in case of difficulties in manually creating the appropriate data.
- **LUSAS_Thermal.vbs** - A file created by PontiEC4. A supplied file is only for use in case of difficulties in manually creating the appropriate data.
- **LUSAS_Comp_Bridge.mdl** – Model file created by a user. A supplied LUSAS model file is only for use in case of difficulties in manually creating the appropriate model.
- **Forces_and_moments.xls** – File created by the LUSAS Export Forces and Moments dialog containing data to import into PontiEC4. A supplied file is only for use in case of difficulties in manually creating the appropriate data.

- **LUSAS_Comp_Bridge_Results.csv** – Contains revised geometric data / definitions for use with PontiEC4
- **Forces_and_moments_optimised.xls** – File created from a modified LUSAS analysis model (not supplied) containing data to import into PontiEC4.

Structural Idealisation

Note. It is understood that the bridge should have different section properties for the internal girders and the external girders due the differences in widths of slab associated with each girder. This is due to the cantilever being shorter but thicker than half the width of slab between girders. So, in reality, the analysis should have different section properties for both the internal and external girders. However for training purposes the section properties will be based on the cross section shown below. This is a gross simplification but has been done to keep the data creation simple for this example.



For the purposes of this example, only two locations will be considered for detailed design. In reality many locations would need to be considered to cover the full design of the bridge, but for training purposes this simplification will be accepted.

Design checks will be carried out:

- Along girder 2 (the first internal girder), towards the middle of span 1
- Over the pier support.

Part 1: Enter section and material data in Steel and Composite Deck Designer (PontiEC4)

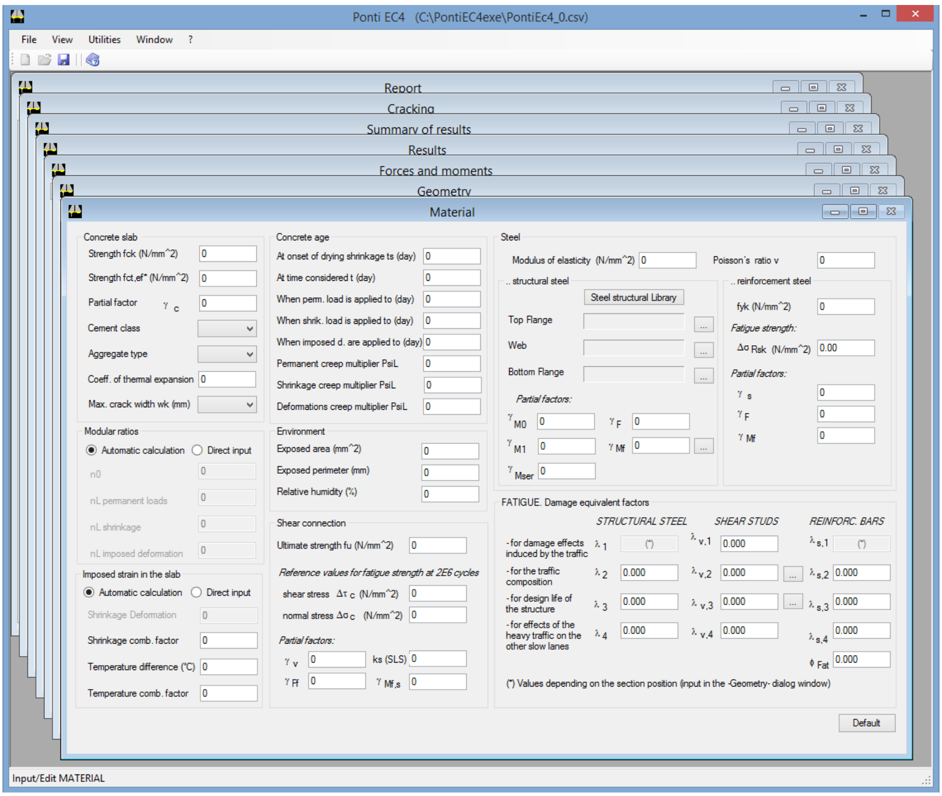
Running Steel and Composite Deck Designer (PontiEC4)

- Run the Steel and Composite Deck Designer (**PontiEC4**)

- Select the **File> New** menu item to create a new project.
- Select the **File> Save as** menu item to save the Steel and Composite Deck Designer (PontiEC4) dialog data to a working project folder different from the installation folder (this would normally be a project folder in which a LUSAS model will also be created), with the name **LUSAS_Comp_Bridge.csv**

Note. All data entered in the Steel and Composite Deck Designer (PontiEC4) dialogs will be saved to this file. Previously created and saved data can be loaded into the Steel and Composite Deck Designer (PontiEC4) dialogs by using the **File> Open** menu item. If this example is undertaken over several sessions, ensure that the entered data is saved before exiting so that the example can be picked up from where it was left.

Having started the Steel and Composite Deck Designer (PontiEC4) a number of blank dialogs forms will be presented cascading across the screen as shown below. Through the design process each of these dialogs will be populated with data, starting with the **Material** dialog.



Note. Under the **Windows** menu a list of all the dialog names are presented. Clicking a dialog name in the list will bring that dialog to the front of the screen. This allows for easy navigation around the various dialogs.

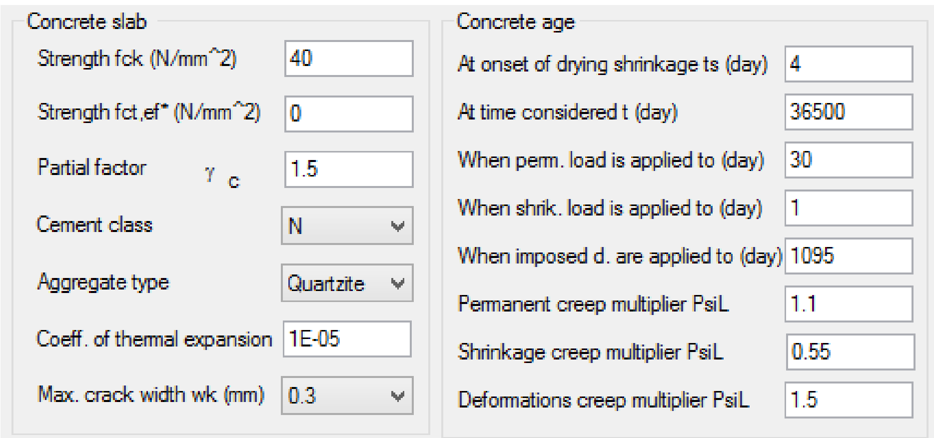
In the Steel and Composite Deck Designer (PontiEC4) the units in use are always indicated in the input and output dialogs, and in general forces are expressed in N (Newton), lengths in mm (millimetres) or m (metres), and moments expressed in Nm (Newton metres). The units used for the LUSAS model must therefore be N, m, kg, s, C.

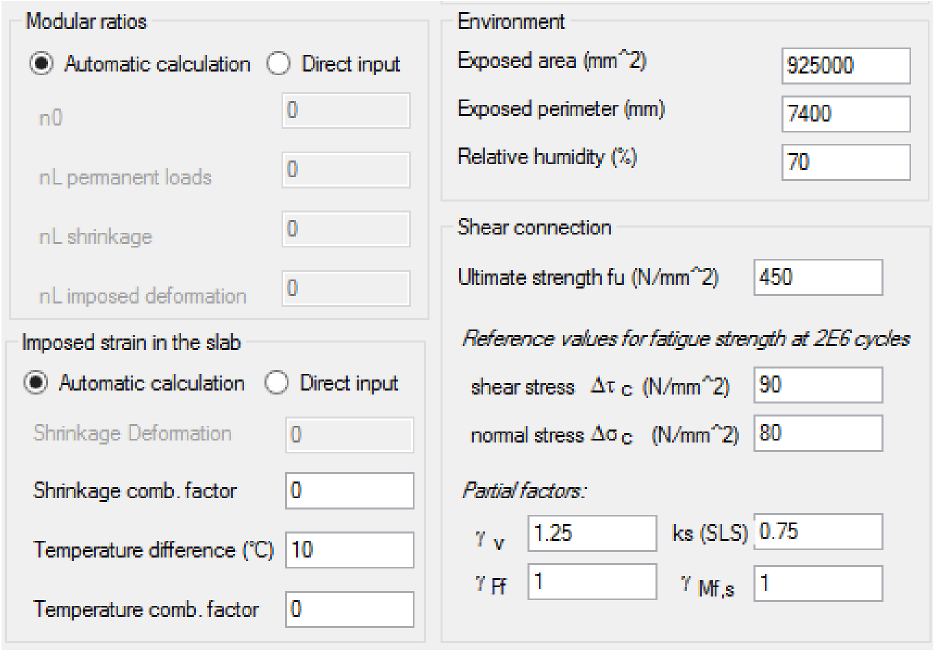
Defining material properties

The material dialog is immediately available when you start a new project or when you open an existing one. The material dialog is organized into separate panels; Concrete, Concrete age, Steel etc.

- For each panel on the dialog enter the material data as shown in the following images.

Concrete Material Properties

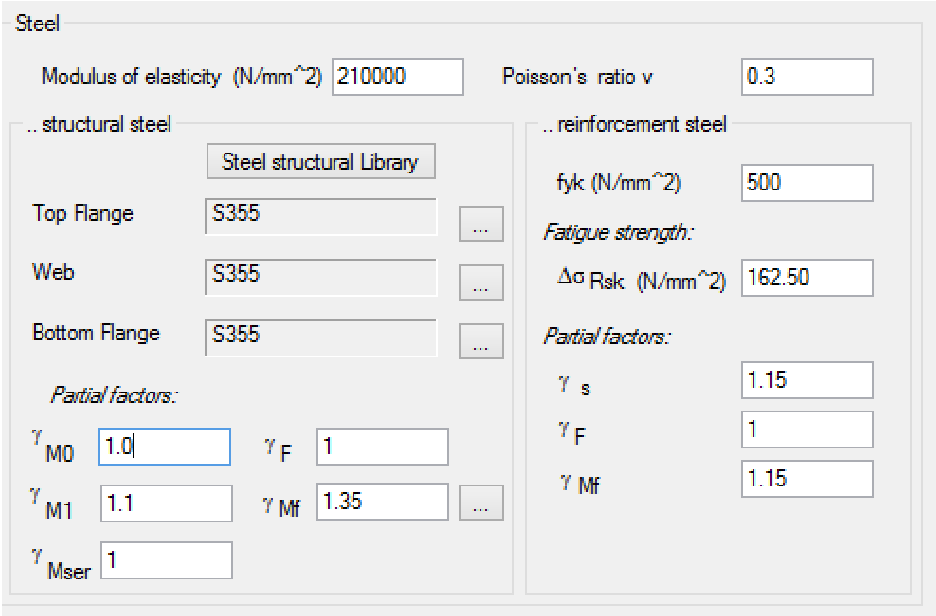




Note. The Steel and Composite Deck Designer (**PontiEC4**) help system contains commentary relating to the information being entered including references to the Eurocode in question.

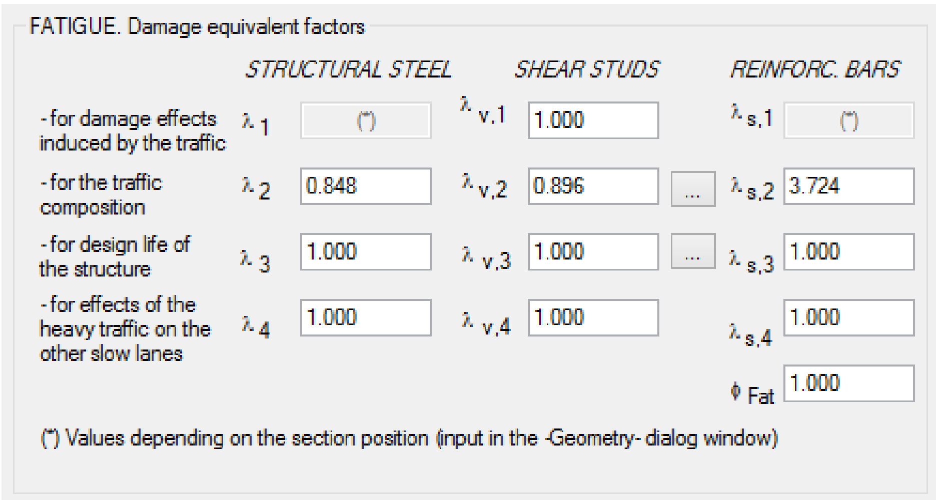
Steel Material Properties

With the Materials dialog still selected, set the following data for the steel properties:



- Click on the ellipsis button  next to the Grade field, to define a Steel grade of **EN 10025-2 S355.**

Fatigue



For the fatigue factors:

- λ_2 can be calculated with data input in the window dialog that appears when you click on the ... button next to the $\lambda_{v,2}$ input field. Select **Traffic category 2** and the heavy vehicle distribution to **Medium distance** and click **OK** to return to the main dialog.
- λ_3 can be calculated with data input in the window dialog that appears when you click on the ... button next to the $\lambda_{v,3}$ input field. Set the design life of the bridge as **100 years** and click **OK** to return to the main dialog.
- Enter the other factors as shown on the dialog above.

Save the data entered

- Select the **File> Save** menu item to save the PontiEC4 dialog data to the current working project folder.

Defining Geometric properties

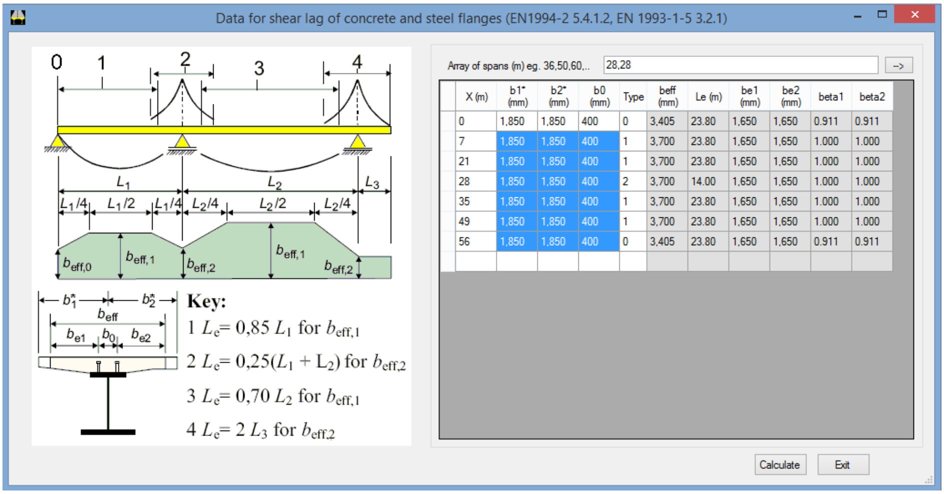
Prior to defining the geometric section properties for the spans, the span lengths and the effective concrete slab width (beff) of each section, defined at various distances along the section, need to be stated.

Defining shear lag slab and flanges

- To input data for the effective slab width calculation, select the **Utilities> Shear lag slab and flanges** menu item.
- Input an array of the bridge spans (the span lengths) in the text box in the top right corner: **28.0, 28.0** and click on the --> button. The X distance where the effective slab width changes will be automatically set up in the table.
- For each row input the flange widths b1*, b2*, b0 as **1850, 1850, 400** respectively.

Note. Values in the table can be selected and copied and pasted into other cells by using the context menu.

- In the **Type** column ensure that the code **0** is used for the first and last section, **1** for the sagging regions, and **2** for the hogging regions over supports.
- Click the **Calculate** button to calculate $\mathbf{b_{eff}}$



- Click the **Exit** button to close the dialog and return to the material dialog.

Note. Values of effective length Le are used to calculate the effective width due to shear lag effects in the slab and the steel flanges. Also once calculated, the numbers are shown with a comma as the thousand separator, so that 1850 becomes 1,850.

Defining geometric data for the bridge

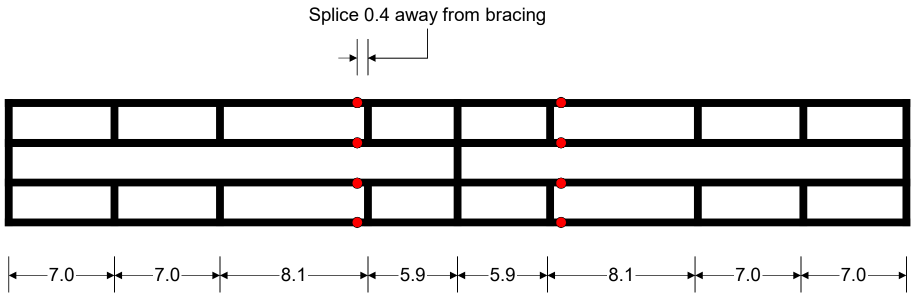
The geometric data for sections along the bridge now needs to be defined. Note at this stage it is not necessary to define all the sections in each segment that will be checked; it will be enough to define just one section for each segment and input a distance to automatically obtain the corresponding effective width of slab from the shear lag slab table previously defined.

To establish the initial section locations some thought needs to be given to the analysis model that will be used.

Analysis model concept layout

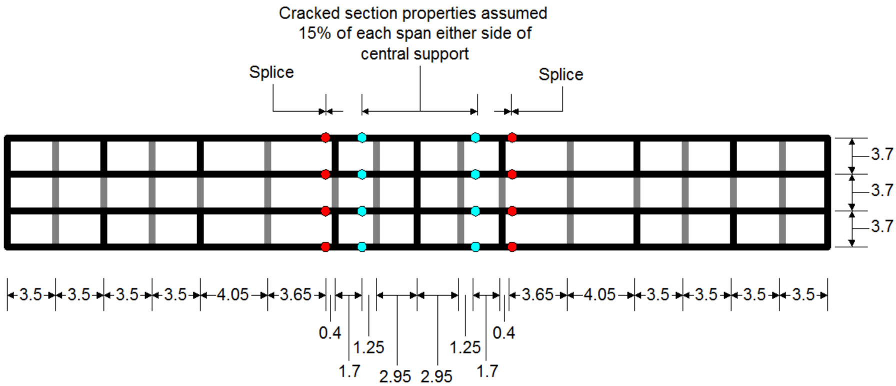
The bridge is to be analysed using the planar grillage analogy with thick beam elements. Although grillage analysis is not the most rigorous analysis type and is unlikely to model the effects of bracing correctly, the ability to obtain simple results from beam elements with composite sections (transformed section properties), means that it is still widely used for bridge analysis.

The basic girder layout is shown below including splice and bracing locations.



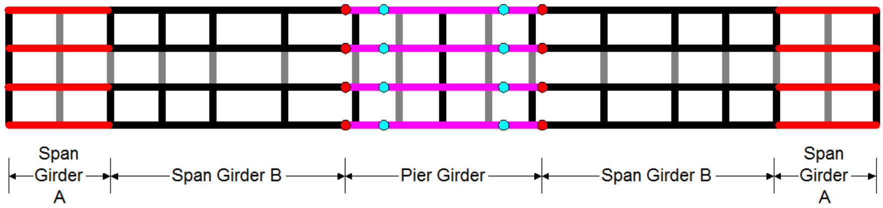
As per EN1004-2 clause 5.4.2.3 (3), the region over the internal supports, cracked section properties will be assumed to exist for a length equal to 15% of each span, either side of the central supports.

Following the basic rules for setting up a grillage analysis, the layout shown below is going to be considered. The virtual grid is formed from lines that follow the original girder and bracing locations (shown in black in the following image), and lines representing the additional transverse members (shown in grey) to ensure a similar longitudinal and traverse member spacing. Points will also need to be added at the splice locations and at the start of the cracked section properties.



¶

So, for the basic geometric sections in the grillage, only four sections will be used: Span Girder A, Span Girder B and Pier Girder (cracked and uncracked) as shown on the following image:



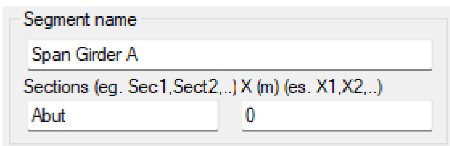
Defining geometric data for Span Girder A

The geometric data for each segment and/or section along the bridge now needs to be defined, starting with Span Girder A.

Note. A girder may be formed / analysed from any number of **Segments** (a length of girder over which all parameters - apart from the total height of steel section and the attributed concrete slab width -remain constant) that are labelled such as Span Girder A, Span Girder B, etc. **Sections** are locations defined within a segment at which geometric properties are defined or calculated. Both terms are used throughout this example.

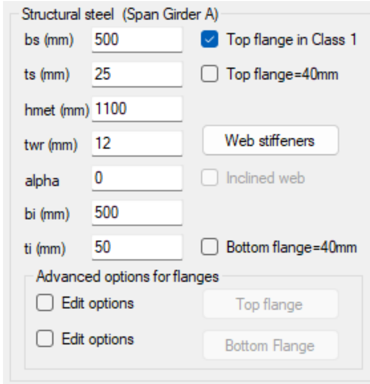
Segment name, sections, X-distances

- From the main menu select **Windows> 2 Geometry** to display the Geometry dialog.
- In the Segment name panel enter the following:



Steel section sizes

In the Structural steel panel enter the following:

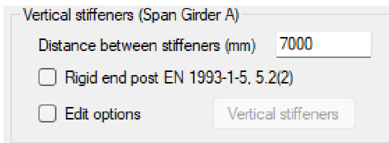


- Make sure that **Top flange in Class 1** is checked (ticked). There is no local buckling because it is attached to the concrete slab.
- Make sure that **Top Flange=40mm** is <u>not</u> checked (ticked)

Note. The checkboxes for Top / Bottom flange ≤ 40mm are only required to be checked if the flanges have total thickness greater than 40mm but are formed from multiple plates each of a thickness less than 40mm. When checked (ticked), ultimate and yielding tension for steel equal to 40mm thickness will be used.

Vertical web stiffener distance

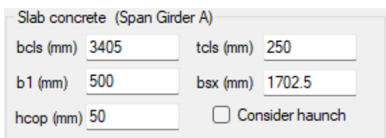
- In the Vertical stiffeners panel enter the following:



Note. A vertical stiffeners check is optional and is not carried out in the example. Specifying the distance between vertical stiffeners is the only mandatory value in this panel.

Slab concrete

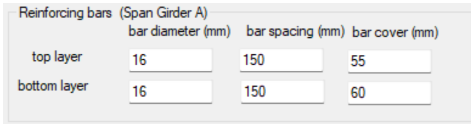
- In the Slab concrete panel, **right-click inside the bcls field** and select **Calculate beff.** The width of slab will be computed using data previously defined in the shear lag table. Input other data as shown in the following image.



For this example, the section properties of the concrete haunch will be ignored in any calculations that are made, so it is left un-checked.

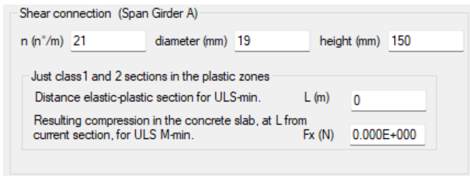
Reinforcing bars

- In the reinforcing bars panel enter the following:



Shear connection

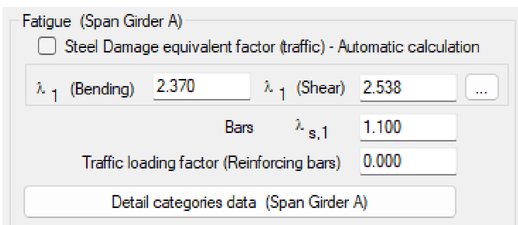
- In the Shear connection panel enter the following:



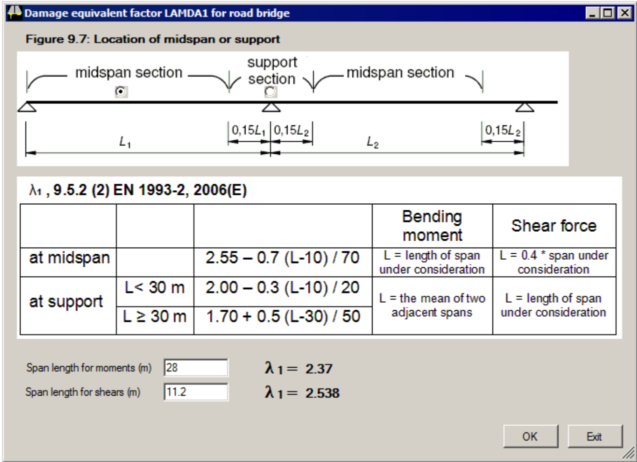
Note. The values for 'Distance elastic-plastic section for ULS-min' and 'Resulting compression in the concrete slab, at L from current section for ULS M-min' cannot be determined this early in the design process and are therefore defined as zero for now. These values must be revisited after the bridge has been analysed and design calculations performed.

Fatigue data

- In the Fatigue panel enter the following:

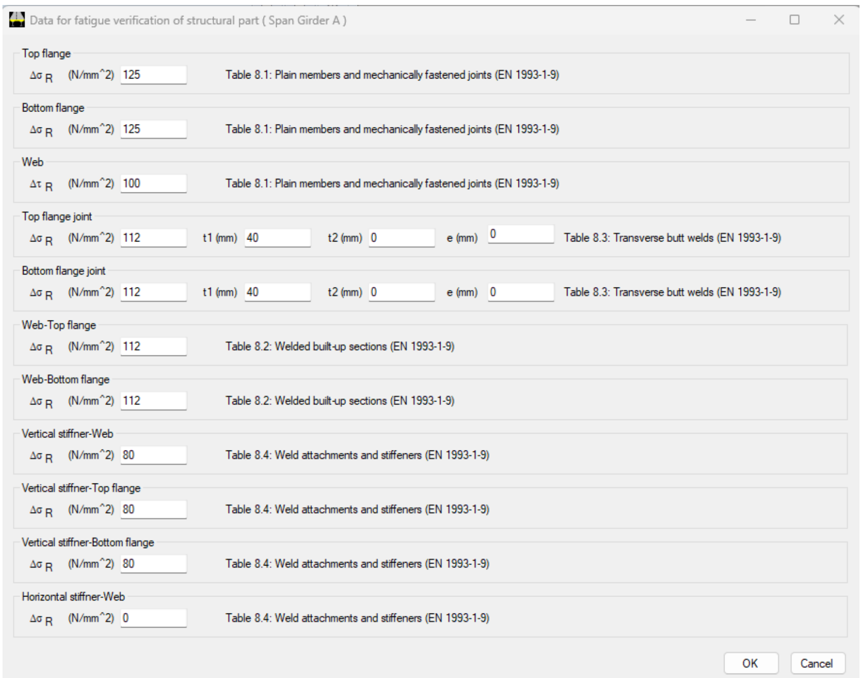


- λ_1 can be calculated automatically by clicking the ... button next to the input field and then, on the Damage equivalent factor dialog, selecting the radio button for the **midspan section** and entering **28.0** for the Span length for moments and **11.2** for the Span length for shears.



Detail categories

- In the Fatigue data panel, select the **Detail categories data** button and define the following data, noting that no value is required for Horizontal stiffener-Web.



Adding data for Span Girder A to the segment treeview

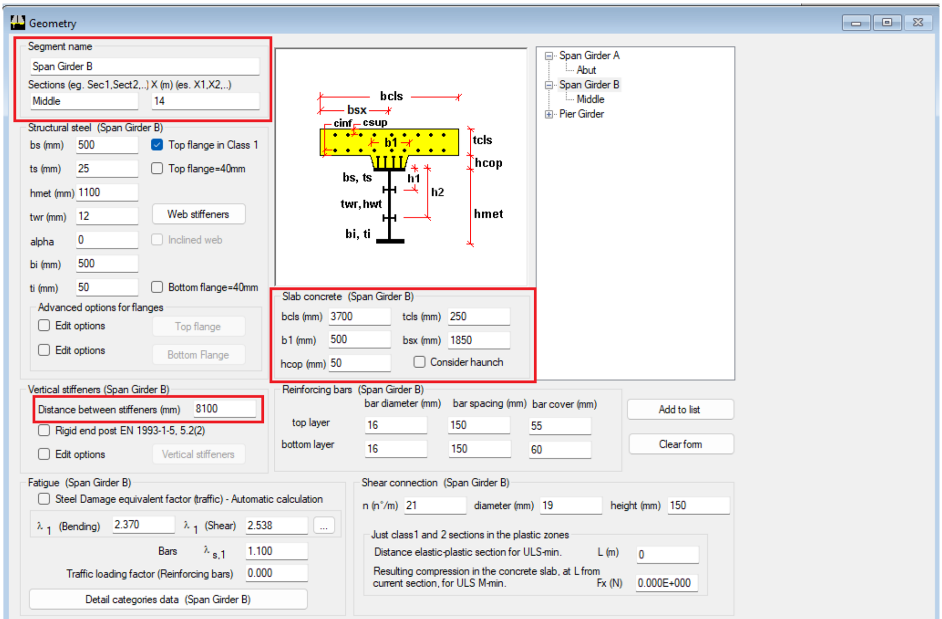
- Click on the **Add to list** button located under the segment treeview to save all the data entered for Span Girder A.
- Before entering data for other segments, in the segment treeview **double-click** on the name of the Span Girder **A**. All data previously input for the previous section is now available for use in defining the next section.

Note. Double-clicking on a segment name in the Segment treeview panel will switch PontiEC4 from EDIT MODE to INPUT MODE. The status bar at the bottom-left of the screen will always show what mode is being used at any time.

Defining geometry for Span Girder B

With the status bar stating 'INPUT GEOMETRY:'

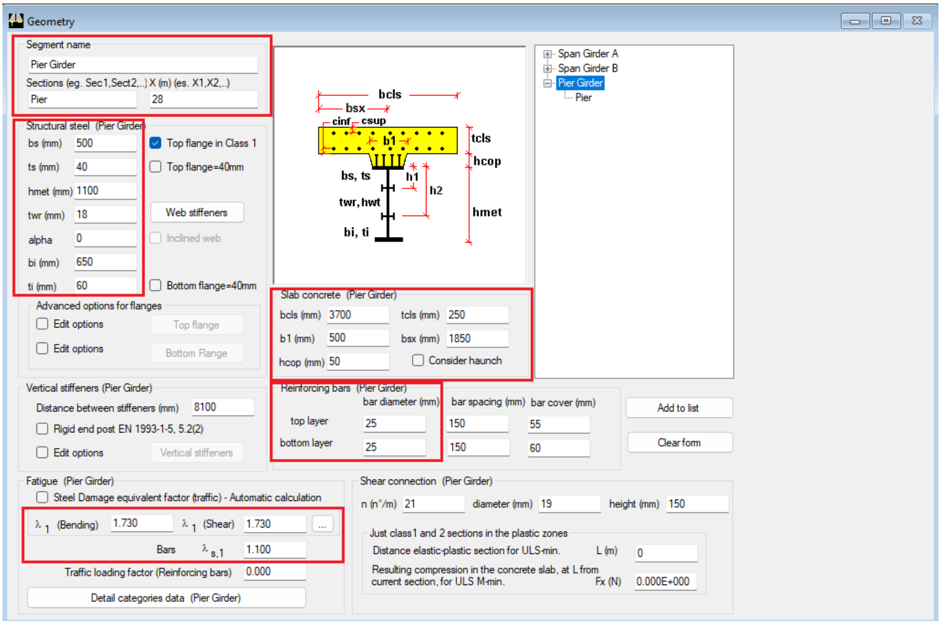
- In the Segment name field delete **Span Girder A** and type **Span Girder B**.
- Change the Section name to **Middle** and Distance **X** to be **14**
- In the Vertical stiffeners panel, set the distance between stiffeners to **8100**
- In the Slab concrete panel, re-calculate the values of bcls and bsx by selecting **Calculate beff** from the context menu for the bcls cell.



- Click on the **Add to list** button located under the segment treeview to save all the data entered for Span Girder B.
- Before entering data for other segments, in the segment treeview **double-click** on the name of the **Span Girder A**. All data previously input for this section is now available for use in defining the next section.

Defining geometric data for the Pier Girder

- In the Segment name field delete **Span Girder A** and type **Pier Girder**.
- Change the Section name to **Pier** and the Distance **X** to be **28**
- In the Structural steel panel, change the top flange thickness **ts** to **40**
- Change the Web thickness **twr** to **18**
- Change the Bottom flange width **bi** to **650** and thickness **ti** to **60**
- In the Vertical stiffeners panel, set the distance between stiffeners to be **8100**
- In the Slab concrete panel, re-calculate the values of bcls and bsx. Remember that the values of bcls and bsx can be calculated automatically by selecting **Calculate beff** from the context menu for both fields.
- In the Reinforcing bars panel, set the top layer reinforcement **bar diameter** to **25** and bottom layer reinforcement **bar diameter** to **25**
- In the Fatigue panel, λ_1 can be calculated automatically by clicking the ellipsis button (…) next to the input field and then, on the Damage equivalent factor dialog, select the radio button for the **support section** and enter **28** for the Span length for moments and **28** for the Span length for shears.



- Click on the **Add to list** button located under the segment treeview to save all the data entered for the Pier Girder.

Save the entered data

- Select the **File> Save** menu item to save the PontiEC4 dialog data to the current working project folder.

Part 2: Export properties from Steel and Composite Deck Designer PontiEC4 to LUSAS Bridge

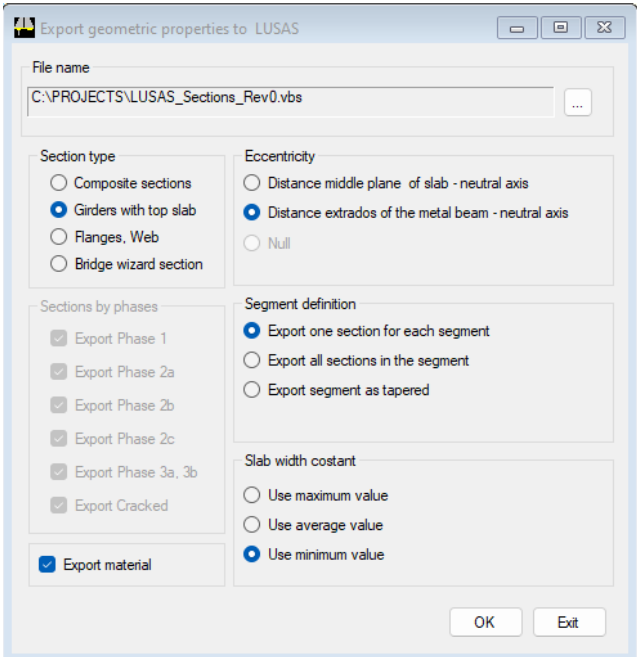
Associated Files

These supplied files are only for use in case of problems in manually creating the appropriate data in this section of the example.

- ❑ **LUSAS_Comp_Bridge.csv** - the input file for PontiEC4 with data on sections and materials.

Prior to exporting geometric properties, the Steel and Composite Deck Designer (PontiEC4) needs to check that all the data that has been input is valid, and then run part of the EC4 code check to calculate the required geometric properties of each defined cross section. This is automatically done when requesting to view the Results window/dialog and should always be done prior to exporting properties subsequently.

- On the main Steel and Composite Deck Designer (PontiEC4) menu select **Window> 4 Results** to set the **Results** dialog active. After a short delay whilst a partial code check is run, the geometric properties of each cross section will be calculated. This can be confirmed by selecting a Cross-section and Design combination with the **Geometric properties 2** tab active.
- When complete, select **File> Export geometric properties to LUSAS**



- On the dialog, browse to your working project folder and enter a filename of **"LUSAS_Sections_Rev0.vbs".**
- Select **Girders with top slab** and check the **Export material** check box.
- Ensure that the option 'Distance middle plane of slab – neutral axis' is not selected
- Ensure that the option **Assume slab width constant and use minimum value** is selected. Click **OK** to finish.

Phases explained

- ❑ **Phase 1** – Steel sections only, non-composite action. Slab will be considered only as load.
- ❑ **Phase 2a** - The composite sections with **long term** properties of the concrete. Loadings considered are the **permanent** ones.
- ❑ **Phase 2b** - The composite sections with **long term** properties of the concrete. Loading considered is the **shrinkage of the concrete**.
- ❑ **Phase 2c** – this represents the actions arising from prestressing and is therefore not required in this example.

- **Phase 3** - The composite sections with **short term** properties of the concrete. Loading considered are the **variable actions.**

Next, the equivalent loading for the shrinkage effects can be exported.

- Select **File> Export equivalent DT to LUSAS**
- Browse to the working project folder and enter a filename of **"LUSAS_Shrinkage.vbs".**
- Select the **Shrinkage** option and Click **OK** to finish.

Finally, the loading for the temperature effects can be exported.

- Select **File> Export equivalent DT to LUSAS**
- Browse to the working project folder and enter a filename of **"LUSAS_Thermal.vbs".**
- Select the **Thermal** option and Click **OK** to finish.

Part 3: Building the LUSAS analysis model

Running LUSAS Modeller

For details of how to run LUSAS Modeller, see the heading ***Running LUSAS Modeller*** in the Examples Manual Introduction.

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 **LUSAS_Comp_Bridge**
- Set the working folder to be the same as that used for the PontiEC4 dialog data.
- Enter the title as **Composite highway bridge design**
- Select units of **N,m,kg,s,C**

Caution. It is important that the units used in the Steel and Composite Deck Designer (**PontiEC4**) are also specified for LUSAS model. If consistent units are not used, the results obtained will be incorrect.

- Leave timescale units as **Seconds**

- Ensure the **Structural** analysis type is selected.
- Set an analysis category of **2D Grillage / Plate**
- Leave the startup template set to **None**
- Click the **OK** button.

Note. Use the Undo button to correct any mistakes made since the last save was done.

Import of section properties

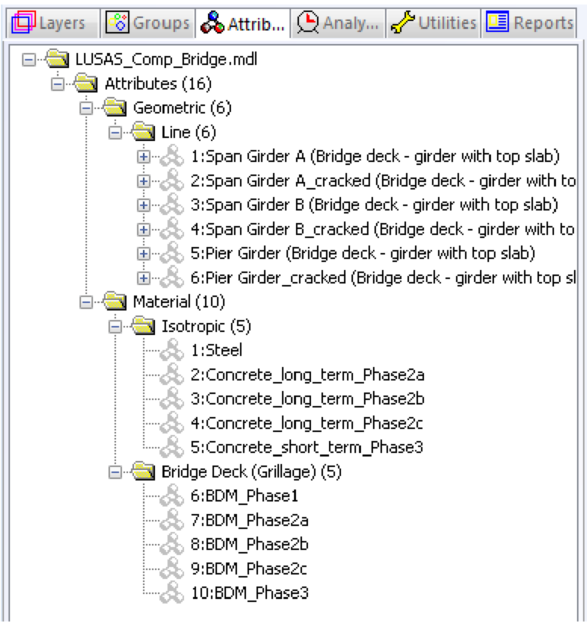
The section properties exported from Steel and Composite Deck Designer (**PontiEC4**) need to be imported into LUSAS.

File
Script >
Run Script...

Locate the "**LUSAS_Sections_Rev0.vbs**" and open in LUSAS.

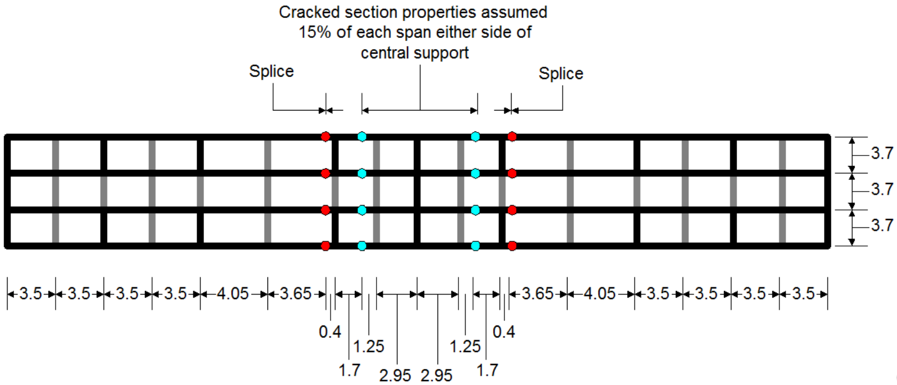
- Select the file **LUSAS_Sections_Rev0.vbs** and click **OK**.

This will populate the Attributes treeview with the geometric section properties and material properties exported from Steel and Composite Deck Designer (PontiEC4) as shown below.



Now that the geometric and material properties have been imported, the analysis model that will use them can be constructed.

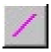
As discussed earlier in this example a simple grillage analogy will be used for the analysis model. The grillage wizard could be used but in this example the grillage will be constructed manually based on the layout and dimensions shown below.



¶

Defining the base geometry

Geometry
Line >
Coordinates…

 Enter coordinates of (**0, 0**), and (**3.5, 0**), to define the first Line of the grillage.

- Select the Line that has just drawn.

Geometry
Line >
Copy…

 Copy the Line through a translation of **X= 3.5, Y= 0, Z = 0,** set the number of copies to **3** and click **OK** to create further Lines.

- Select the Point at the right hand end of the Line that is furthest to the right.

Geometry
Line >
By Sweeping…

 Sweep the Point through a translation of **X= 4.05, Y= 0, Z = 0** and click **OK** to create a Line.

Geometry
Line >
By Sweeping…

 Repeat the process of selecting the Point at the right hand end of the Line that is furthest to the right and sweeping using the following translations:

X= 3.65, Y= 0, Z = 0

X= 0.4, Y= 0, Z = 0

X= 1.7, Y= 0, Z = 0

X= 1.25, Y= 0, Z = 0

X= 2.95, Y= 0, Z = 0

X= 2.95, Y= 0, Z = 0

X= 1.25, Y= 0, Z = 0

X= 1.7, Y= 0, Z = 0

X= 0.4, Y= 0, Z = 0

X= 3.65, Y= 0, Z = 0

X= 4.05, Y= 0, Z = 0

X= 3.5, Y= 0, Z = 0

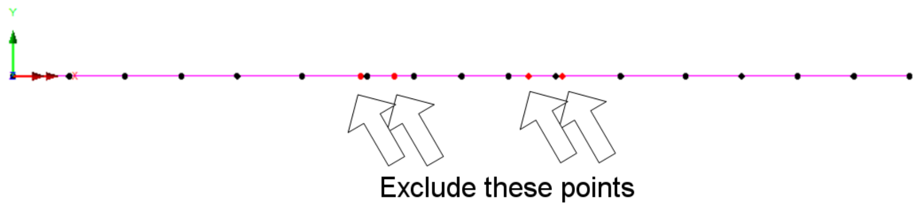
- Select the line furthest to the right.

Geometry
Line >
Copy…

Copy the Line through a translation of **X= 3.5, Y= 0, Z = 0,** set the number of copies to **3** and click **OK** to create further Lines.

- Select all the Points in the model except the four points that represent the splice locations and where the cracked section properties start – as shown by arrows on the image that follows.

Tip. Select the 'Select Points' cursor and use the Shift key to add to each set of selections made.



Geometry
Line >
By Sweeping…

Sweep the selected Points through a translation of **X= 0, Y= 3.7, Z = 0** and click **OK** to create Lines.

- Ensure the 'Select Any' cursor is back in use.

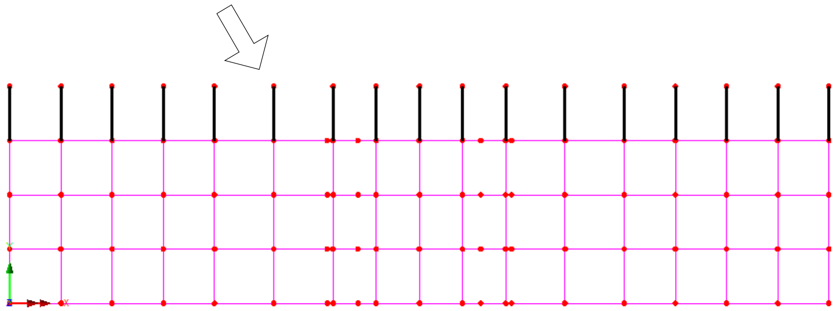
- Press **Control+A** to select everything on the screen, all the lines.

Geometry
Line >
Copy…

Copy the Lines through a translation of **X= 0, Y= 3.7, Z = 0,** set the number of copies to **3** and click **OK** to create further Lines.

- Select the additional unwanted Lines shown below.

Tip. Hold down the Alt key whilst dragging a selection box through the extent of all the lines.



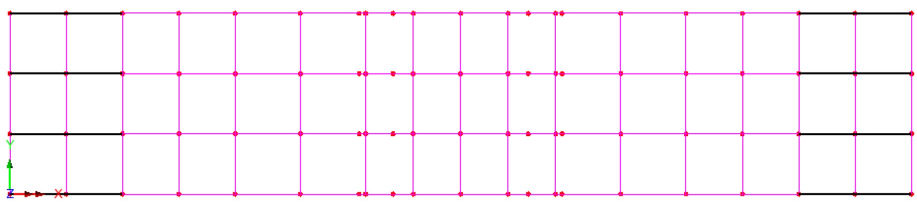
 Delete the selected Lines, confirming that Lines and Points are to be deleted.

Defining groups

Model features will be grouped together to make assignment of attributes and viewing of model results easier. Lines representing the different geometric sections will be formed into named groups.

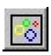
Span girder A

- Select the eight lines at each end of the deck which will have **Span Girder A** properties assigned to them.



Tip. Select the 'Select X axis' cursor and box-select the lines at each end of the deck. Use the Shift key to add to the previous selection made.

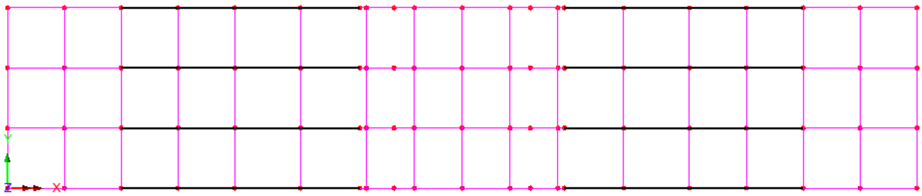
Geometry
Group >
New Group

 This adds a New Group entry in the  Treeview for the features selected.

- Enter the group name as **Span Girder A** and click **OK** to finish defining the group.

Span girder B

- Next, select the thirty-two lines which will have **Span Girder B** properties assigned to them.



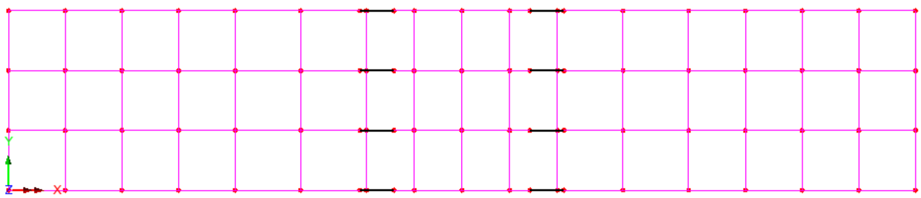
Geometry
Group >
New Group

This adds a New Group entry for the features selected.

- Enter the group name as **Span Girder B** and click **OK** to finish defining the group.

Pier girder uncracked region

- Next, select the sixteen lines which will have **Pier Girder Uncracked** properties assigned to them. These are the lines between the splice and the point where the cracked section properties start.



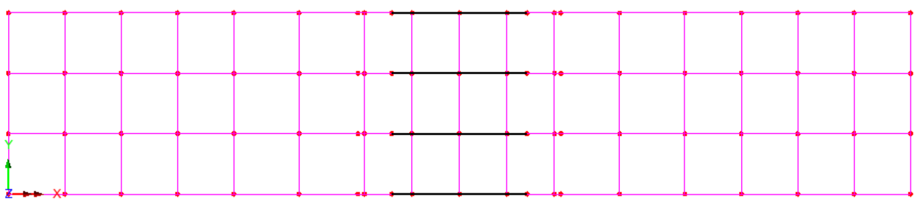
Geometry
Group >
New Group

This adds a New Group entry for the features selected.

- Enter the group name as **Pier Girder Uncracked** and click **OK** to finish defining the group.

Pier girder cracked region

- Select the sixteen lines which will have the **Pier Girder Cracked** properties assigned to them.



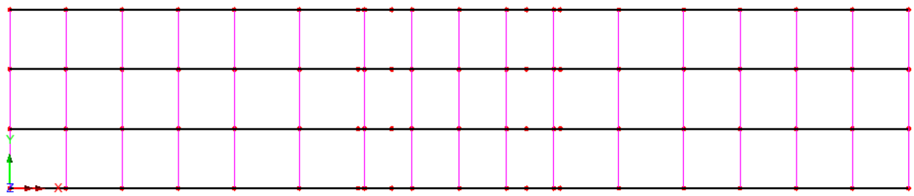
Geometry
Group >
New Group

This adds a New Group entry for the features selected.

- Enter the group name as **Pier Girder Cracked** and click **OK** to finish defining the group.

Main girders

- Select all the longitudinal lines.



Geometry
Group >
New Group

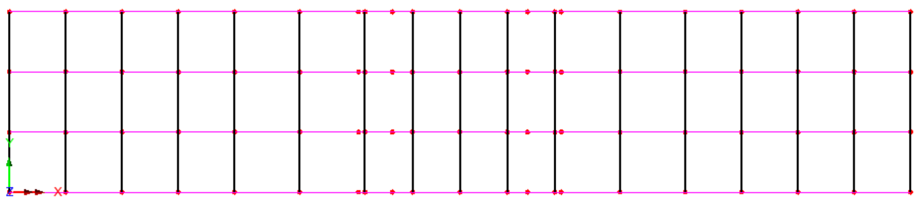
This adds a New Group entry for the features selected.

- Enter the group name as **Main Girders** and click **OK** to finish defining the group.

Transverse slab

- Select all the transverse lines

Tip. Select the 'Select Y axis' cursor and box-select the whole model.



Geometry
Group >
New Group

This adds a New Group entry for the features selected.

- Enter the group name as **Transverse Slab** and click **OK** to finish defining the group.
- Ensure the 'Select Any' cursor is back in use.

Construction phases

Note. To carry out the design of the bridge the analysis model will need to have the relevant construction phases **(Phase 1, Phase 2a, Phase 2b** and **Phase 3)** simulated within the analysis model. Normally the order that these would be listed in the Analyses treeview for the model would mimic the construction process, so **Phase 1** would come before **Phase 2a.** However it is intended to use the **Traffic Load Optimisation** facility with LUSAS to generate the Traffic load patterns for **Phase 3**. This will require influence analysis to be undertaken and currently when running an influence analysis the properties of the base analysis are always considered for the influence analysis. Therefore the base analysis will actually be set with the short term properties that are used for **Phase 3**. So the actual order that the Phases will be dealt with in this example will be **Phase 3** first, followed by **Phase 2b, Phase 2a** and finally **Phase 1.**

Defining and Assigning Mesh Attributes

Attributes
Mesh >
Line…

- The grillage Lines are to be modelled with **Grillage, Linear** elements (GRIL elements) with **1** mesh division.
- Enter the attribute name as **Grillage** and click **OK**.
- Select all Lines in the model and drag and drop the **Grillage** mesh from the Treeview onto the selected features. Click **OK** to assign.

Defining Geometric Properties

The section properties for the longitudinal members in the grillage have been imported in LUSAS earlier in this example. However the transverse section properties for the slab sections now need to be created.

Only a single transverse section will be defined for this example, and this will be used in all locations. This is a simplification as the grillage bays are of different widths and the transverse section should be based on size of grillage bays.

Attributes
Geometric >
Bridge Deck (Grillage) > Slab

- Ensure that the slab type is set to **Solid**
- Enter the Breadth (b) = **3.5**, Thickness (t) = **0.25**
- Enter the attribute name as **Transverse Slab** and click **OK**.

Assigning Geometric Properties

As an alternative to selecting features by dragging a box around them, the previously named Groups will be used.

Select the fleshing on/off button to turn-off geometric property visualisation.

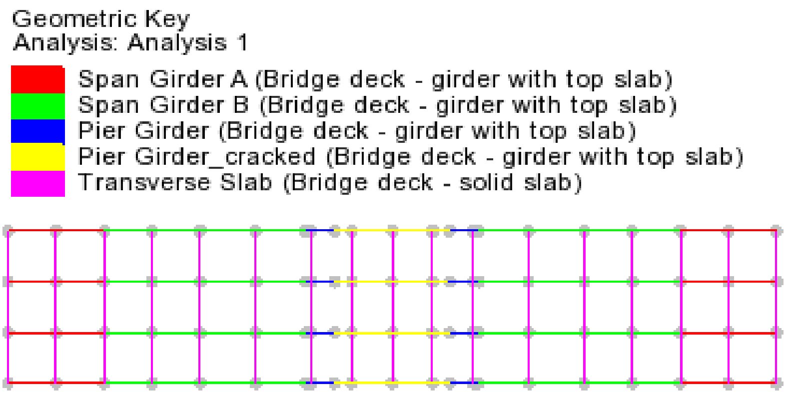
- In the Treeview, click the right-hand mouse button on the group name **Span Girder A** and select the **Select Members** option from the menu. If you already have some features selected, click **Yes** to deselect them first.
- From the Treeview, drag and drop the geometric attribute **Span Girder A** onto the selected Lines.

Repeat this operation for the following assignments:

• In the Treeview, select the members of each group, clicking **Yes** to deselect any existing selection.	• From the Treeview, drag and drop each named geometric attribute onto the selected lines
• Span Girder B	• Span Girder B
• Pier Girder Uncracked	• Pier Girder
• Pier Girder Cracked	• Pier Girder_cracked
• **Transverse Slab**	• Transverse slab

Checking Geometric Assignments

- Double-click on the **Geometry** layer in the Treeview and select **Assignment** from the Colour by drop down list. Click the **Set** button and select **Geometric** from the Attribute Type drop down list.
- With the **Generate key** option selected click the **OK** button to return to the Geometry properties dialog and **OK** again to display the geometry coloured by geometric assignment with the key annotated.



- To remove the 'Colour by Attribute' assignment double click on **Geometry** in the Treeview, select **Own Colour** from the Colour by drop down list and click **OK** to update the display.

Defining and Assigning Material Properties

The materials for the grillage have already been imported into LUSAS earlier in this example. These properties now need to be assigned to selected members of the model as follows:

• In the Treeview, select the members of each group, clicking **Yes** to deselect any existing selection.	• From the Treeview, drag and drop each named material attribute onto the selected lines	• Assign to Analysis	• Assign to Loadcase
• Main Girders	• BDM_Phase3	• **Analysis 1**	• **Loadcase 1**
• Transverse **Slab**	• BDM_Phase3	• **Analysis 1**	• **Loadcase 1**

Defining and Assigning Supports

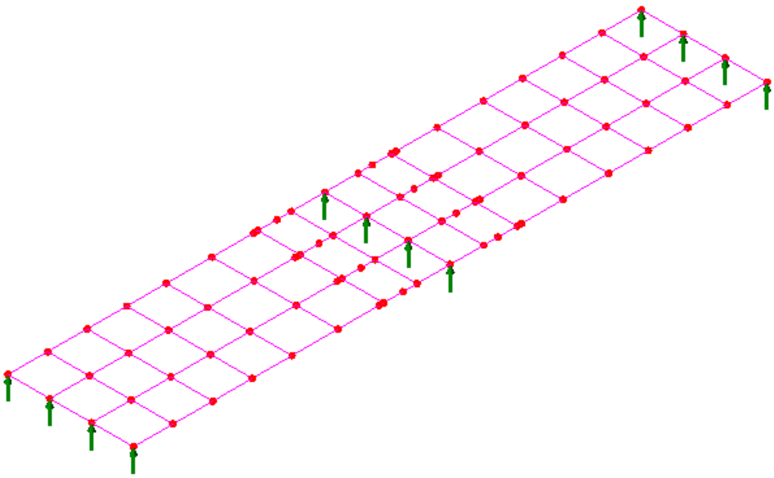
Attributes
Support…

Define supports that set up the correct articulation for the structure. In this example the bridge will have one pinned support, one guided support and the remainder of the supports will be vertical.

- Ensure only Translation in **Z** is **Fixed**
- Enter the attribute name as **Vertical (Z)** and click the **OK** button to add the attribute to the Treeview.

Assign the supports to the model

- Box-select the Points that represent each of bearing locations and drag and drop the support attribute **Vertical (Z)** from the Treeview onto the selection. With the **Assign to Points** option selected click **OK** to assign the support attribute to **Analysis 1** for all Analysis loadcases.



Defining and Assigning Loading

In the first analysis, variable actions will be considered. Thermal effects are variable actions that will be applied to the analysis.

Note. In this example it is going to be assumed that the simplified approach to the thermal loading can be used. However it should be noted that in BS EN 1991-1-5, 6.1.2(2) it is stated that this simplified approach should only be used if it is agreed for the individual project with the relevant authority. Otherwise an approach using a nonlinear thermal profile should be adopted.

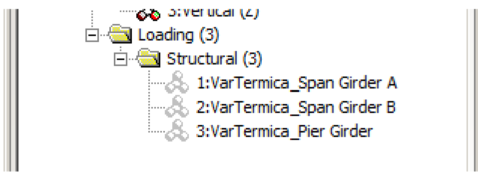
Earlier in the example the Thermal effects were exported from the Composite Deck Designer (**PontiEC4**) into a file called **"LUSAS_Thermal.vbs".**

File
Script >
Run Script…

Locate the "**LUSAS_Thermal.vbs"** and open in LUSAS.

- Select the file **LUSAS_Thermal.vbs** and press OK to open in LUSAS.

This adds some thermal load attributes to the Attributes treeview as shown:



- If required, right-click on each in turn and rename the label **VarTermica** to be **Thermal.**

Assigning thermal loading

- In the Groups Treeview, click the right-hand mouse button on the group name **Span Girder A** and select the **Select Members** option from the menu. If you already have some features selected, click **Yes** to deselect them first.

- From the Attributes Treeview, drag and drop the load attribute **Thermal_Span Girder A** onto the selected Lines. Make sure that the analysis option is set to **Analysis 1** and the loadcase option is set **Loadcase 1** to assign the properties into the first analysis.

- Repeat these operations to make the following assignments:

• In the Treeview, Select Members of each group, clicking **Yes** to deselect any existing selection.	• From the Treeview, drag and drop each named loading attribute onto the selected lines	• Assign to Analysis	• Assign to Loadcase
• **Span Girder B**	• **Thermal_Span Girder B**	• **Analysis 1**	• **Loadcase 1**
• **Pier Girder Uncracked**	• **Thermal_Pier Girder**	• **Analysis 1**	• **Loadcase 1**
• **Pier Girder Cracked**	• **Thermal_Pier Girder**	• **Analysis 1**	• **Loadcase 1**

Rename analysis 1

- In the Analyses treeview, right-click on **Analysis 1** and rename as **Analysis 1 Phase 3 Composite Variable Actions**

- In the treeview, right-click on **Loadcase 1** in Analysis 1 and rename as **Thermal heat**.

Assigning Thermal Loading for Cooling Case

- In the Groups Treeview, click the right-hand mouse button on the group name **Span Girder A** and select the **Select Members** option from the menu. If you already have some features selected, click **Yes** to deselect them first.

- From the Attributes Treeview, drag and drop the load attribute **Thermal_Span Girder A** onto the selected Lines. Make sure that the analysis option is set to

Analysis 1. Then, using the loadcase select, select **New** from the drop-down list, and enter the name **Thermal cool.** Click **OK**. Then click the **More** button and enter the load factor as **-1**.

- Repeat these operations to make the following assignments:

• In the Treeview, Select Members of each group, clicking **Yes** to deselect any existing selection.	• From the Treeview, drag and drop each named loading attribute onto the selected lines	• Assign to Analysis	• Assign to loadcase • With a load factor of
• **Span Girder B**	• **Thermal_Span Girder B**	• **Analysis 1**	• **Thermal cool** • **-1**
• **Pier Girder Uncracked**	• **Thermal_Pier Girder**	• **Analysis 1**	• **Thermal cool** • **-1**
• **Pier Girder Cracked**	• **Thermal_Pier Girder**	• **Analysis 1**	• **Thermal cool** • **-1**

Tip. Select both Pier Girder Uncracked and Pier Girder Cracked groups in the treeview to select the contents of both groups

Save the model

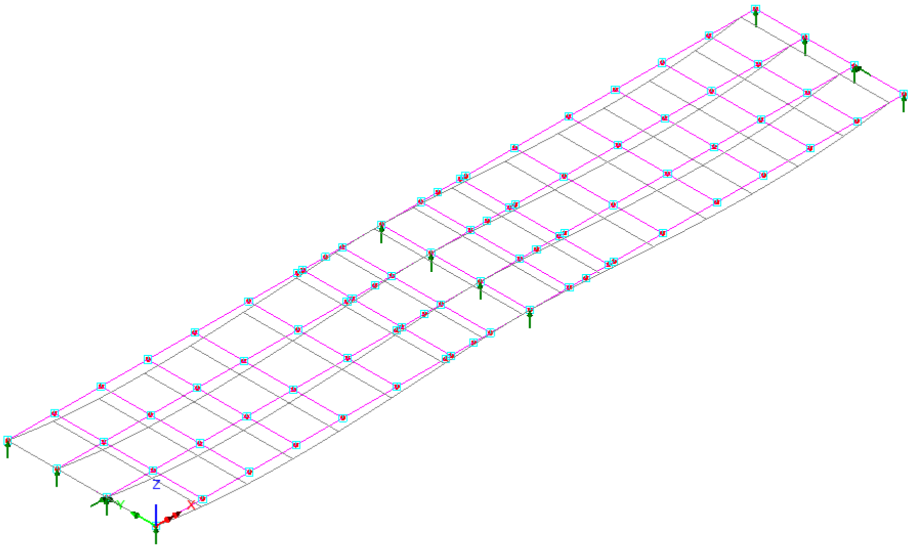
File
Save

Save the model file.

Tip. Before creating the other analyses or further loadcases it is good practice to check that Analysis 1 will solve without errors.

Check the model will solve

Click the **Solve now** button and press **OK** to run the analysis using LUSAS Solver and load the results file. The deformed mesh plot for Analysis 1, loadcase Thermal cool is shown below.



Once Analysis 1 has been checked to ensure that it solves without errors, the additional analyses and phases required for the design process can be added.

Defining Analysis 2 Settlement and Shrinkage Loads

Analyses
Structural Analysis…

- Ensure that the option **Inherit from the base analysis (All)** is checked (ticked).
- Enter the analysis name as Analysis 2 Phase 2b Composite Settlement/Shrinkage, then press OK.

This will create a second analysis that has the same engineering properties (geometric sections, materials properties and supports) as the base analysis, **Analysis 1 Phase 3 Composite Variable Actions.**

- In the Analyses treeview right-click on **Loadcase 2** in Analysis 2 and rename as **Settlement**.

Assigning Geometric and Material Properties in Analysis 2

Currently Analysis 2 has inherited the section properties from Analysis 1. When using grillage sections this is a simple case of assigning a grillage material to the analysis and the section properties will be recalculated using the material assignment.

- Select all the lines in the model and from the Treeview, drag and drop the grillage section material **BDM_Phase2b** onto the model. Ensure Analysis 2 is selected in the Attribute Assignment dialog and click OK.

Phase 2b will use the age adjusted "long term" concrete material.

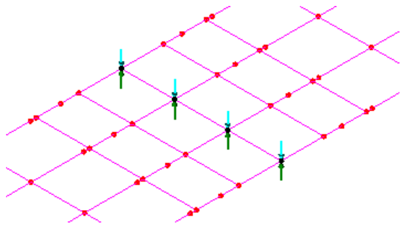
Defining and Assigning Loading in Analysis 2

In the second analysis the settlement and shrinkage loads will be applied to the analysis model.

A settlement of 10mm will be considered at the internal pier.

Attributes
Loading…

- Select the **Prescribed Displacement** option and click **Next.**
- Set the translation in the **Z** direction to **Fixed**, the displacement as **Total, -0.01**m.
- Enter an attribute name of **Settlement 10mm** and click **Finish**.
- Select the four points at the internal pier location that have supports assigned to them.
- From the Treeview, drag and drop the point loading attribute **Settlement 10mm** onto the selected Points. Make sure that the load is applied into **Analysis 2 Phase 2b Composite Settlement/Shrinkage** and the loadcase **Settlement**.

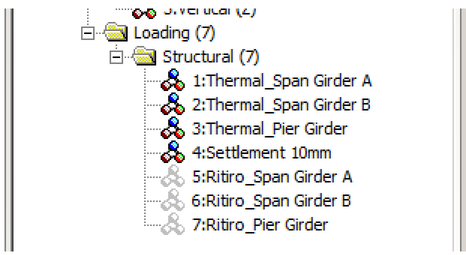


Next, the effects of shrinkage can be applied to the analysis. Earlier in the example the shrinkage effects were exported from the Composite Deck Designer (**PontiEC4**) into a file called **"LUSAS_Shrinkage.vbs".**

File
Script >
Run Script…

- Locate the "**LUSAS_Shrinkage.vbs"** and open in LUSAS.
- Select the file **LUSAS_Shrinkage.vbs** and press OK to open in LUSAS.

This adds some shrinkage load attributes to the Attributes Treeview as shown:



These attributes are to be renamed.

- Right-click on each in turn and rename the label **Ritiro** with the word **Shrinkage.**

- In the Treeview, click the right-hand mouse button on the group name **Span Girder A** and select the **Select Members** option from the menu. If you already have some features selected, click **Yes** to deselect them first.

- Drag and drop the loading attribute **Shrinkage_Span Girder A** from the Treeview onto the selected Lines. Make sure that the analysis option is set to **Analysis 2 Phase 2b Composite Settlement/Shrinkage** to assign the properties into the second analysis. Using the loadcase select, select **New** from the drop-down list, and enter the name **Shrinkage**. Click **OK**.

- Repeat these operations to make the following assignments:

• In the Treeview, Select Members of each group, clicking **Yes** to deselect any existing selection.	• From the Treeview, drag and drop each named loading attribute onto the selected lines	• Assign to Analysis	• Assign to Loadcase
• **Span Girder B**	• **Shrinkage_Span Girder B**	• **Analysis 2 Phase 2b Composite Settlement/ Shrinkage**	• **Shrinkage**
• **Pier Girder Uncracked**	• **Shrinkage_Pier Girder**	• **Analysis 2 Phase 2b Composite Settlement/ Shrinkage**	• **Shrinkage**
• **Pier Girder Cracked**	• **Shrinkage_Pier Girder**	• **Analysis 2 Phase 2b Composite Settlement/ Shrinkage**	• **Shrinkage**

Defining Analysis 3 Permanent Loads

Analyses
Structural Analysis…

- Ensure that the option **Inherit from the base analysis (All)** is checked (ticked).

- Enter the analysis name as Analysis 3 Phase 2a Composite Permanent, then press OK.

- In the Analyses treeview right-click on **Loadcase 2** in Analysis 3 and rename as **Surfacing**.

Assigning Geometric and Material Properties in Analysis 3

- Select all the lines in the model and from the Treeview, drag and drop the grillage section material **BDM_Phase2a** onto the model. Ensure Analysis 3 is selected in the Attribute Assignment dialog and click OK.

Phase 2a will use the age adjusted "long term" concrete material.

Defining and Assigning Loading in Analysis 3

In the third analysis the permanent loads due to surfacing will be applied on the main girders.

Attributes
Loading…

- Select the **Global distributed** option and click **Next.**
- Set the type to **Per unit length**, the Z direction as **-8700** N/m.
- Enter an attribute name of **Weight of Surfacing** and click **Finish**.
- In the Treeview, click the right-hand mouse button on the group name **Main Girders** and select the **Select Members** option from the menu. If you already have some features selected, click **Yes** to deselect them first.
- From the Treeview, drag and drop the loading attribute **Weight of Surfacing** onto the selected Lines. Make sure that the load is applied into **Analysis 3 Phase 2a Composite Permanent** and the loadcase **Surfacing**.

Defining Analysis 4 Permanent Loads

Analyses
Structural Analysis…

- Ensure that the option **Inherit from the base analysis (All)** is checked (ticked).
- Enter the analysis name as Analysis 4 Phase 1 Non-Composite Self Weight, then press OK.
- In the Analyses treeview, right-click on **Loadcase 1** in Analysis 4 and rename as **Steel Self Weight**.

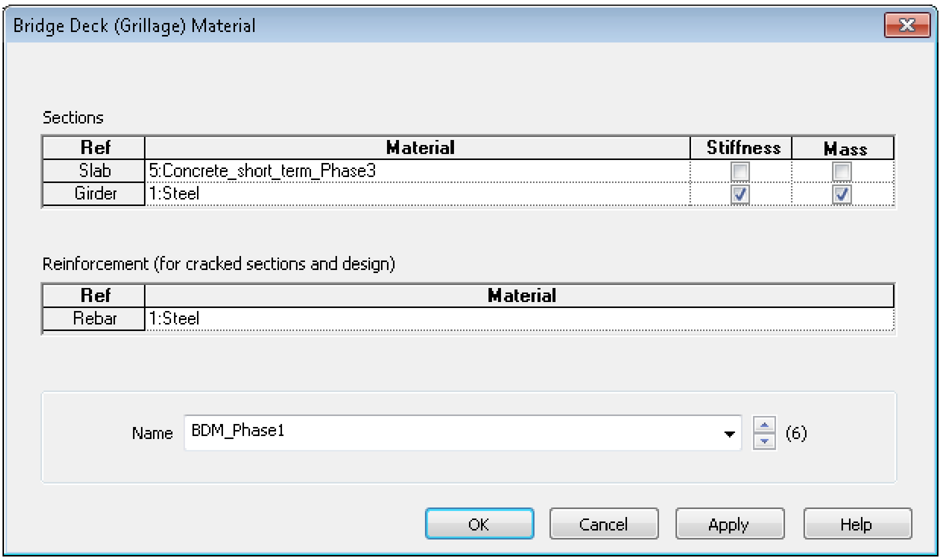
Assigning Geometric and Material Properties in Analysis 4

- Select all the lines in the model and from the Treeview, drag and drop the grillage section material **BDM_Phase1** onto the model. Ensure Analysis 4 is selected in the Attribute Assignment dialog and click OK.

Phase 1 will use only the stiffness of the steel to represent the phase before the concrete deck has been cast.

The grillage material provides options for inclusion of mass and stiffness. Since we will apply the concrete weight separately, so it maybe subsequently factored separately we need to remove the mass contribution from the concrete deck in the phase 1 material.

- Double click the material attribute **BDM_Phase1** in the Treeview, <u>deselect</u> the option for **Mass** in the slab and click **OK**.



Defining and Assigning Deactivation dataset in Analysis 4

Since Analysis 4 is only going to consider the steel girder sections on their own the Transverse Slab will be deactivated.

Attributes
Activate and Deactivate…

- Select the **Deactivate** option and click **Next**
- Select the **Percentage to redistribute** option and leave the value as **100%**. Enter the attribute name as **Deactivate**, then press **Finish**

- In the Treeview, click the right-hand mouse button on the group name **Transverse Slab** and select the **Select Members** option from the menu. If you already have some features selected, click **Yes** to deselect them first.
- From the Treeview, drag and drop the deactivation attribute **Deactivate** onto the selected Lines. Make sure that the analysis option is set to **Analysis 4 Phase 1 Non-Composite Self Weight** and the loadcase **Steel Self Weight**.

Defining and Assigning Loading in Analysis 4

In the fourth analysis the dead load of the steel beams and the weight of the concrete acting on the main girders will be considered.

Attributes
Loading…

- Select the **Body Force** option and click **Next.**
- Enter an acceleration of **-9.81** in the Z direction.
- Enter an attribute name of **Gravity** and click **Finish**.

Attributes
Loading…

- Select the **Global distributed** option and click **Next.**
- Set the type to **Per unit length**, the Z direction as **-22685** N/m.
- Enter an attribute name of **Weight of Slab** and click **Finish**.
- In the Treeview, click the right-hand mouse button on the group name **Main Girders** and select the **Select Members** option from the menu. If you already have some features selected, click **Yes** to deselect them first.
- From the Treeview, drag and drop the loading attribute **Gravity** onto the selected Lines. Make sure that the analysis option is set to **Analysis 4 Phase 1 Non-Composite Self Weight** and loadcase **Steel Self Weight**.
- With the members of the group **Main Girders** still selected, drag and drop the loading attribute **Weight of Slab** from the Treeview onto the selected Lines. Make sure that the analysis option is set to **Analysis 4 Phase 1 Non-Composite Self Weight** and using the loadcase selection, select **New** from the drop-down list, and enter the name **Concrete Self Weight**. Click **OK**.

Additional Traffic load patterns

Now that the basic analysis models have been defined, the additional Traffic load Patterns can be created.

In order to use the vehicle load optimisation facility the locations (points or nodes) at which the optimised load patterns are to be created need to be selected as well as the LUSAS component of interest.

To do this, influence attributes are defined and assigned to the model. LUSAS then calculates the influence surfaces for any specified position in the structure. Two methods are available: the Reciprocal Method and the Direct Method. Since in this example only two locations are going to be considered the Reciprocal method will be used. See the Online Help for details of these two methods.

Note. Only two locations are going to be considered as this example is only considering two locations for detailed design. In reality many locations would need to be considered to cover the full design of the bridge, but for clarity this simplification will be accepted. The design checks will be carried out along girder 2, at the middle of span 1 and also over the pier support.

Defining Influence Attributes

The first influence attribute will be used to investigate the bending moment at near the middle of span 1 on girder 2.

Attributes
Influence >
Reciprocal Theoreml…

- Select a **Moment** influence type for an **About transverse** influence direction for a **Negative** displacement direction. Enter the influence attribute name as **Sagging moment** and click **OK**. The influence attribute will be added to the Treeview.

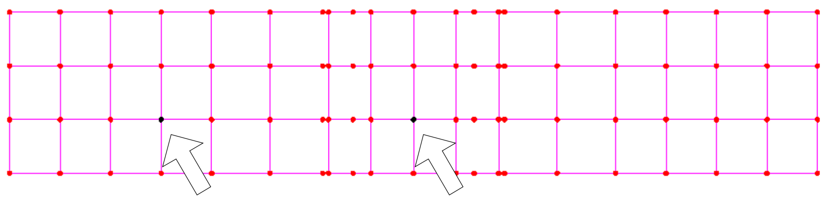
The second influence attribute will be used to investigate the bending moment over the pier on girder 2.

Attributes
Influence >
Reciprocal Theorem…

- Select a **Moment** influence type for an **About transverse** influence direction for a **Positive** displacement direction. Enter the influence attribute name as **Hogging moment** and click **OK**. The influence attribute will be added to the Treeview.

Assigning Influence Attributes

Influence attributes will be assigned at the middle of span 1 on girder 2, and at the pier on girder 2. See image that follows.

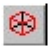


- Select the Point (10.5, 3.7) as shown on girder 2.
- From the Treeview, drag and drop the influence attributes **Sagging moment** onto the selected Point. This will automatically be assigned into a new Reciprocal Influence Analysis.
- Select the Point 28.0, 3.7) at the pier on girder 2.

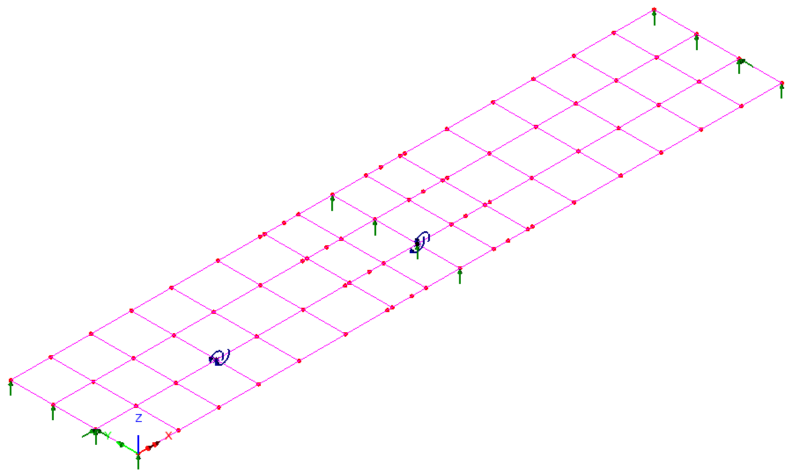
- From the  Treeview, drag and drop the influence attributes **Hogging moment** onto the selected Point. This will automatically be assigned into a new Reciprocal Influence Analysis.

Visualising the Defined Influence Points

Influence attribute assignments are visualised as they are assigned to the model. To check that the influence attribute orientations are correct (meaning that the correct influence directions have been defined) an isometric view of the model should be used.

 Select the isometric button.

- Ensure that the influence visualisations are as shown



Using the Vehicle Load Optimisation facility to calculate the worst case traffic loading patterns

The vehicle load optimiser automates the creation of load datasets in accordance with the chosen loading code for the locations and effects specified.

Before invoking the Vehicle Load Optimisation Wizard the kerb lines need to be defined.

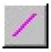
 Select the Home button to return the model to the default view.

Defining Kerb Lines

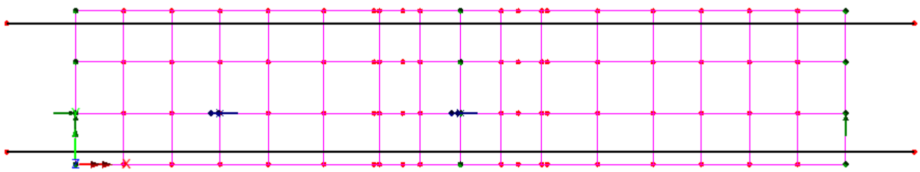
Geometry >
Line >
Coordinates…

 Enter coordinates of (**-5, 0.9**), and **(61.0, 0.9)**, to define a line representing one of the kerbs, and click OK.

Geometry >
Line >
Coordinates…

 Enter coordinates of (**-5, 10.2**), and **(61.0, 10.2)**, to define a line representing the other kerb, and click OK.

- Select the first and second kerb line. (Use the **Shift** key to add to the initial selection)



Vehicle Load Optimisation

Now that influence attributes have been defined and assigned, the VLO facility can be used to define optimised traffic loading for the bridge, based on a chosen code of practice. For this example design code EN1991-2 UK will be used. In this code, normal traffic is represented by Load Model 1 (LM1).

For the road carried by this bridge, the highway authority has also specified that the abnormal traffic be represented by the special vehicle SV100, as defined in the UK National Annex.

It will be assumed that the traffic loading will be the leading variable action and therefore only characteristic and frequent values will be considered within the VLO facility.

- Ensure that the lines representing the kerbs are still selected

Bridge
Vehicle Load Optimisation

- Select **United Kingdom** from the Country drop down list and choose **EN1991-2 UK 2009** as the Design code.
- Press the **Optional Code settings** button and ensure that only **Characteristic** and **Frequent** representative values required are checked (ticked). Ensure that **Group 1a –LM1, Group 4 –LM4 and Group 5 –LM3** load groups are to be included. For the vehicle selector, press the button and deselect the SV80 option and select the **SV100** vehicle only. Press **OK** to exit the dialog.
- Press the **Define carriageways** button. Ensure **Kerbs from selection** is selected and press **Apply** to exit the dialog.
- Press the Set influence surfaces button, and on the next dialog select Include all influence surfaces.

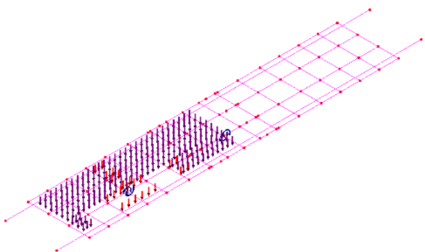
- Ensure that the **Positive** checkboxes are selected for both the Sagging and Hogging moment. This means that the VLO analysis will produce loading patterns for positive regions of the influence shape.

Note. Clicking in the header cell of a column will select the whole column so that checking (ticking) one entry will check (tick) all entries in that column.

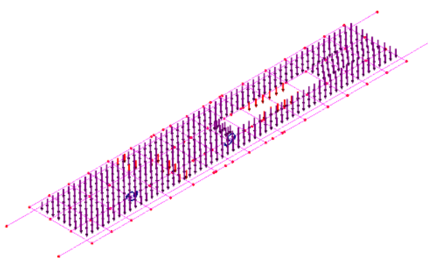
- Press **OK** to return to the main VLO dialog.
- Ensure the option to **View onerous effect table** is not selected.
- Ensure the option to Create loading patterns for All chosen influences is selected.
- To specify a non-default analysis name choose **New** from the drop-down list, and enter the analysis name to be **VLO Analysis 5**
- Change the VLO run Name to be **VLO Eurocode**
- Press **OK** which will run generate the optimised loading patterns for the influence points of interest.

After a short time, and at the bottom of the Treeview, a new entry **VLO Analysis 5** will be created. This contains the loading from VLO run, as denoted by this icon, , which includes Characteristic and Frequent load combinations for the locations requested.

- By setting a loadcase active and turning on the loading, the traffic loading patterns for each influence result can be seen.



Loading pattern
for influence result in span 1 for loadcase
Sagging moment - Positive - Characteristic



Loading pattern
for influence result at pier for loadcase
Hogging moment – Positive - Characteristic

File
Save

Save the model file.

Solve the model

Click the **Solve now** button and press **OK** to run all the analyses using LUSAS Solver and load the results file.

Combinations and envelopes

Now that all the Analyses and their respective load cases have been created, combinations and envelopes for each Phase and for each Limit State the can be defined.

Four cases will be defined, **ULS Fundamental, SLS Characteristic, SLS Frequent** and **Fatigue**.

Before the limit state combinations can be created some initial envelopes need to be created.

Analyses
Envelope...

- Include the **Thermal heat** and the **Thermal cool** loadcases from Analysis 1.
- Click the >> button to add these loadcases to the included list.
- Change the envelope name to **Thermal Envelope**
  - Click the **OK** button to complete the definition and create an envelope in the Treeview.

Repeat the above steps for the following envelopes

Note. The loadcase names used below will depend on exact point selected in model and how the model was constructed. Therefore the loadcase names might have different point numbers in the name string.

Analyses
Envelope...

- Include the **Sagging moment - Point 76 - Positive – Characteristic** and the **Hogging moment - Point 86 - Positive – Characteristic loadcases** from Analysis 5, change the name to **Characteristic Envelope** and click **OK**.

Analyses
Envelope...

- Include the **Sagging moment - Point 76 - Positive – Frequent** and the **Hogging moment - Point 86 - Positive – Frequent** loadcases from Analysis 5, change the name to **Frequent Envelope** and click **OK**.

Limit state Combinations

Before creating the combinations the way that the factors are displayed will be changed.

- In the Treeview double-click on the Combinations and envelope options and ensure that the option Display beneficial/adverse factors for smart combinations is selected (checked).

Note. If the last action is not undertaken the smart combination will show factors as permanent and variable. **Permanent load factor** is always applied. A **Variable load factor** is only applied if the effect is adverse. The option that is set changes the dialog Permanent/Variable to Beneficial/Adverse.

The following table shows the analyses and the loadcases that they contain – the loadcases that will be combined to form the required limit state combinations.

Analysis	Construction Phase	Loadcase
Analysis 4	Phase 1	Steel Self Weight Concrete Self Weight
Analysis 3	Phase 2a	Surfacing
Analysis 2	Phase 2b	Settlement Shrinkage
Analysis 1	Phase 3	Thermal heat Thermal cool
Analysis 5	VLO using properties from Phase 3	Sagging moment - Point 76 - Positive – Characteristic Sagging moment - Point 76 - Positive – Frequent Hogging moment - Point 86 - Positive – Characteristic Hogging moment - Point 86 - Positive – Frequent

Note. In this example the number of loadcases included into each construction phase has been kept to a minimum to ensure the example is as concise as possible but still covers the basics of what is required. It is recognised that further actions could be included in each of the construction phases, for example **Analysis 5** could include the variable action **Wind**. However, it is intended that this example shows the process of using the Composite Deck Designer (**PontiEC4**) with **LUSAS** from start to finish. Once this is understood additional actions can easily be fitted into the process.

The following table shows the envelopes that will also be combined with the basic loadcases to form the required limit state combinations.

Envelope	Envelope loadsets
Thermal Envelope	Thermal Envelope (Max) Thermal Envelope (Min)
Characteristic Envelope	Characteristic Envelope (Max) Characteristic Envelope (Min)
Frequent Envelope	Frequent Envelope (Min)

	Frequent Envelope (Max)

ULS Combinations

The factors in the combinations will be based on the traffic loading being the leading variable action.

ULS_Fundamental_Phase1

Analyses
Smart Combination...

This creates a smart load combination in the Treeview.

- Include the Steel and Concrete **Self Weight** loadcases from Analysis 4 in the smart combination.
- Click the >> button to add these loadcases to the included list.
- Set the permanent and variable factors as shown in bold in the table that follows:

Name	Beneficial factor	Adverse factor
Steel Self Weight	**0.95**	**1.2**
Concrete Self Weight	**0.95**	**1.35**

Values are taken from UK NA to EN1990: Table NA.A2.4(B) Design Values For Actions. Steel Self weight $\gamma_{G,sup}$ = 1.20 and $\gamma_{G,inf}$ = 0.95 and Concrete Self weight $\gamma_{G,sup}$ = 1.35 and $\gamma_{G,inf}$ = 0.95

- Change the smart combination name to **ULS_Fundamental_Phase1** and Click **OK.**
- Repeat the above procedure for the following smart combinations:

ULS_Fundamental_Phase2a

Name	Beneficial factor	Adverse factor
Surfacing	**0.95**	**1.2**

Values are taken from UK NA to EN1990: Table NA.A2.4(B) Design Values For Actions. Road Surfacing $\gamma_{G,sup}$ = 1.20 and $\gamma_{G,inf}$ = 0.95

- Enter smart combination name: **ULS_Fundamental_Phase2a**

ULS_Fundamental_Phase2b

Name	Beneficial factor	Adverse factor
Settlement	**0.0**	**1.2**
Shrinkage	**0.0**	**1.0**

Values are taken from UK NA to EN1990: Table NA.A2.4(B) Design Values For Actions. Settlement $\gamma_{G,sup}$ = 1.20 and $\gamma_{G,inf}$ = 0.0. The Partial factor for shrinkage γ_{Shr} is set to unity for both ULS and SLS by EN 1992-1-1, Clause 2.4.2.1.

No clear guidance is given on factors for shrinkage but in this example it will be considered $\gamma_{G,sup}$ = 1.20 and $\gamma_{G,inf}$ = 0.0

- Enter smart combination name: **ULS_Fundamental_Phase2b**

ULS_Fundamental_Phase3a

Name	Beneficial factor	Adverse factor
Thermal Envelope (Max)	**0**	**1.55*0.6**
Thermal Envelope (Min)	**0**	**1.55*0.6**

*Values are taken from UK NA to EN1990: Table NA.A2.4(B) Design Values For Actions. Thermal $\gamma_{G,sup}$ = 1.55 and $\gamma_{G,inf}$ = 0.0 and Table NA.A2.1 – Recommended values of ψ factors for road bridges ψ_0=0.6. This can be entered and will be evaluated by Modeller as 1.55*0.6*

- Enter smart combination name: **ULS_Fundamental_Phase3a**

ULS_Fundamental_Phase3b

Name	Beneficial factor	Adverse factor
Characteristic Envelope (Max)	**0**	**1.35**
Characteristic Envelope (Min)	**0**	**1.35**

Values are taken from UK NA to EN1990: Table NA.A2.4(B) Design Values For Actions. Road Traffic $\gamma_{G,sup}$ = 1.35 and $\gamma_{G,inf}$ = 0.0. Note the Vehicle Load Optimiser has already included the appropriate ψ values from Table NA.A2.1.

- Enter smart combination name: **ULS_Fundamental_Phase3b**

SLS Combinations

SLS_ Characteristic_Phase1

Name	Beneficial factor	Adverse factor
Steel Self Weight	**1.0**	**1.0**
Concrete Self Weight	**1.0**	**1.0**

Values are taken from EN 1990 Clause: 6.5.3 (2)

- Enter smart combination name: **SLS_Characteristic_Phase1**

SLS_ Characteristic_Phase2a

Name	Beneficial factor	Adverse factor
Surfacing	**1.0**	**1.0**

Values are taken from EN 1990 Clause: 6.5.3 (2)

- Enter smart combination name: **SLS_ Characteristic_Phase2a**

SLS_ Characteristic_Phase2b

Name	Beneficial factor	Adverse factor
Settlement	**0.0**	**1.0**
Shrinkage	**0.0**	**1.0**

Values are taken from EN 1990 Clause: 6.5.3 (2). The Partial factor for shrinkage γ_{Shr} is set to unity for both ULS and SLS by EN 1992-1-1, Clause 2.4.2.1.

- Enter smart combination name: **SLS_Characteristic_Phase2b**

SLS_ Characteristic_Phase3a

Name	Beneficial factor	Adverse factor
Thermal Envelope (Max)	**0**	**1.0 x 0.6**
Thermal Envelope (Min)	**0**	**1.0 x 0.6**

Values are taken from EN 1990 Clause: 6.5.3 (2) and Table NA.A2.1 – Recommended values of ψ factors for road bridges ψ_0=0.6.

- Enter smart combination name: **SLS_Characteristic_Phase3a**

SLS_Characteristic_Phase3b

Name	Beneficial factor	Adverse factor
Characteristic Envelope (Max)	**0**	**1.0**
Characteristic Envelope (Min)	**0**	**1.0**

Values are taken from EN 1990 Clause: 6.5.3 (2)

- Enter smart combination name: **SLS_Characteristic_Phase3b**

SLS_ Frequent_Phase1

Name	Beneficial factor	Adverse factor
Steel Self Weight	**1.0**	**1.0**
Concrete Self Weight	**1.0**	**1.0**

Values are taken from EN 1990 Clause: 6.5.3 (2)

- Enter smart combination name: **SLS_Frequent_Phase1**

SLS_ Frequent_Phase2a

Name	Beneficial factor	Adverse factor
Surfacing	**1.0**	**1.0**

Values are taken from EN 1990 Clause: 6.5.3 (2)

- Enter smart combination name: **SLS_ Frequent_Phase2a**

SLS_ Frequent_Phase2b

Name	Beneficial factor	Adverse factor
Settlement	**0.0**	**1.0**
Shrinkage	**0.0**	**1.0**

Values are taken from EN 1990 Clause: 6.5.3 (2). The Partial factor for shrinkage γ_{Shr} *is set to unity for both ULS and SLS by EN 1992-1-1, Clause 2.4.2.1.*

- Enter smart combination name: **SLS_Frequent_Phase2b**

SLS_ Frequent_Phase3a

Name	Beneficial factor	Adverse factor
Thermal Envelope (Max)	**0**	**1.0 x 0.5**
Thermal Envelope (Min)	**0**	**1.0 x 0.5**

Values are taken from EN 1990 Clause: 6.5.3 (2) and Table NA.A2.1 – Recommended values of ψ *factors for road bridges* ψ_2*=0.5.*

- Enter smart combination name: **SLS_Frequent_Phase3a**

SLS_Frequent_Phase3b

Name	Beneficial factor	Adverse factor
Frequent Envelope (Max)	**0**	**1.0 x 0.75**
Frequent Envelope (Min)	**0**	**1.0 x 0.75**

Values are taken from EN 1990 Clause: 6.5.3 (2) and Table NA.A2.1 – Recommended values of ψ *factors for road bridges* ψ_1*=0.75*

- Enter smart combination name: **SLS_Frequent_Phase3b**

Fatigue Combinations

The fatigue load model to use is LM3, and this should be moved along the actual lane.

Geometry
Line >
By Coords…

- Enter coordinates of (**0, 3.725, 2.0**), and **(64.4, 3.725, 2.0)**, to define a line representing the axis of the actual lane.

Now define the fatigue load model:

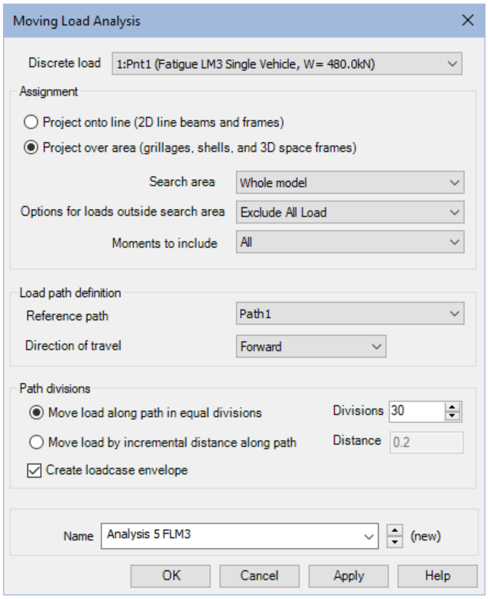
Bridge
Bridge loading >
Eurocode…

- Select **Fatigue Load Models** and from the drop-list select **LM3 Single Vehicle**, then click **OK**, and then **Close**.
- Select the line previously defined that will be used to define a reference path for the moving load.

Now create a new analysis – a moving load one:

Analyses
Moving load analysis…

- For Discrete load, select **Fatigue LM3 Single Vehicle…**
- Ensure the assignment is set to **Project over area.**
- For Search area select **Whole model**.
- Ensure **Move along path in equal number of divisions**, is selected and enter **Divisions** to be 30.
- Ensure **Create loadcase envelope** is selected.
- Enter the analysis name to be **Analysis 5 FLM3** and click **OK**.
- In the Analyses treeview rename the envelope that was automatically created to be **Fatigue_Phase3b**



Running the model

File
Save

Save the model file.

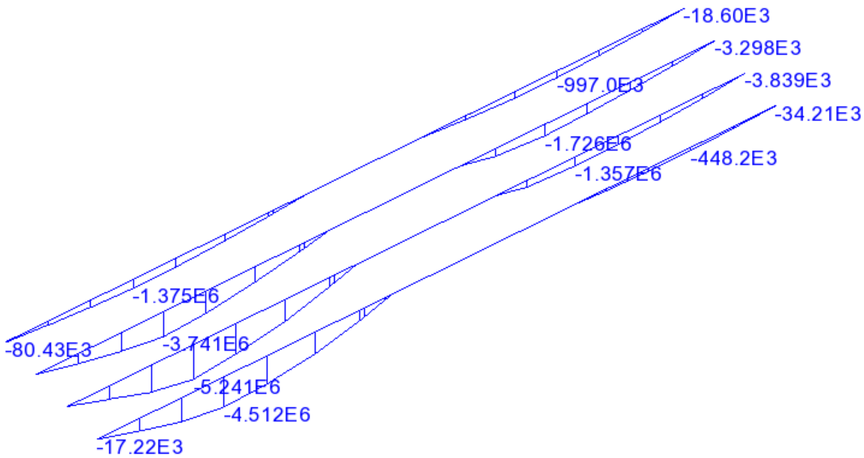
Solve the model

Click the **Solve now** button and press **OK** to run the analysis using LUSAS Solver and load the results file.

View initial results

Once the model has been solved the results can be viewed within LUSAS Modeller to create bending moment diagrams etc. To show how this is done:

- In the Analyses treeview, set active loadcase **ULS_fundamental_Phase3b (Min),** choose entity **Force/Moment** and component **My**
- Right-click in the view window and add the **Diagrams** layer, selecting entity **Force/Moment** and component **My**. Ensure **Show minimum only** is selected and click **OK**.
- In the Groups treeview, right click on **Main Girders** and select **Set As Only Visible**.



This worked example will not carry out any other processing or viewing of results within LUSAS Modeller, instead the results will be exported back to the Composite Deck Designer (PontiEC4) where the detailed design checks will be made.

Part 4: Exporting the force and moment data from LUSAS to the Composite Deck Designer (PontiEC4)

Associated File

This supplied file is only for use in case of problems in manually creating the appropriate data in the previous section of the example.

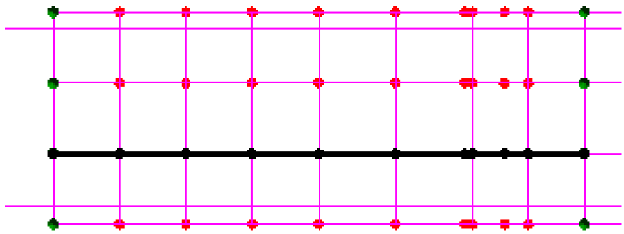
- ❑ **Force_and_Moments.xls** File created by the LUSAS Export Forces and Moments dialog containing data to import into PontiEC4.

Creation of design member

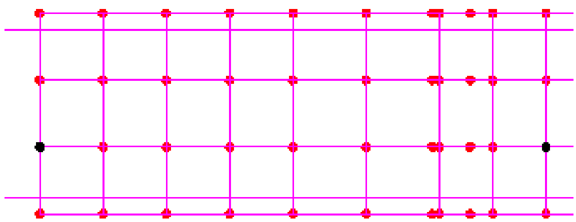
A design member is used to define locations that will be designed and at which forces and moments will be exported. The design member uses Beam/Shell slicing to export forces and moments and therefore can be used on all beam and shell models.

Whilst this is only a simple grillage model, the procedure is the same for more complex analytical models.

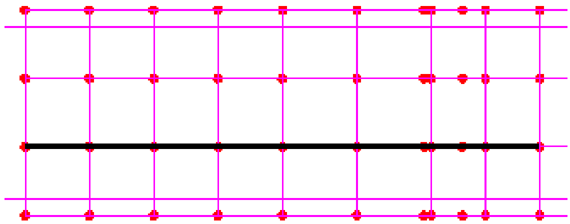
- Right click in the main view and deselect the diagrams layer.
- Right click in the main view and select **All visible**
- Select all of the lines that make up Span 1 Girder 2 and click the grpups button to create a new group, entre the name **Design Member**



- Select only the end points of the design member selected select the new line button to create a line which is the same length of the span

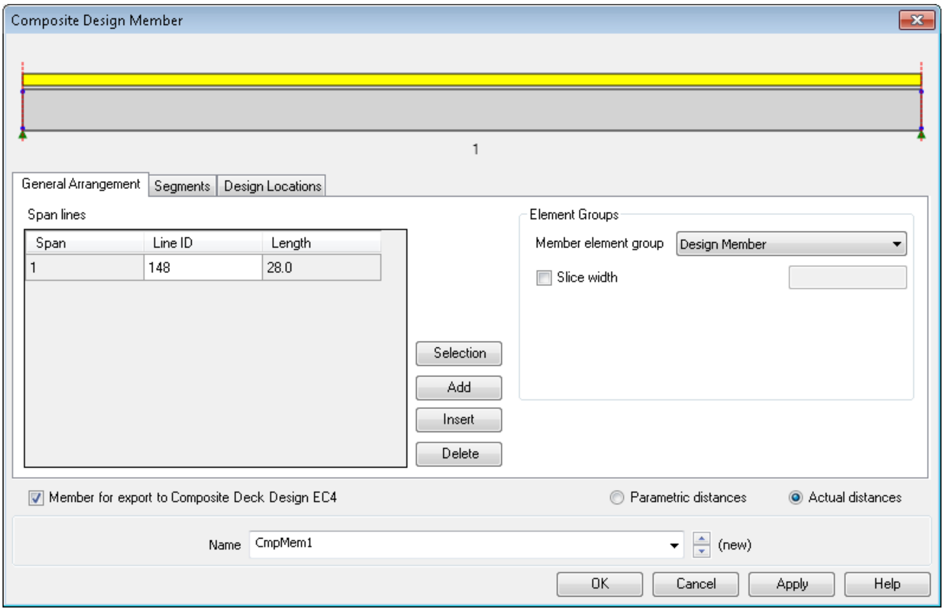


- Select the newly created line



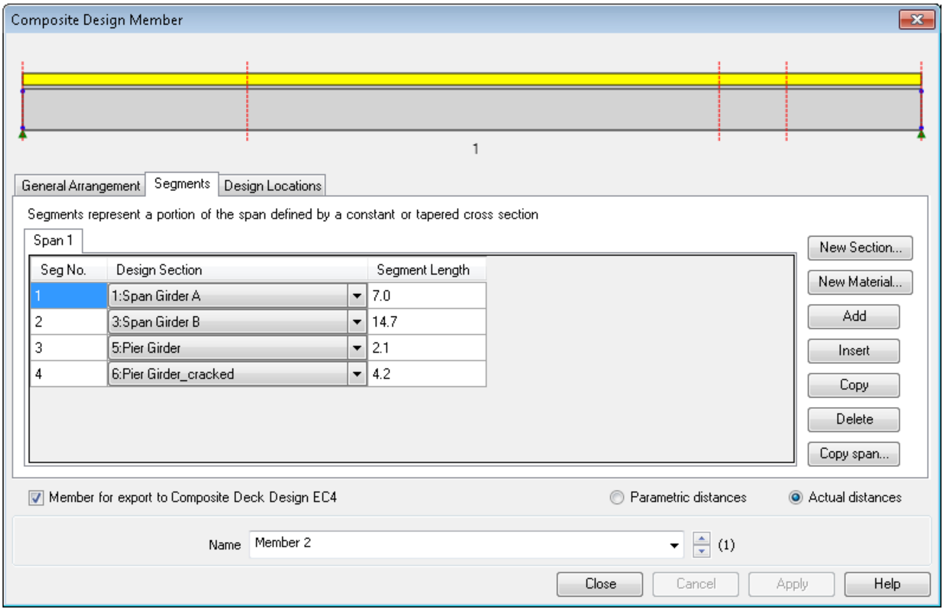
- This line will be used to identify locations at which forces and moments will be exported via a composite design member.

Utilities >
Bridge Design >
Composite Design Member…

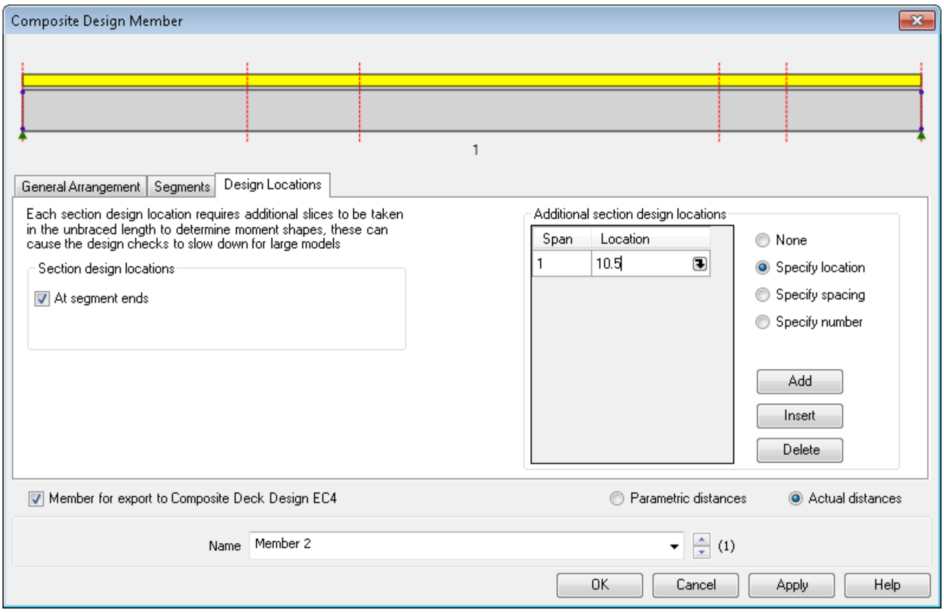


- In the Composite Design Member dialog ensure the **Member for export to Composite Deck Design EC4** is selected, this limits the inputs to only those required when design will be carried out in Ponti EC4.

- Select the Member element group to be the **Design Member** group



- Ensure **Actual distances** is selected and enter the segment lengths as shown above

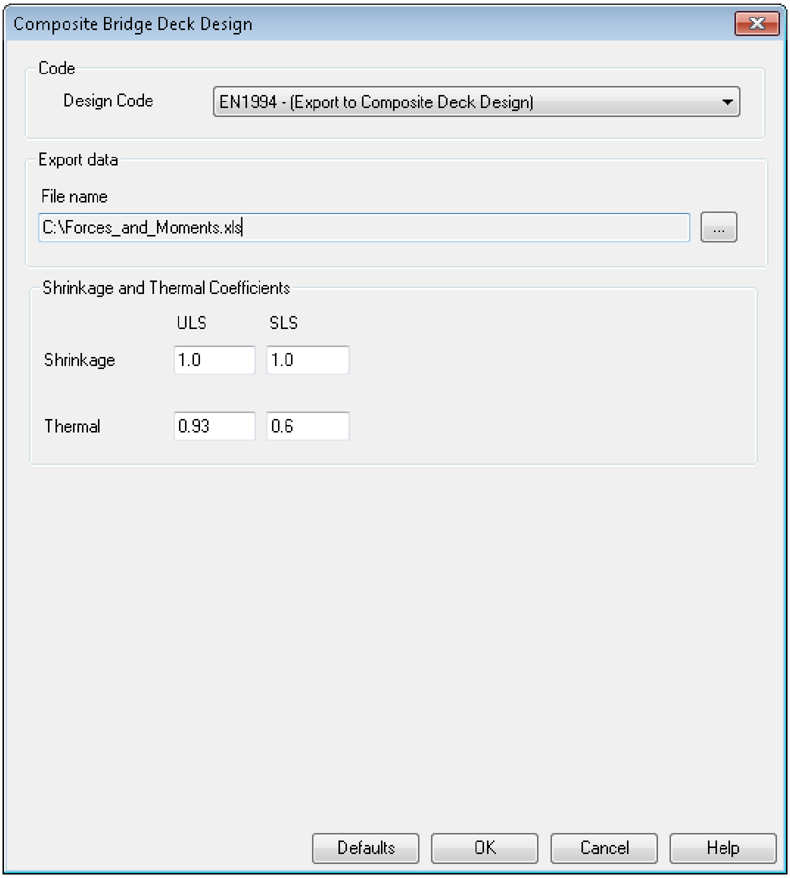


- In the design locations tab keep **At segment ends** selected and add a single additional design location at a distance of **10.5** (m) which corresponds to the influence assignment.

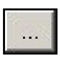
- Enter the name **Member 2** and click **OK** to create the member utility.

Selection of composite bridge deck design Code

The composite deck design code is set via the **Design > Composite Deck Design** menu item, and also the options for the export path and the factors for primary effects of shrinkage and thermal effects.



- Select **EN1994 – (Export to Composite Deck Design)** in the drop down list.

- Click the ellipsis button  to choose the folder and name of the xls file to create (e.g.: <current work directory>**Forces_and_Moments.xls**), pressing the **Save** button to return to the main dialog

- Input a Shrinkage coefficient of **1.0** for the ULS and SLS.

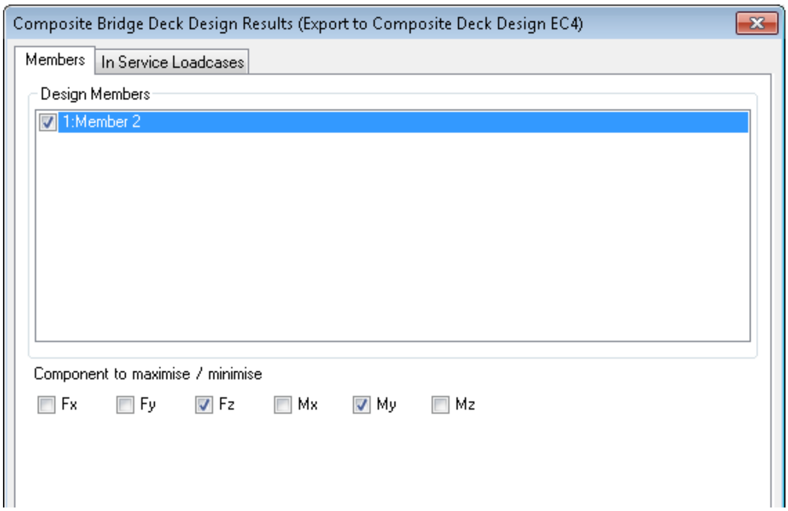
- Input a Thermal coefficient of 1.35x0.6 = **0.93** for the ULS and 1.0x0.6 = **0.6** for the SLS

Note. These coefficients will be applied to the characteristic primary effects of the shrinkage as directly calculated in PontiEC4. The isostatic effect will be neglected ($\gamma * \psi = 0$) in the cracked section (where Slab treatment = Cracked in Tension as defined in the geometric section). The hyperstatic effect of the shrinkage comes from the LUSAS model and the combination factors have been already applied in the LUSAS smart combinations.

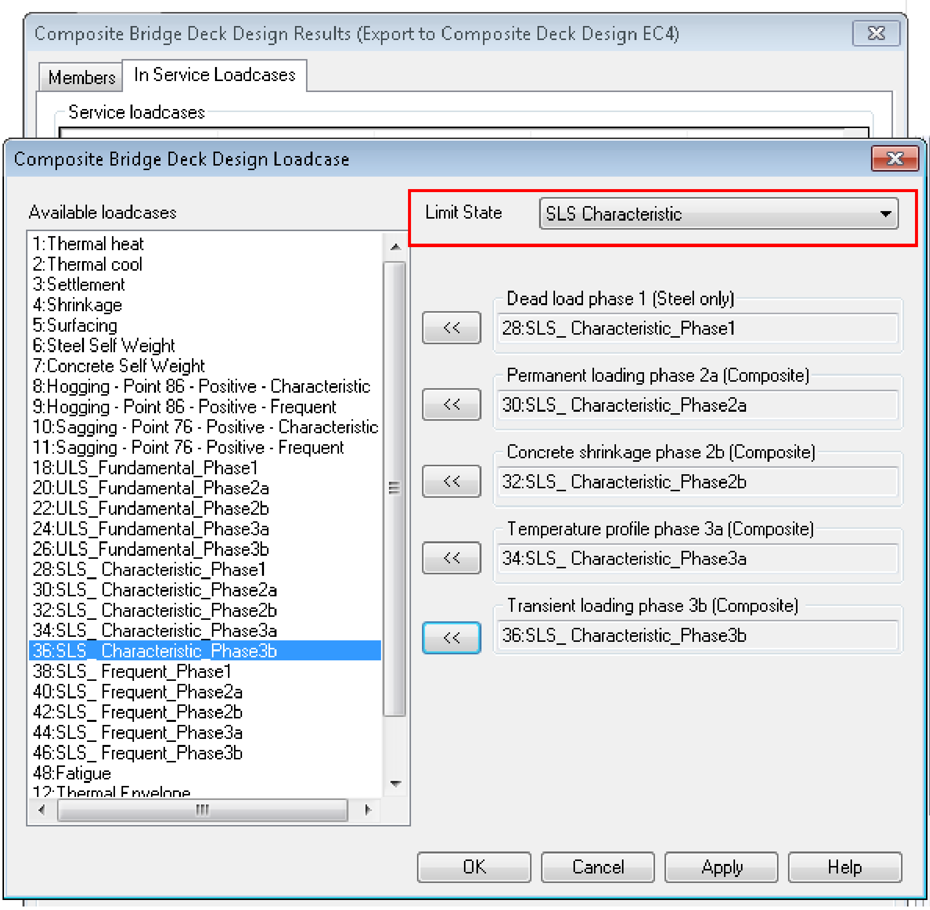
- Then, finally, click **OK** to return to the Modeller

Export to Composite Deck Design

Design
Export to Composite Deck Design



- Select the single design member **Member 2** and ensure the **Fz** and **My** components are selected to be used as primary components when exporting smart combination results.

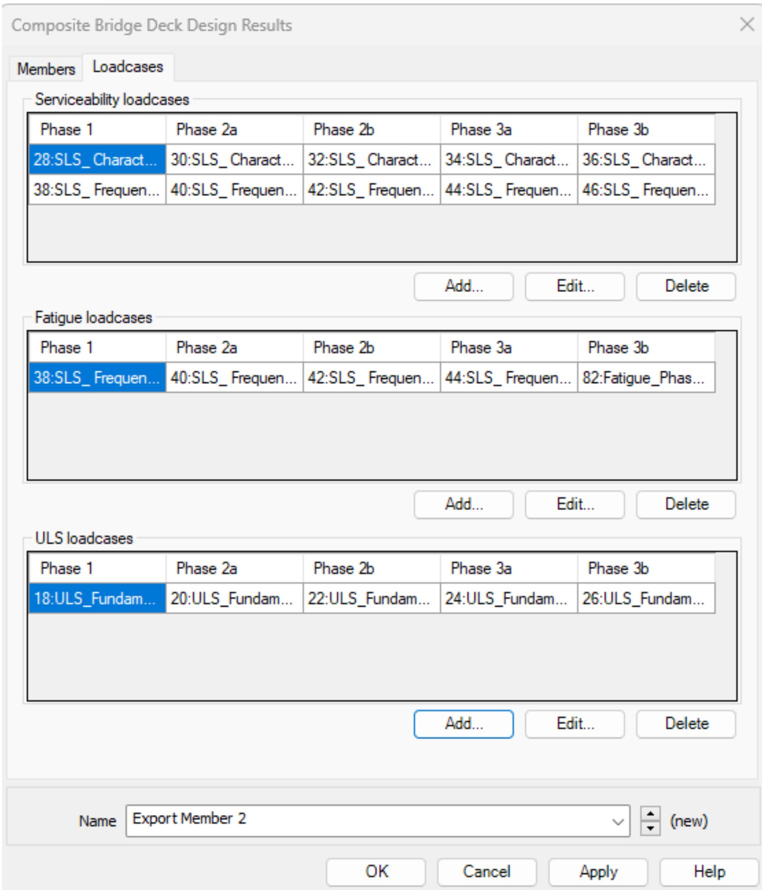


- In the **In Service Loadcases** tab click **add** in the service loadcase group. In the loadcase selection dialog ensure the limit state is set to **SLS Characteristic**. In turn select each characteristic loadphasee from the available loadcase list on the left and use the buttons to map them to the corresponding phases on the right. Click **OK**.

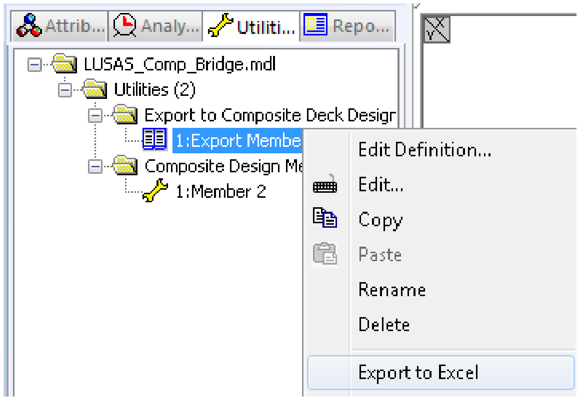
- Repeat this process for the service loadsets selecting **SLS Frequent** and mapping the Frequent loadcases

- Add the Frequent loadcases to the fatigue section taking care to select the Fatigue combination for phase 3b.

- Similarly add the fundamental combinations. The dialog should be as follows



- Change the name to **Export Member 2** and click OK to create a utility in the utilities treeview.
- In the Utilities  treeview right click on the newly created results utility and select **Export to Excel**



The forces and moment are now exported to an Excel spreadsheet in the specified folder ready for import to Ponti EC4.

Part 5: Importing force and moment data to the Composite Deck Designer (PontiEC4) and assessing design results

Associated File

This supplied file is only for use in case of problems in manually creating the appropriate data in the previous section of the example.

- ❑ **Forces_and_moments.xls** - File created by the LUSAS Export Forces and Moments dialog containing data to import into PontiEC4.
- ❑ **LUSAS_Comp_Bridge.csv** - the input file for PontiEC4 with data on sections and materials.

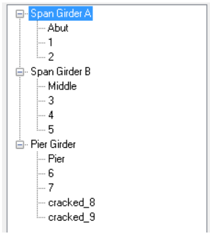
Importing force and moment data into PontiEC4

This section covers the import of section force and moment data from LUSAS into PontiEC4. Refer to the "Forces and moments" section in the PontiEC4 online help file for more information.

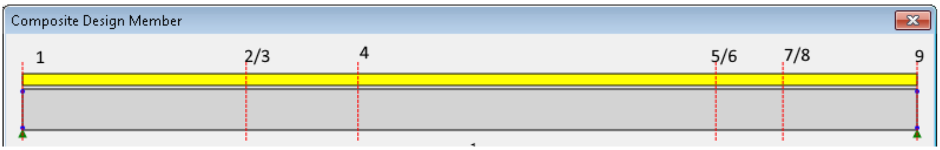
Whilst it is possible to manually enter forces and moments into PontiEC4 in the "Forces and moments" dialog, and it is also possible to copy and paste data into each section, when used with LUSAS any importing of data is far better achieved by using the automated procedure as follows.

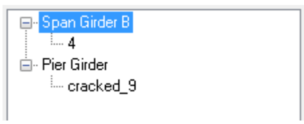
- In PontiEC4, with the current data still loaded (or with the data from the previously saved file **"LUSAS_Comp_Bridge.csv"** loaded if the example had been saved and closed for some reason at that point), select the **File> Import forces/moments** menu item and load the file **Forces_and_moments.xls** from working project folder.

PontiEC4 will detect that nine additional sections are present (these are the sections at the beginning and end of each beam element that was selected to signify a change of cross-section geometric property in the LUSAS model) and ask if they are to be added to their corresponding segments.



- Click **Yes** to import the new sections. This operation may take a few seconds to complete. The status bar will show the progress.
- From the main menu select **Windows> 2 Geometry** to display the Geometry dialog. In the Segment treeview the new sections added to each segment can be seen.
- With reference to the design member dialog below, when design locations at segment ends is selected two design locations will be created, one for each section. For "additional design locations" only a single location is created.





- For simplicity, in this example we will consider only the sections corresponding to our influence locations in the LUSAS model. For each section not required, click the right-hand mouse button, and choose **Delete**. This leaves the section treeview as shown.

Note. If any mistakes are made in deleting the unwanted sections, a re-import of the forces_and_moments.xls file will re-insert any deleted sections.

- Select **File> Save As** and enter a file name of **LUSAS_Comp_Bridge_Results.csv** to save the PontiEC4 dialog data to the working project folder before proceeding to view the results.

This concludes the data input.

Assess the results and detailed design checks

The design locations can now be assessed for Ultimate, Serviceability, and Fatigue limit states. It involves:

1. **Viewing utilisation factors for all checks performed**
2. **Updating section sizes for any over-utilised sections**

3. **Viewing updated results**
4. **Updating the LUSAS model and re-analysing the revised sections**

It is usually enough to cycle through steps 1 to 4 once, or perhaps twice, to obtain an optimized structure.

Limited details have been given about all the checks that are carried out. To find out more about the checks performed refer to the PontiEC4 online Help, where design code and theory details are also supplied.

Viewing design results in PontiEC4

- With the results loaded in **PontiEC4** select the **Window> Summary of results** menu item. This carries out the design computations in PontiEC4 and displays the results for deck sections in a summary form.

The meaning of the coloured utilization factor entries is explained in the following table:

Colour of value / text	Utilization factor	Meaning
Red	Greater than 1.0	Value has failed the check carried out.
Green	Less than 1.0	Value has passed the check carried out.
Grey - without parentheses	Any	Value of potential interest but the utilisation factor is not relevant to the code.
(Grey) - with parentheses	Any	Value of potential interest but the utilisation factor is not relevant to the code in this particular context.
Grey stating '**No int.**'	Not applicable	No interaction between bending and shear.

A check of the utilization factors obtained should be made for each limit state. Only a check for the Fundamental ULS combination will be shown here.

Note. By clicking on a particular header the values in a column can be sorted in increasing or decreasing value

Fundamental ULS combination

Section	X (m)	Combination	Class Ph.1	Class Ph.3b	MEd/MR	SigEd /fy	VEd/VRd	MEd/Mf,Rd	VEd/Vbw,Rd	V/M/N	vEd/(n*PRd)	End Studs	Longitudinal Stiffeners LTB	Vertical Stiffeners LTB	Vertical Stiffeners lst min/lst	Vertical Stiffeners Sig/(fy/gM1)	Vertical Stiffeners w/(hw/300)	Section
Span Girder B_4	10.500	Fund. ULS, Mmax	4	1	0.14	(.172)	0.046	0.17	0.049	No int.	0.011	0	0	0	0	0	0	Span Girder B_4
Span Girder B_4	10.500	Fund. ULS, Mmin	4	1	0.63	(.704)	0.171	0.72	0.177	No int.	0.151	0	0	0	0	0	0	Span Girder B_4
Span Girder B_4	10.500	Fund. ULS, Vmax	4	1	0.57	(.627)	0.235	0.65	0.245	No int.	0.137	0	0	0	0	0	0	Span Girder B_4
Span Girder B_4	10.500	Fund. ULS, Vmin	4	1	0.59	(.734)	0.218	0.72	0.226	No int.	0.176	0	0	0	0	0	0	Span Girder B_4
Pier Girder_crack...	28.000	Fund. ULS, Mmax	1	3	(.61)	0.789	0.485	0.73	0.485	No int.	0.473	0	0	0	0	0	0	Pier Girder_crack...
Pier Girder_crack...	28.000	Fund. ULS, Mmin	1	3	(.13)	0.323	0.148	0.15	0.148	No int.	0.035	0	0	0	0	0	0	Pier Girder_crack...
Pier Girder_crack...	28.000	Fund. ULS, Vmax	1	3	(.61)	0.789	0.485	0.73	0.485	No int.	0.473	0	0	0	0	0	0	Pier Girder_crack...
Pier Girder_crack...	28.000	Fund. ULS, Vmin	1	3	(.13)	0.323	0.148	0.15	0.148	No int.	0.035	0	0	0	0	0	0	Pier Girder_crack...

Characteristic SLS combination

Section	X (m)	Combination	Sig id / Sig amm	vEd / (ksnPRd)
Span Girder B_4	10.500	Charact. SLS, M...	0.185	0.015
Span Girder B_4	10.500	Charact. SLS, M...	0.527	0.159
Span Girder B_4	10.500	Charact. SLS, V...	0.451	0.15
Span Girder B_4	10.500	Charact. SLS, Vmin	0.571	0.199
Pier Girder_crack...	28.000	Charact. SLS, M...	0.654	0.54
Pier Girder_crack...	28.000	Charact. SLS, M...	0.343	0.055
Pier Girder_crack...	28.000	Charact. SLS, V...	0.654	0.54
Pier Girder_crack...	28.000	Charact. SLS, Vmin	0.343	0.055

Frequent SLS combination

Section	X (m)	Combination	Web breathing
Span Girder B_4	10.500	Freq. SLS, Mmax	0.221
Span Girder B_4	10.500	Freq. SLS, Mmin	0.262
Span Girder B_4	10.500	Freq. SLS, Vmax	0.304
Span Girder B_4	10.500	Freq. SLS, Vmin	0.197
Pier Girder_crack...	28.000	Freq. SLS, Mmax	0.177
Pier Girder_crack...	28.000	Freq. SLS, Mmin	0.096
Pier Girder_crack...	28.000	Freq. SLS, Vmax	0.177
Pier Girder_crack...	28.000	Freq. SLS, Vmin	0.096

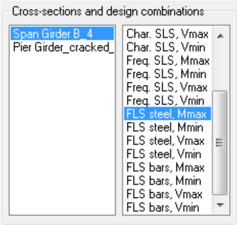
Fatigue combination

Section	X (m)	Combination	Studs Eta1	Studs Eta2	Studs Eta3	Flange top	Flange bot	Web	Fltop-Fltop	Flbot-Flbot	Web-Fltop	Web-Flbot	VStiff-Web	VStiff-Fltop	VStiff-Flbot	HStiff1-Web	HStiff2-Web	Bars	Section
Span Girder B_4	10.500	Fatigue LS, Mmax	0.107	0.998	0.851	0.639	0.678	0.156	0.783	0.831	0.680	0.689	0.965	0.952	0.965	0.000	0.000	0	Span Girder B_4
Span Girder B_4	10.500	Fatigue LS, Mmin	0.107	0.998	0.851	0.639	0.678	0.156	0.783	0.831	0.680	0.689	0.965	0.952	0.965	0.000	0.000	0	Span Girder B_4
Span Girder B_4	10.500	Fatigue LS, Vmax	0.24	0.038	0.213	0.024	0.026	0.349	0.029	0.031	0.026	0.026	0.036	0.036	0.036	0.000	0.000	0	Span Girder B_4
Span Girder B_4	10.500	Fatigue LS, Vmin	0.24	0.001	0.185	0.000	0.021	0.349	0.001	0.026	0.000	0.022	0.031	0.000	0.031	0.000	0.000	0	Span Girder B_4
Pier Girder_crack...	28.000	Fatigue LS, Mmax	0.197	0.187	0.296	0.120	0.161	0.133	0.147	0.197	0.122	0.162	0.227	0.171	0.227	0.000	0.000	0	Pier Girder_crack...
Pier Girder_crack...	28.000	Fatigue LS, Mmin	0.197	0.033	0.177	0.021	0.140	0.133	0.026	0.172	0.017	0.147	0.205	0.023	0.205	0.000	0.000	0	Pier Girder_crack...
Pier Girder_crack...	28.000	Fatigue LS, Vmax	0.267	0.167	0.334	0.107	0.143	0.180	0.131	0.176	0.109	0.145	0.203	0.153	0.203	0.000	0.000	0	Pier Girder_crack...
Pier Girder_crack...	28.000	Fatigue LS, Vmin	0.267	0.015	0.217	0.010	0.124	0.180	0.012	0.152	0.005	0.130	0.182	0.007	0.182	0.000	0.000	0	Pier Girder_crack...

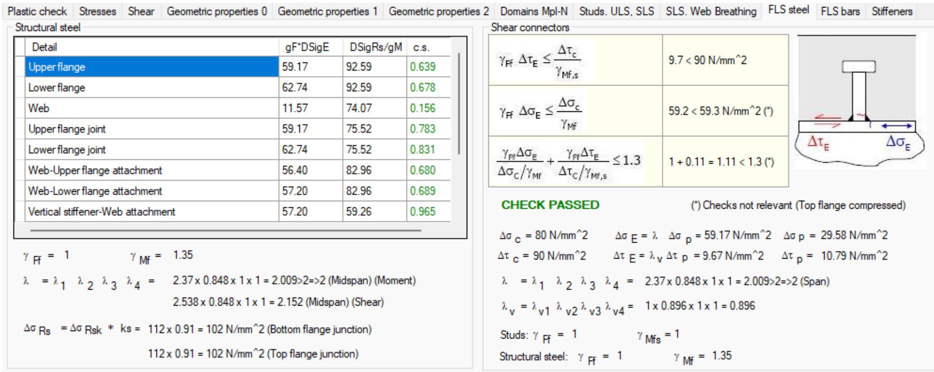
Note. Changing from the Geometry, Materials or Forces and moments dialogs to the Results, Summary of results, Report, or Cracking dialogs causes a re-analysis of results.

It can be seen that the summary of the results page is showing that all checks have passed- as shown in **Green**.

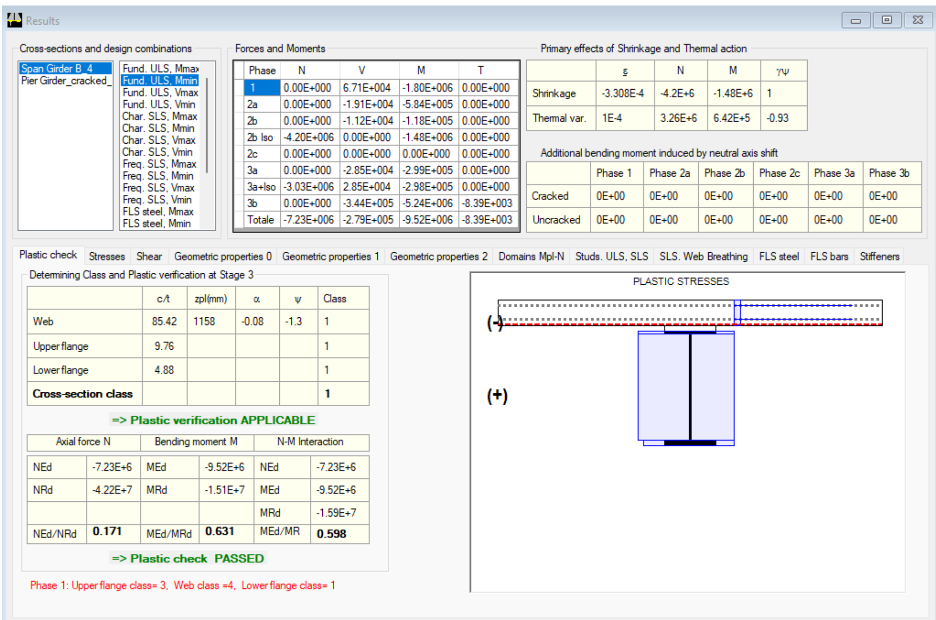
- Select the **Window> Results** menu item in order to find out more about a specific check.
- In the Cross-sections and design combinations panel select for example section **Span Girder B_4** and **FLS steel, Mmax**



- Then, in the lower part of the Results dialog, select the **FLS steel** tab:



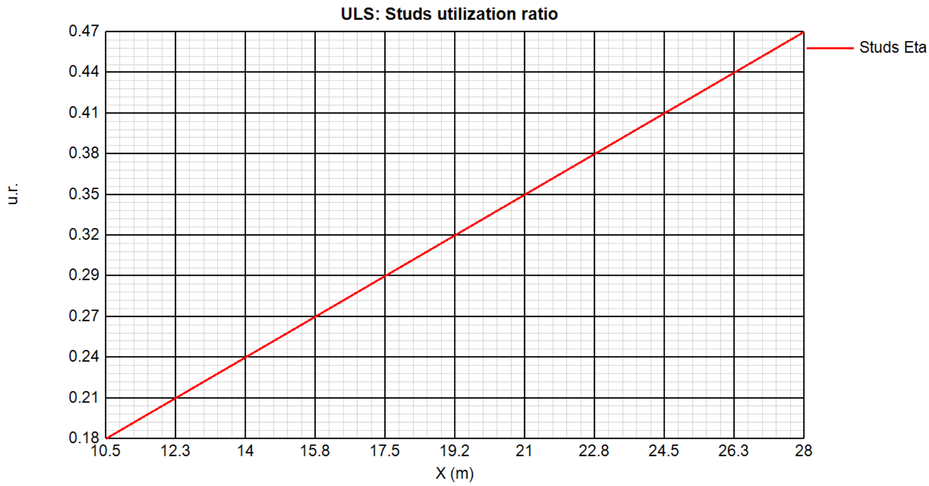
- In the ‘Cross-sections and design combinations’ panel, select section **Span Girder B_4** and **Fund. ULS, Mmin**.
- Select the **Stresses** tab to have a look at the elastic stresses on the gross section.
- Then, select the **Plastic check** tab. From this, all details about the classification and plastic check are supplied, and it can be seen that for Stage 3, this section, that is part of the segment in the middle of the span, is in Class 1.



Note. At the bottom-left of the Plastic check page the section classification is also provided for Phase 1, where the top (upper) flange cannot be considered to be restrained by the slab. For this case, the section is in Class 4 as highlighted in the red message. For this situation a check of stresses in Phase 1 should be performed, using an effective section rather than a gross section.

Creating Graphs

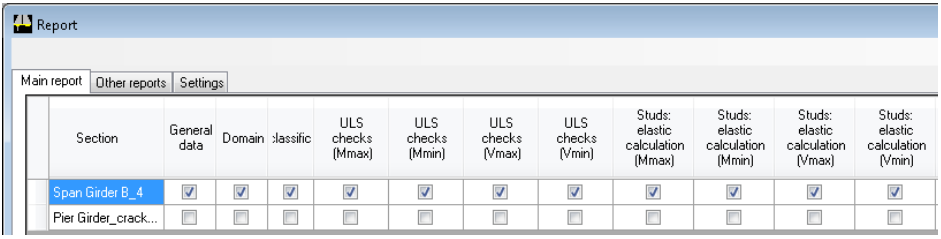
- In PontiEC4, with the optimized data still loaded, select the **Utilities> Graphs** menu item.
- From the first drop-down list choose **ULS: Studs utilization ratio** to plot the following graph:



Note. In this example the graph only contains 2 design locations and therefore varies linearly between these two points. Had we not deleted the other sections utilisations would have been provided for all section locations

Creating Reports

- In PontiEC4, with the optimized data still loaded, select the **Window > 6 Report** menu item.
- For section **Span Girder B_4** check (tick) to select all optional design checks, and then select the **Create report** button at the bottom-right corner of dialog.



- Enter a suitable file name for the report, in this case **Span Girder B**, and then press the **Save** button to create the report.

This completes the example.