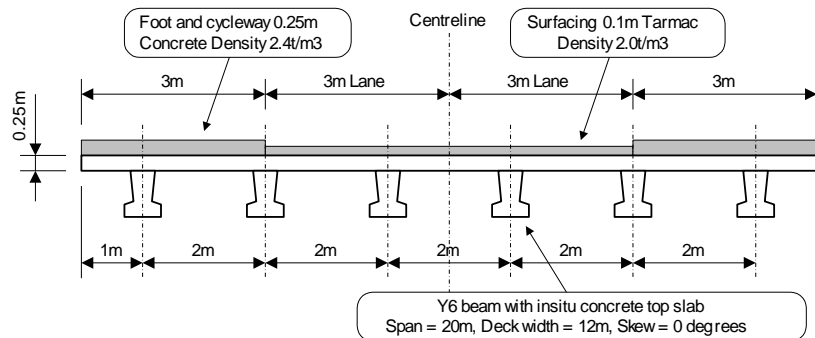


Simple Grillage

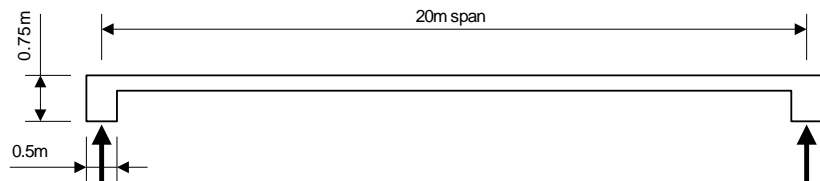
For LUSAS version:	19.0
For software product(s):	LUSAS <i>Bridge</i> .
With product option(s):	None.
This example can be used with the LUSAS Teaching and Training version by making one change to the entered data where shown.	

Description

A bridge deck is to be analysed using the grillage analogy. The geometry is as shown below.



Cross-section through deck



Longitudinal section along deck

All concrete members, both insitu and precast, are grade C50/60 to EN1992-2. Section properties for the end diaphragms are calculated using the Section Property Calculator. Parapet edge beams are treated as non-structural.

Simple Grillage

The structure is subjected to four loadcases: Dead load, Superimposed dead load, Group 1 loads in both lanes (LM1 tandems and lane loads), and Group 5 loads (LM3 special vehicle in one lane and LM1 in the other).

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



Note. When a bridge deck is modelled using the grillage analogy, the plan area of slab is represented by the longitudinal and transverse members.

Objectives

The required output from the analysis consists of:

- A deformed shape plot showing displacements caused by the imposed loading
- A diagram showing bending moments in the longitudinal members for the design load combination

Keywords

2D, Inplane, Y7 Precast Section, Section Property Calculation, Local Library, Grillage/Plate, Grillage, Basic Load Combination, Smart Load Combination, Enveloping, Deformed Mesh, Bending Moment Diagram, Print Results Wizard

Associated Files



- grillage_modelling.vbs** carries out the modelling of the example.

Modelling

Running LUSAS Modeller

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



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 the Grillage Model



Create a new model.

- Enter a file name of **grillage**

File
New...

- Use the **Default** user-defined working folder.
- Ensure an Analysis type of **Structural** is set.
- Select an Analysis Category of **2D Grillage/Plate**
- Set Model units of **kN,m,t,s,C**
- Leave the Timescale units as **Seconds**
- Ensure the Startup template is set as **None**
- Ensure the Layout grid is set as **None**
- Enter a Title of **Simple grillage analysis**
- Click the **OK** button



Note. Save the model regularly as the example progresses. Use the undo button to correct any mistakes made since the last save.

Using the Grillage Wizard

In this example the grillage wizard is used to generate a model of the bridge deck. The grillage wizard defines the grillage geometry, assigns grillage elements to each of the lines, and assigns supports to the end beams. It also creates Groups to ease member identification and the application of section properties.



Note. It is difficult to make absolute recommendations as to how individual structures should be modelled using a grillage. A few basic recommendations are however valid for most models:

- a) Longitudinal grillage members should be coincident with the main lines of stiffness i.e. passing over the supports and typically coincident with the girders (and edge beams, if these are to be treated as structural). A spacing less than span/4 is essential. A spacing of less than 3d is quite unnecessary
- b) Transverse members should have a spacing which is similar to, or greater than, that of the longitudinal members. The total number of transverse beams should be odd, to position nodes at mid span.

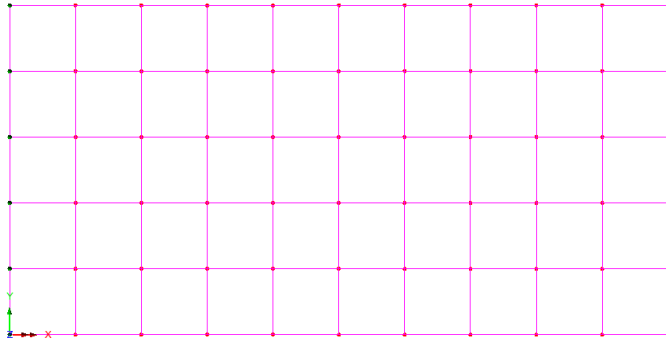
Bridge

Grillage Wizard...

- Select the **Set default** button
- Ensure **Slab deck** is selected and click **Next**
- The grillage is **Straight** with **0** degrees skew so click **Next** again
- Select **evenly spaced** longitudinal beams, and enter the Width of grillage as **10** and the number of longitudinal (including edge beams) as **6**. Click **Next**

Simple Grillage

- Leave the number of spans set as **1**
- Enter the length of span as **20** and the number of internal transverse beams as **9**
- Click **Finish** to generate the grillage model.



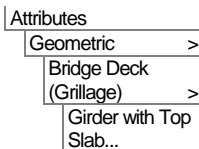
Defining the Geometric Properties

Bridge deck (grillage) attributes will be used to represent the deck sections. A rectangular solid section is also used to represent the end diaphragm beam. In total, three geometric attributes need to be created, namely:


- A **'girder with top slab' grillage attribute** representing a precast girder with slab.
- A **'slab' grillage attribute**, representing transverse members other than diaphragm beams.
- A **'girder with top slab' grillage attribute** representing an end diaphragm beam with slab

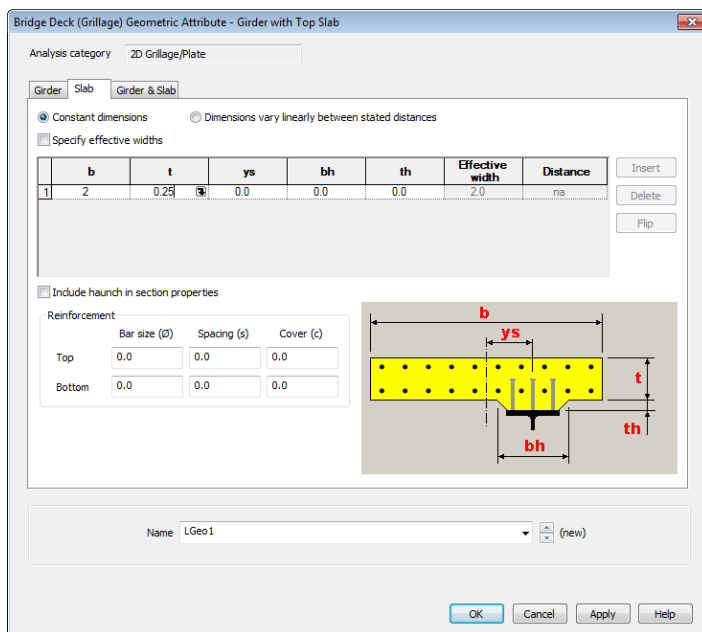
It is important to note that the section properties for grillage members are not calculated in the same way as beam sections, in particular, the torsional constants are based on strain energy, taking into account the way that longitudinal and transverse members together represent the twisting action which would be otherwise represented using plate elements. The software uses formulae taken from Hambly's "Bridge Deck Behaviour" and summarised in CSA S6-14 Annex A5.1.

Defining 'girder with top slab' grillage attribute



The precast girder is selected from one of the standard pre cast beam shapes provided in the section library.

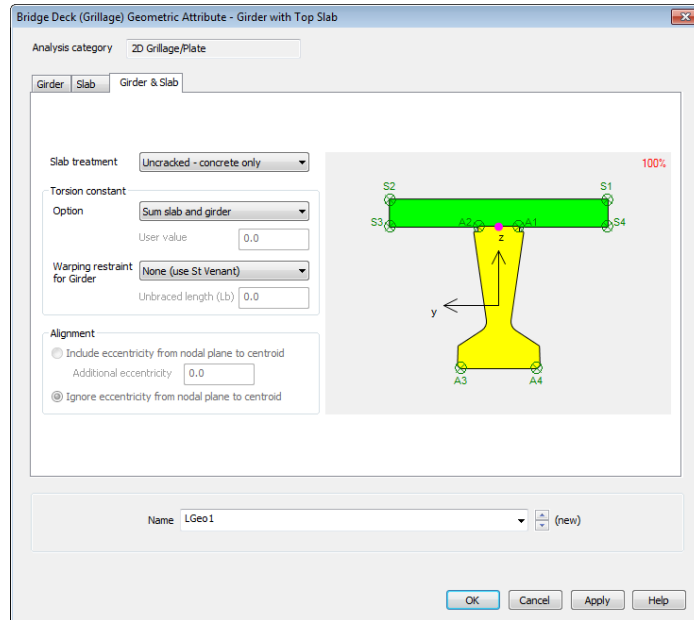
- With the **Girder** tab selected, define a **Constant section**.
- Click on the  image in the Section cell to allow a user-defined section to be chosen from a library.
- On the 'Enter Section' dialog, select **UK sections** from the **Precast Y Beams** library and choose the **Y7** section. Then click **OK** to return to the previous dialog.
- Next, select the **Slab** tab.



- Ensure **Constant dimensions** is selected. In this example, the reduction in slab width to account for shear lag is being neglected (as is customary in some Codes of Practice e.g. AASHTO). In practice, if this is to be considered, the 'Specify effective widths' check box allows values to be entered in the grid, which may vary linearly between stated distances across the span of the bridge deck.
- Enter a slab width (b) of **2** and a slab thickness (t) of **0.25**
- The slab section is centrally positioned with respect to the girder and so the offset to the centre of the slab from the girder (ys) is left unaltered as **0**.

Simple Grillage

- There is no thickening or “haunch” in the concrete slab above the girder, so there is no need to enter any further dimensions.
- Lastly, select the ‘**Girder & Slab**’ tab.



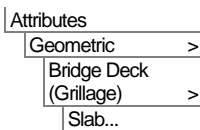
- For the slab treatment ensure that **Uncracked (concrete only)** is selected. (The ‘Uncracked (gross section)’ option uses an increased stiffness to account for any reinforcing bars specified, a refinement often felt unnecessary by practicing engineers.)

The torsional constant calculations settings are left as their default settings (‘Sum slab and girder’; ‘no warping restraint – use St Venant’). The warping restraint options include those from the NCHRP Report 725 (Section 3.3.2) and have been shown to be superior, particularly for cases of high skew and/or curved girders.

- Name the geometric attribute as **Y7 and slab** and press **OK**.

Defining a ‘slab’ grillage attribute

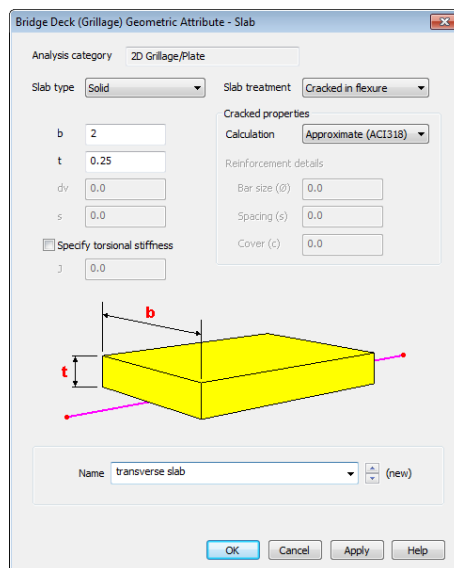
‘Slab’ grillage attributes are used to represent the transverse slab members. These model a 1m width of slab either side of the grillage member.



- Select a **Solid** slab type and enter a slab width (b) of **2** and slab thickness (t) of **0.25**.
- Ensure the slab is treated as **Cracked in flexure** with the Cracked properties calculated according to **Approximate (ACI318)**.
- Name the attribute as **transverse slab** and click **OK**.

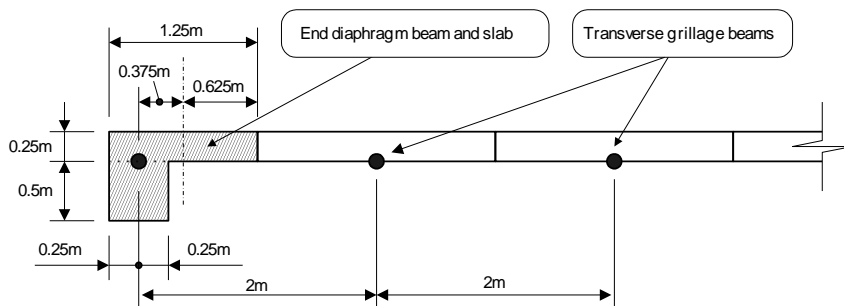


Note. Cracked properties are calculated according to ACI318 clause 10.10.4.1 (b) and (c)), which essentially takes the cracked bending stiffness to be 25% of the gross uncracked stiffness. Alternatively, if the reinforcement dimensions are known, the ‘From reinforcement’ option may instead be used and the additional dimensions entered.



Defining the end diaphragms

Finally the end diaphragm beams will be represented using another ‘girder with top slab’ grillage attribute, where a 0.5m x 0.5m rectangular solid section will model the drop-beam beneath a slab of 1.25m width and 0.25m thickness that is offset from the beam.



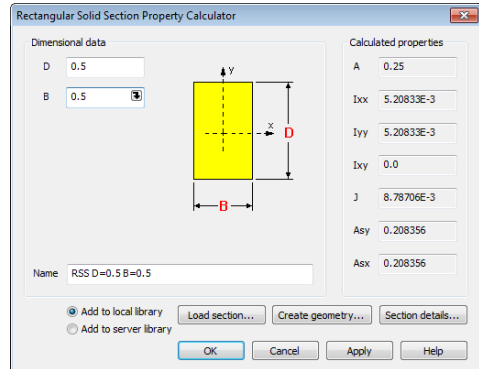
Longitudinal section near diaphragm



Note. This section could alternatively be represented using a ‘single ‘L-shaped section’. Such an approach would give the correct bending stiffness but overestimate the torsional stiffness.


Defining a rectangular solid section

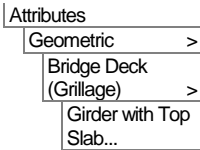
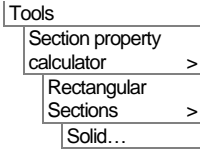
- Enter a drop-beam depth (D) of **0.5** and a beam width (B) of **0.5**
- Ensure that **Add to local library** is selected.
- Accept the default attribute name of **RSS D=0.5 B=0.5** that is created, and click **OK**.

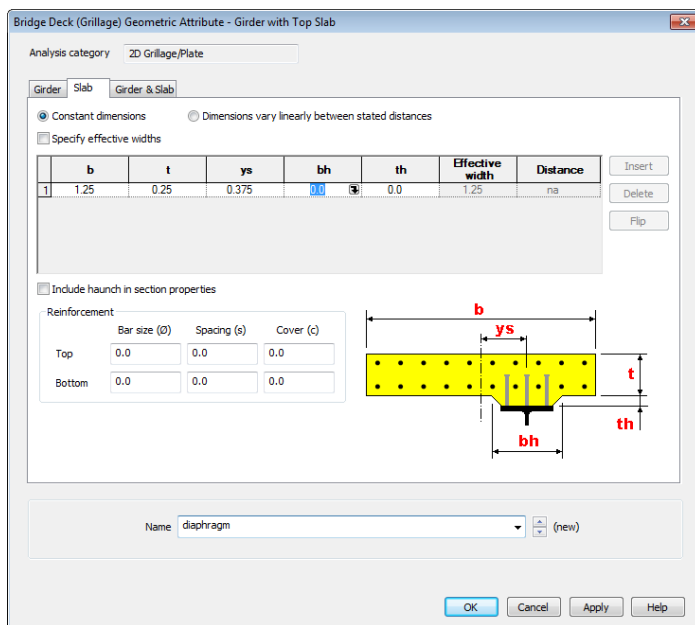


Defining a 'girder with top slab' grillage attribute for the diaphragms

The diaphragm properties may be defined by selecting the relevant User Section on the girder slab.



- With the **Girder** tab selected, define a **Constant section**.
- Click on the  image in the Section cell to allow a user-defined section to be chosen from a library.
- On the 'Enter Section' dialog, select **User sections** from the **Local** library and choose the **RSS D=0.5 B=0.5** section. Then click **OK** to return to the previous dialog.
- Next, select the **Slab** tab.





- Ensure **Constant dimensions** is selected.
- Enter a width (b) of **1.25** and a thickness (t) of **0.25**
- The slab is offset with respect to the girder and so the offset to the centre of the slab from the girder (ys) is specified as **0.375** (see earlier image labelled ‘Longitudinal section near diaphragm’)
- There is no thickening or “haunch” in the concrete slab above the girder and so no need to enter any further dimensions.
- Lastly, select the ‘**Girder & Slab**’ tab.
- For the slab treatment ensure that **Uncracked (concrete only)** is selected. (The ‘Uncracked (gross section)’ option uses an increased stiffness to account for any reinforcing bars specified, a refinement often felt unnecessary by practicing engineers.)
- Name the geometric attribute as **Diaphragm** and press **OK**.

Assigning Geometric Properties

This is best done by using copy and paste in the  and  Treeview panes.

Simple Grillage









Use the Isometric View button to rotate the model so that the following assignments of the geometric properties can be seen.

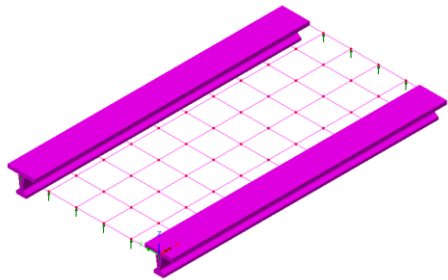


Ensure the fleshing button is depressed in the toolbar menu.

Longitudinal members





The 'Y7 and slab' attribute is assigned to all the longitudinal members - the edge beams and the internal longitudinal beams.

- In the  Treeview select the 'Y7 and slab' entry and click on the  copy button.
- In the  Treeview select the **Edge Beams** group and click the  paste button to assign the Y6 beam section properties to the edge beams.
- An assignment dialog clarifies that this assignment is to lines in Analysis 1, and the edge beams will be visualised in the view window.
- Now, in the  Treeview, select the **Longitudinal Beams** group and click the  paste button again to assign the 'Y7 and slab' properties.





Transverse slab members

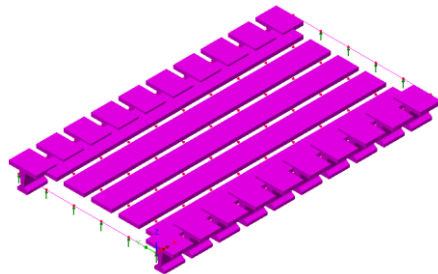
The slab section properties are assigned to the transverse members in a similar fashion.

- In the  Treeview select the **Transverse Slab** entry and click on the  copy button.
- In the  Treeview select the **Transverse Beams** group and click the  paste button to assign the slab section properties.





Diaphragm members

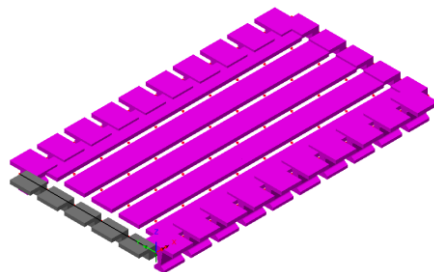
To clarify the display prior to assigning the diaphragm members the extent of each fleshed grillage member can be modified as follows:

-  From the main toolbar, select the black arrow to the right of the Fleshing button to display the shrinkage options.
- Select the **End shrinkage auto** option.
- In the  Treeview, right-click on the **Longitudinal Beams** entry and select **Invisible** from the context menu. This makes it easier to see the orientation of the end diaphragm members when they are assigned.



The diaphragm section properties are assigned to the end diaphragms in a similar fashion to that done for the other members.


- In the  Treeview select the **diaphragm beam** entry and click on the  copy button.
- In the  Treeview select the **End Diaphragms** group and click the  paste button to assign the slab section properties.




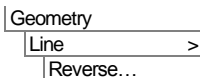
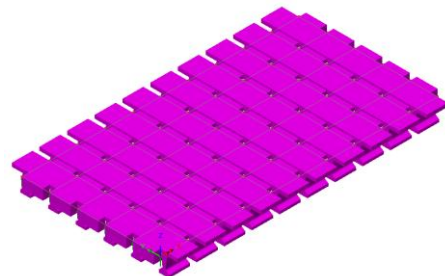
From the fleshed image it can be seen that the end diaphragm members for the near end are incorrectly orientated. The directions of the lines which have that assignment need to be reversed.

- Select the five lines at the near end of the grillage



This reverses the line directions of the selected lines.

- In the  Treeview right-click on the **Longitudinal Beams** entry and select **Visible** from the context menu.

 Turn off the display of the fleshed members





Note. Assignments can also be checked by right-clicking on a group name in the  Treeview and choosing the **Select Members** option, or by right-clicking on an attribute name in the  Treeview and choosing **Select Assignments**.

Defining the Materials

The materials needed for the Bridge deck (grillage) geometric attributes are defined using corresponding Bridge deck (grillage) material attributes.

To obtain correct deformations for the different loads applied to this grillage model, in principle three analyses are required, each referencing appropriate material properties:

- Analysis 1 – Short term** - This is for distribution of short term loads such as traffic loading – both girder and slab have stiffness according to Table 3.1 of EN1992-1-1.
- Analysis 2 – Long term** - This is for distribution of long term loads such as surfacing – both the girder and the slab have stiffness reduced to account for creep deformations. In this example, this is approximated by using one third of the elastic modulus of the short-term case.
- Analysis 3 – Dead load** - This is for distribution of dead load stresses, which occurs while the girders are in place but the insitu concrete slab is wet. This includes the mass of both slab and girder but the long-term stiffness of the precast girders only.

The materials required for these analyses are defined and assigned in turn, using a new analysis for each material and its appropriate loading.



Note. Where deformations are not required, relative stiffness rather than absolute stiffness is the important criteria for the distribution of stresses and for an all-concrete bridge such as this, long term and short term loading can use a single analysis corresponding to Analysis 1 above.



Note. With regard to dead loads, technically the distribution of stress would be according to Analysis 3 above at the time of construction but creep deformations would mean some re-distribution of stresses with time more similar to Analysis 2. However pragmatically this is usually overlooked with the simple distribution in Analysis 3 being conservative in most cases. For a more accurate approach, a nonlinear/transient analysis using the viscous creep and shrinkage materials may be adopted.

Short term material properties

On the dialog make the following selections:

Slab

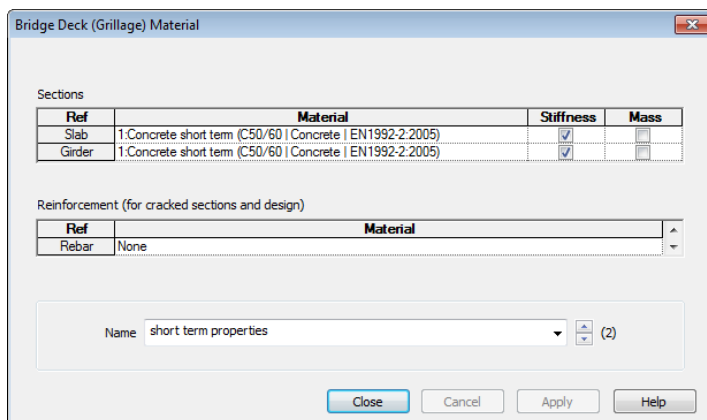
- Click in the material cell and choose **New...**


Attributes	
Material	>
Bridge Deck (Grillage)	

- On the material library dialog select material type **Concrete**, region **Europe**, standard **EN1992-2:2005**, and grade **C50/60**. Name the material **Concrete short term** and press **OK**.
- Ensure that the **Stiffness** is included by checking (ticking) that box, but the **Mass** is excluded.

Girder

- Click in the material cell and choose **Concrete short term (C50/60 | Concrete | EN1992-2:2005)**
- Ensure that the **Stiffness** is included by checking (ticking) that box, and ensure that **Mass** is excluded.
- Name the material **Short term properties** and click **OK**.



- With the whole model selected (**Ctrl** and **A** keys together) drag and drop the material attribute **Short term properties** from the  Treeview onto the selected features, assignment it to **Analysis 1**.



Note. No rebar material is required since no rebar arrangements have been specified in the geometric attributes. In other cases, of course, different grades or indeed materials may be used for the girder as compared to the slab and a rebar material may be needed.

Long term material properties

Long term properties will be set to be one third of the elastic modulus of the short-term properties. First, create the appropriate concrete properties:

- In the  Treeview right-click on the **Concrete short term** attribute and select **Edit Attribute** from the context menu.

Simple Grillage

- Edit the name of the attribute to be **Concrete long term**. This ensures that the short term attribute will not be overwritten.
- Change the Young's modulus value to be **12e6** and press **OK**.



Note. This change could also have been made by appending **/3** to the value already present.

Now, define a new Bridge Deck (Grillage) material using the new Long term properties for both the slab and the girder:

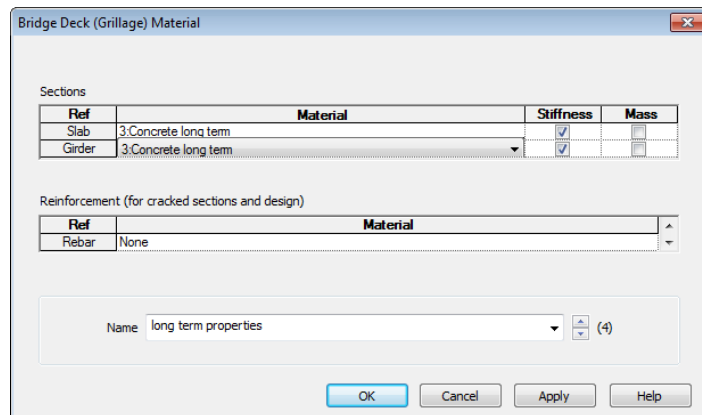
On the dialog make the following selections:

Slab

- Click in the material cell and choose **Concrete long term**
- Ensure that the **Stiffness** is included, but the **Mass** is excluded.

Girder

- Click in the material cell and choose **Concrete long term**
- Ensure that the **Stiffness** is included, but the **Mass** is excluded.
- Name the material **Long term properties** and click **OK**.




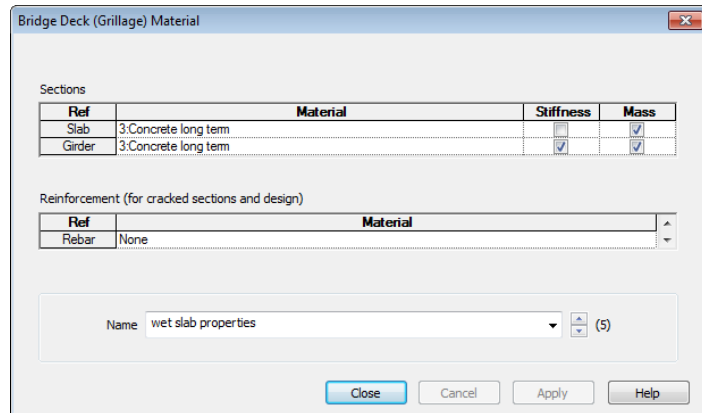
- With the whole model selected (press the **Ctrl** and **A** keys together) drag and drop the material attribute **Long term properties** from the Treeview onto the selected features, assigning it to a **New** analysis named **Analysis 2 – Long term**.


Dead load only material properties

To model dead load only, long term properties are used but omitting the stiffness of the slab and end diaphragms. Two properties will be required.


Wet Slab material

- In the  Treeview right-click on the **Long term properties** attribute and select **Edit...** from the context menu.
- For the Slab row ensure that **Stiffness** is unchecked and **Mass** is checked.
- For the Girder row ensure that **Stiffness** is checked and **Mass** is checked.



- Name the attribute to be **Wet slab** and press OK.
- With the whole model selected (Press the **Ctrl** and **A** keys together) drag and drop the material attribute 'Wet slab properties' from the  Treeview onto the selected features, with the assignment being for a New analysis named **Analysis 3 – Dead load**

Wet diaphragm material

- In the  Treeview right-click on the **Wet slab properties** attribute and select **Edit...** from the context menu.
- For the Slab row ensure that both **Stiffness** and **Mass** are 'unchecked'.
- For the Girder row ensure that **Stiffness** is unchecked and **Mass** is checked.



Simple Grillage

Ref	Material	Stiffness	Mass
Slab	3.Concrete long term	<input type="checkbox"/>	<input type="checkbox"/>
Girder	3.Concrete long term	<input type="checkbox"/>	<input checked="" type="checkbox"/>

Ref	Material
Rebar	None

Name: wet diaphragm properties (new)

OK Cancel Apply Help

- Name the attribute to be **Wet diaphragm properties** and press **OK**.
- In the  Treeview right-click on the **Diaphragm beam** attribute and choose **Select assignments** from the context menu. Then drag and drop the material attribute **Wet diaphragms** from the  Treeview onto the selected features, assigning it to **Analysis 3 – Dead load**.

Supports

The deck is supported in the vertical direction at each of the diaphragms. Since there are no in-plane degrees of freedom, no in-plane supports are required. The grillage wizard automatically restrains the grillage from vertical displacement at each diaphragm.



Note. Some engineers choose to use 3D beam elements in order to use a grillage-type model to consider in-plane effects such as braking forces. In accordance with the recommendations of Hambly, this is, broadly speaking, not recommended.

Loading

In this example seven loadcases will be applied to the grillage. These will be enveloped and combined together to form the design combination.

Dead Load

Dead load is made up of the self-weight of the girders, the weight of the wet concrete from the slab, and the weight of the wet concrete from the end diaphragms.






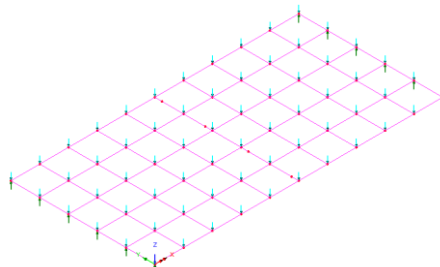
Note. When a bridge deck is modelled using the grillage analogy, the plan area of slab is represented by the longitudinal and transverse members. As a result, gravity should only be applied to the longitudinal members only.




Turn off the display of the fleshed members.

Bridge
 Bridge Loading >
 Gravity

- A load dataset named **Gravity vertical** will be added to the  Treeview.
- In the  Treeview right-click on the **Y7 and slab** attribute and choose **Select assignments** from the context menu. Then drag and drop the loading attribute **Gravity vertical** from the  Treeview onto the selected features, ensuring it is assigned to **Analysis 3 – Dead load and Loadcase 3**.



Loading arrows confirming the assignment of gravity to the longitudinal grillage members will be displayed.


- In the  Treeview expand **Analysis 3 - Dead load** and right click on **Loadcase 3** and select the **Rename** option.
- Rename the loadcase to be **Dead load** by over-typing the previous name.

Superimposed Dead Load

Superimposed dead load consists of the surfacing loads. These represent the self-weight of the foot and cycleways and the surfacing on the road.

Foot and cycleway surfacing

Specify the surfacing loading for the foot and cycleway:


- Leave the density as **2.4** (reference EN1991-1-1 table A.6)
- Change the thickness to **0.25**
- Set the length to **20** and set the width to **3.0**
- Leave the skew angle as **0** and the origin as **Centre**
- Click the **Apply** button to add a **Pch2 (Surfacing 20.0m x 3.0m Skew=0.0deg Thickness=0.25m Density=2.4 t/m³)** loading attribute to the  Treeview.

Now specify the tarmac highway surfacing load:

- Change the density to **2.0**
- Change the thickness to **0.1**
- Leave the length as 20 and the width as 3.0

Bridge
 Bridge Loading >
 Surfacing...

Simple Grillage

- Click the **OK** button to add a **Pch3 (Surfacing 20.0m x 3.0m Skew=0.0deg Thickness=0.1 Density=2.0t/m³)** loading attribute to the  Treeview.

Discrete point and patch loads are positioned by assigning them to points which do not necessarily have to form part of the model.

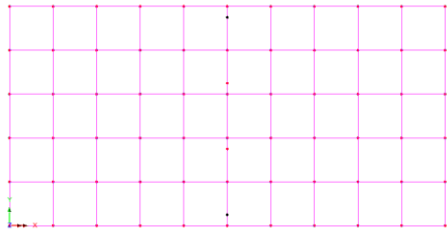


Enter the coordinates of the mid-point of each foot and cycleway and each notional lane **(10,0.5), (10,3.5), (10,6.5), (10,9.5)** and click **OK**

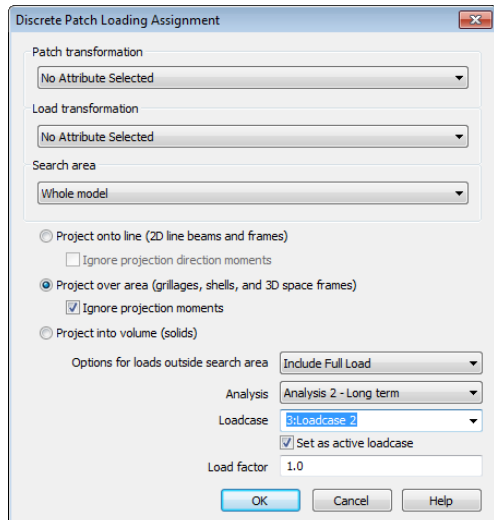
Z: N/A

On the status bar at the bottom of the display, click the Z axis button to return to a global Z direction view.

- Select the two previously defined points at the centre of each foot and cycleway by holding the **Shift** key down to add another point to the initial selection.
- Drag and drop the discrete loading dataset **Pch2 (Surfacing 20.0m x 3.0m Skew=0.0deg Thickness=0.25m Density=2.4 t/m³)** onto the selected points.




- On the Discrete Patch Loading Assignment dialog select **Include Full Load** from the drop down list. This will ensure the portion of the pavement load which is overhanging the edge of the grillage model is applied to the edge beams.
- Select **Analysis 2 – Long term** from the droplist and click **OK** to assign the loading to **Loadcase 2**.




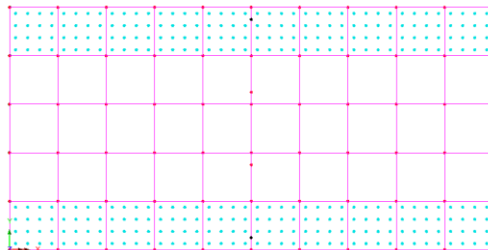
The (currently incorrect) patch loading pattern will be visualised.

Patch load divisions

The Patch divisions object seen in the  Treeview controls the number of discrete point loads used to represent a patch load. By default a specified number of 10 divisions is used. However, for this example, and for most real life uses a greater number of divisions is required to accurately reflect the surfacing loading.


Geometry
Point >
By Coords...

- In the  Treeview double-click the **Patch divisions** object
- Select the **Distance between loads** option and specify **0.5**. Click **OK** to update the patch divisions and see the updated loading visualisation.




Roadway surfacing

Now the road surfacing is to be assigned:

- Select the two points at the centre of each notional lane.
- Drag and drop the discrete load dataset **Pch3 (Surfacing 20.0m x 3.0m Skew=0.0deg Thickness=0.1 Density=2.0t/m³)** from the  Treeview onto the selected points.
- Leave 'Options for loads outside the search area' set as **Exclude All Load** because for this load type it is irrelevant whether include or exclude is used since the load length, which is positioned centrally, is the same length as the span length
- Select **Analysis 2 – Long term** from the droplist and click **OK** to assign the loading to **Loadcase 2**.
- Click **OK** to assign the road surfacing load.

Loading arrows confirming the assignment of surfacing in both the foot and cycleway and roadway will be displayed.

- In the  Treeview expand **Analysis 2 – Long term** and right click on **Loadcase 2** and select the **Rename** option.
- Rename the loadcase name to be **Surfacing** by over-typing the previous name.

Vehicle Loading

For the purposes of this example, static vehicle loads and lane loads will be applied to the model.


In LUSAS, moving loads can also be defined or vehicle load optimisation software can be used to automatically identify the most onerous vehicle loading patterns on bridges for a supported design code and to apply these loading patterns to LUSAS models. A separate worked example titled 'Grillage Load Optimisation' shows the vehicle load optimisation capability.

Static vehicle loading


LM1 loading is to be applied to each notional lane. These loads are defined using the Eurocode vehicle loading definitions supplied with LUSAS *Bridge*.

LM1 Lane load

Lane 1 UDL loading will be specified first:


- Select the **Load Model 1 - Lane Load** button.
- Leave the width as **3.0**. Change the length to **20**.
- Ensure the surface load is set to **9.0**. Note that the droplist offers 9.0 and 2.5 in keeping with EN1991-2 Table 4.2, however other values may be typed into the box to suit National Annexes where this value is amended by an α -factor as per clause 4.3.2(3).
- Click the **Apply** button to add the load dataset **Pch4 (Eurocode lane Load 20.0m x 3.0m Skew=0.0deg Load = 9.0kN/m2 (Centre))** to the  Treeview, and allow more loading to be defined.

Now, Lane 2 UDL loading will be defined.


- Leave the width as **3.0** and the length as **20**.
- Change the surface load to **2.5**
- Click the **OK** button to add a **Pch5 (Eurocode lane Load 20.0m x 3.0m Skew=0.0deg Load = 2.5kN/m2 (Centre))** loading attribute to the  Treeview.

LM1 Tandem loading

Using the same dialog, Lane 1 Tandem loading will be specified first:

- Select the **Load Model 1 - Tandem** button.
- Leave the Tandem System droplist with **Lane number 1 – 300kN** selected
- Click the **Apply** button to add the load dataset **Pnt6 (Eurocode Load Model 1 300kN)** to the  Treeview.

Now Lane 2 Tandem loading will be defined.

- Change the Tandem System droplist to select **Lane number 2 – 200kN**
- Click the **OK** button to add the load dataset **Pnt7 (Eurocode Load Model 1 200kN)** to the  Treeview.

Bridge
Bridge Loading >
Eurocode...

LM3 Special Vehicle

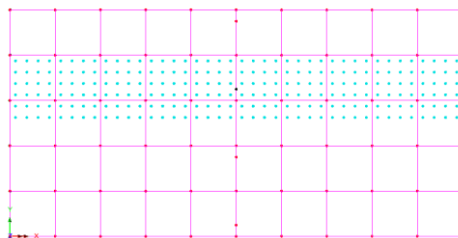
The 1200/200 special vehicle from EN1991-2 Annex A Table A1 will be used. This has 6 axle-lines of 200kN each as per Figure A1(a), with a spacing of 1.5m according to Table A2

- On the same dialog, select the **Load Model 3** button.
- From the Vehicle Type droplist select **1200/200**
- Click the **OK** button to add the load dataset **Pnt8 (Eurocode Load Model 3 1200/200)** to the Treeview.
- Click the **Close** button to close the Eurocode bridge loading dialog.

Assigning LM1 Lane Loading

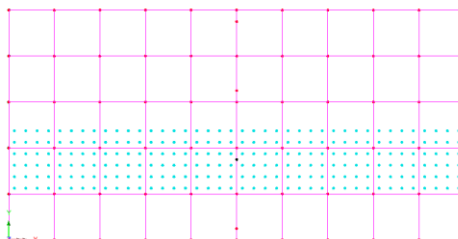
Lane load upper

- Select the point defined at the centre of the upper notional lane.
- Drag and drop the dataset **Pch4 (Eurocode lane Load 20.0m x 3.0m Skew=0.0deg Load = 9.0kN/m2 (Centre))** from the Treeview.
- Ensure the Analysis droplist is set to **Analysis 1**, the Loadcase is **Loadcase 1**, leave other values as their defaults, and click **OK**. The loading will be visualised.
- In the Treeview expand **Analysis 1** and right click on **Loadcase 1** and select the **Rename** option.
- Rename the loadcase to **Lane load upper** by over-typing the previous name.



Lane load lower


- Select the point defined at the centre of the lower notional lane.
- Drag and drop the dataset **Pch5 (Eurocode lane Load 20.0m x 3.0m Skew=0.0deg Load = 2.5kN/m2 (Centre))** from the Treeview.
- Ensure the Analysis droplist is set to

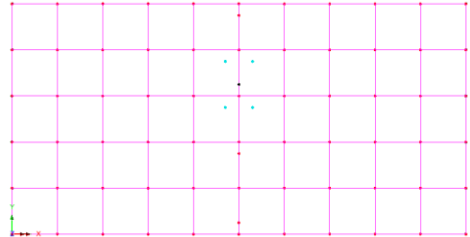


Analysis 1, enter the loadcase name as **Lane load lower**, leave other values as their defaults, and click **OK**. The loading will be visualised.


Assigning Tandem Loading

Tandem load upper

- Select the point defined at the centre of the upper notional lane.
- Drag and drop the dataset **Pnt6 (Eurocode Load Model 1 300kN)** from the  Treeview.
- Enter **Tandem load upper** as the Loadcase name, leave the other values as their defaults and click **OK**. The loading will be visualised.




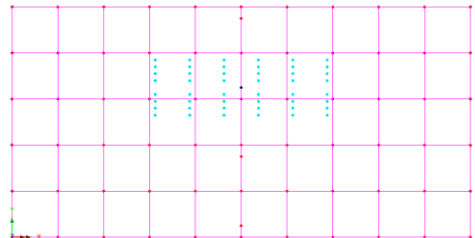
Tandem load lower


- Select the point defined at the centre of the lower notional lane.
- Drag and drop the dataset **Pnt7 (Eurocode Load Model 1 200kN)** from the  Treeview.
- Enter **Tandem load lower** as the Loadcase name, leave other values as their defaults, and click **OK**. The loading will be visualised

Assigning LM3 Special Vehicle

For this example, the Load Model 3 vehicle is only assigned to the upper lane.

- Select the point defined at the centre of the upper notional lane.
- Drag and drop the dataset **Pnt8 (Eurocode Load Model 3 1200/200)** from the  Treeview onto the selected point.
- Enter **LM3 upper** as the Loadcase, leave other values as their defaults, and click **OK**. The loading will be visualised



- **Note.** Loading assignments can be checked in a number of ways. This can be done by either selecting a point and accessing its context menu to view its Properties (which will include Loading), or a loading attribute in the Attributes  Treeview can be interrogated by accessing its context menu and choosing 'View

Assignments', or a Loadcase and loading folder can be expanded in the Analyses Treeview.



Note. The most onerous location of traffic loads has not been determined for this example which simply places tandems and vehicles at midspan. As stated previously, a moving load generator could be used to create many loadcases considering vehicles moving along a defined path, or the Vehicle Load Optimisation facility can determine worst-case patterns of loads automatically. Both of these facilities are, however, outside the scope of this example.

Save the model



Save the model file.

Running the Analysis

With the model loaded:



Open the Solve Now dialog and press **OK** to solve all three analyses.

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

If the analyses are successful...

Analysis loadcase results are added to the Treeview.

In addition, 2 files will be created in the directory where the model file resides:

If an analysis fails...

If an analysis fails, information relating to the nature of the error encountered will be written to the text output window. Any errors listed should be corrected in LUSAS Modeller before saving the model and re-running the analysis.

Rebuilding a Model

If it proves impossible for you to correct the errors, reported a file is provided to enable you to re-create the model from scratch and run an analysis successfully.



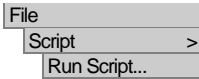
grillage_modelling.vbs carries out the modelling of the example.

Simple Grillage



Start a new model file. If an existing model is open, Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **grillage** and click **OK**




To recreate the model, select the file **grillage_modelling.vbs** located in the \<LUSAS Installation Folder>\Examples\Modeller directory.




Rerun the analysis to generate the results.

Viewing the Results

Analysis loadcase results are present in the  Treeview, and results for the last active loadcase will be set to be active by default.

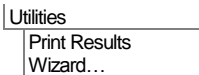
Checking the basic results

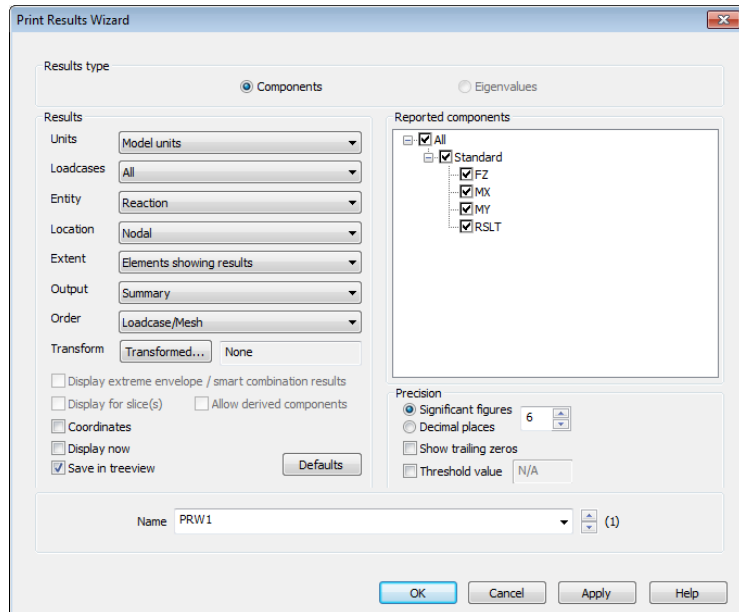
The print results wizard can be used to view summary results for checking purposes.

- In the  Treeview right-click on loadcase **Lane Load Upper** and select **Set Active**.

Reactions

- Set the Loadcases to be printed to **All**
- Set the Entity droplist to **Reaction** and the Output droplist to **Summary**.
- Leave all other settings and press **OK**.







On the results table, a tab for each loadcase gives Summary information.

Component	FZ[N]	MX[N.m]	MY[N.m]	RSLT[N]
1: Maximum	121.595	N/A	N/A	121.595
2: Node	34	N/A	N/A	34
3: Minimum	-5.3088	N/A	N/A	5.30883
4: Node	1	N/A	N/A	88
5: Total	540.0	N/A	N/A	N/A




The total FZ reaction for each loadcase is shown in the ‘Summary of key results’ table below.

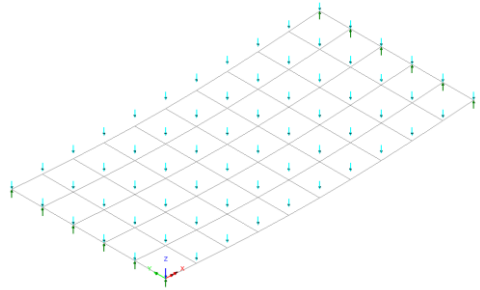
Deformed shape and Summary (deformations)

A deformed mesh plot helps highlight any obvious errors with an analysis before progressing to detailed results processing. The deformed shape will usually show up errors in loading or supports and may also indicate incorrect material property assignments (e.g. where the results show excessive displacements).

-  Press the **Isometric** button to orientate the model.
-  In the **Treeview**, turn off the **Geometry** and **Mesh** layers by right clicking on each in turn and selecting ‘On/Off’ from the context menu. The **Deformed mesh** layer should be on by default.



Simple Grillage

- In the  Treeview, turn on the Window Summary by checking (ticking) the box near the bottom of the panel. If necessary, the window summary itself can be clicked and dragged to a suitable 'blank' part of the screen.
- In the  Treeview right-click on the **Dead load** loadcase and select the **Set Active** option.
- It is good practice to step through each of the loadcases in the  Treeview using the **Set Active** option to check each deformed shape looks correct for the supposed loading.



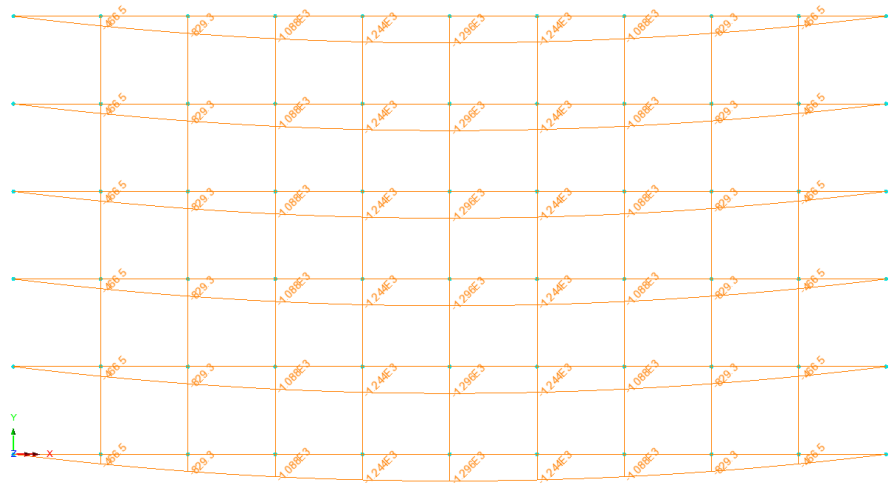
The maximum displacement from the Window Summary for each loadcase is shown in the 'Summary of key results table' below.

Bending moments

-  Press the **Home** button.
- For the **Dead load loadcase**, and with no features selected, click the right-hand mouse button in a blank part of the graphics window and select the **Diagrams** option to add the diagram layer to the  Treeview.

The diagram properties will be displayed.

- Select entity **Force/Moment** results of component bending moment **My**
- Select the **Diagram Display** tab.
- Specify the angle to be **45**
- Select the **Orientate flat to screen/page** radio button.
- Leave all other defaults and click the **OK** button to display the bending moment diagram initially for the currently active Dead load loadcase.



The maximum BM at midspan for each loadcase is shown in the ‘Summary of results table’ below.

Summary of key results table

Analysis	Loadcase	Reaction (kN)	Maximum deformation (mm)	Max BM (kNm)
1 (short term)	Lane load upper	540	1.943	-462
	Lane load lower	150	0.540	-128
	Tandem upper	600	3.310	-1001
	Tandem lower	400	2.206	-667
	LM3 upper	1200	6.414	-1675
2 (long term)	Surfacing	942	6.269	-502
3 (dead load)	Dead load	3110	49.227	-1296



Note. Maximum deformation values may also be obtained by adding a Values layer to the view window.

Validation of results

Some results can be validated by hand. For example a good first check would be to check reactions since applying an incorrect loading will inevitably lead to incorrect load effects.

Simple Grillage

Reaction F_z for Lane 1 (upper) = $UDL\ 9.0kN/m^2 \times 20m \times 3m = 540kN$.

Dead Load effects do not include transverse distribution and so can be validated by comparison to a simply supported beam. For example:

Dead load per beam = $3100/6=518.3kN$

Deformation = $5WL^2/384EI = 5 \times 518.3 \times 20^2 / (384 \times 12.0E6 \times 0.0928583) = 48.4mm$

Peak BM = $WL/8 = (518.3) \times 20/8 = 1296kNm$

Defining Design Combinations

The design combination will consist of dead load, surfacing, and traffic loading factored by the appropriate adverse or relieving factor.


For the characteristic combination, the effects from Group 1 traffic (LM1 in lanes 1 and 2) must be compared with Group 5 traffic (LM3 in lane 1 and LM1 in lane 2).

Defining a Basic Load Combination

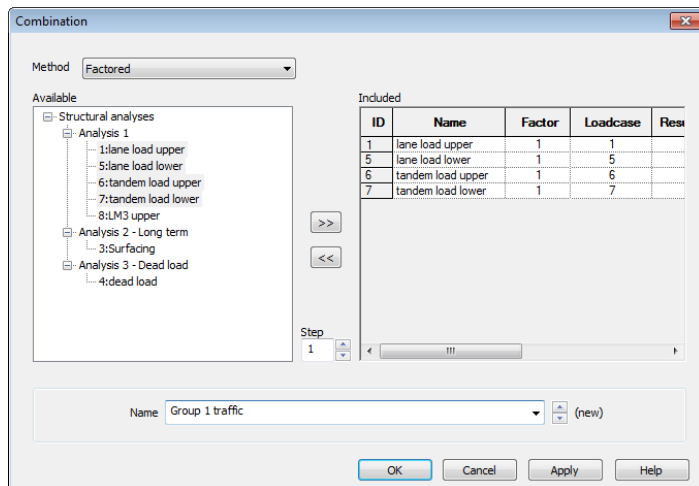
Group 1 traffic

A basic load combination will be used to combine LM1 lane loads and tandem loads from the two lanes.


On the Basic Combination dialog:

 Add loadcases (1) Lane load upper, (4) Lane load lower, (5) Tandem load upper, (6) Tandem load lower

Analyses
Basic
Combination...





Note. To add a number of loadcases all together select the first loadcase in the list, hold down the **Shift** key and select the last loadcase in the list (scrolling down the list if necessary) and click the  button.

Each loadcase selected may be given a factor, for example the psi-factors of EN1990 Table A2.1. In this case all the loads will have a factor of **1.0**

- Change the combination name to **Group 1 traffic**
- Click **OK** to save the combination definition.


Group 5 traffic

A basic load combination will be used to combine LM3 in lane 1 and LM1 in lane 2.

On the Basic Combination dialog:

Analyses

Basic
Combination...

 Add result loadcases (4) **Lane load lower**, (6) **Tandem load lower**, (7) **LM3 upper**

Again, in this case, all the loads will have a factor of **1.0**.

- Change the combination name to **Group 5 traffic**
- Click **OK** to save the combination definition.

Enveloping the Traffic Load Combinations

On the Properties dialog:

Analyses

Envelope

 Add the combinations **Group 1 traffic** and **Group 5 traffic** to the 'Included' panel.

- Change the envelope name to **Traffic Load Envelope**
- Click **OK** to save the envelope definition.



Note. When either a Max or Min smart combination or envelope is modified the corresponding Max and Min dataset will be updated automatically.

Defining a Smart Combination


Smart load combinations take account of adverse and relieving effects for the loadcase being considered. The Self-weight, Superimposed Dead Load, and the Traffic Load Envelope will all be combined using the Smart Load Combination facility to give the design combination.

Simple Grillage

Analyses
Smart Combination...

On the Smart Combination dialog:

 Add loadcase **Dead load** and **Superimposed DL** to the Included panel.

 Add **Traffic Load Envelope (Max)** and **Traffic Load Envelope (Min)** to the Included panel.




Each loadcase/envelope selected needs the Beneficial and Adverse factors to be specified – as shown in the table below, in accordance with EN1990 Table A2.4(C)

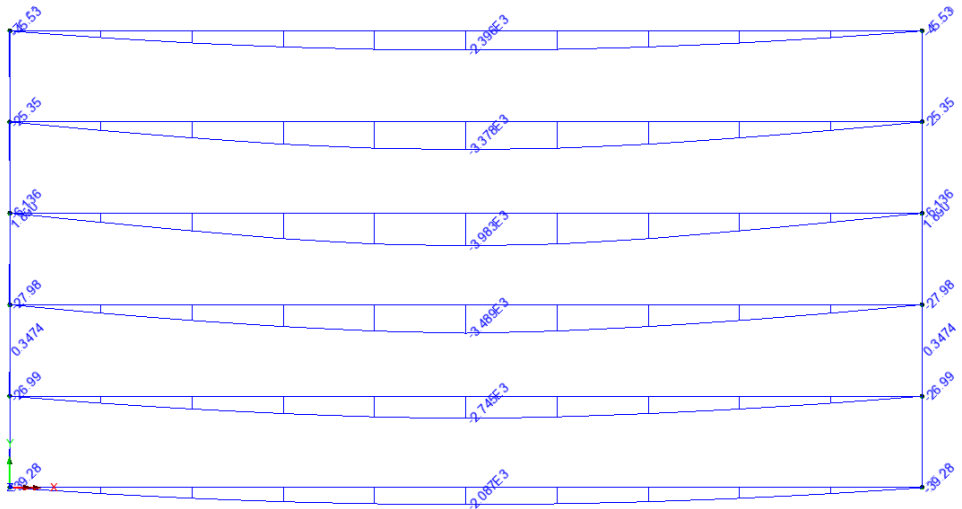
Loadcase name	Beneficial	Adverse
Dead Load	1.0	1.0
Surfacing	1.0	1.0
Traffic Load Envelope (Max)	0	1.15
Traffic Load Envelope (Min)	0	1.15

- Change the combination name to **Design Combination**
- Click **OK** to save the smart combination.

Bending Moment Diagram

A plot showing the bending moment from the design combination is to be displayed for the main members of the grillage.

- In the  Treeview select **Design Combination (Min)** and drag and drop it onto the graphical window (this is an alternative method to setting the loadcase active using the loadcase's context menu). Select **Force/Moment, My**.
- In the  Treeview right-click the **Transverse Beams** group and select **Results plots > Do not show results**.
- In the  Treeview, turn off the Window Summary by unchecking (unticking) the box near the bottom of the panel.



Note. When activating a smart combination the selected component is used to decide if the adverse or beneficial factor should be applied. Viewing results for a component other than the selected component will result in display of the associated values (coincident effects). When the results of an envelope or smart combination are displayed in tabular form, the results column used to compute the combination or envelope is denoted with an asterisk in the column header.

Page layout view



Note. Results plots which are to be printed are best created in the page layout view. This provides a view that will appear similar to the printed output. Labels however may be difficult to read in the page layout view since they reflect the size of the labels on the final printout. When this situation arises the zoom facility may be used to examine labels of interest more closely.

View
Page Layout Mode

- Switch to page layout view.

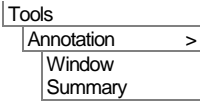
Tools
Annotation >
Window border

- Add a border to the page which contains the title, date and version of the LUSAS software in use.

File
Page Setup

- Ensure the orientation is set to **Landscape**. Change the page margins to enable the annotation to be added without obscuring the display. Set the left margin to **50**, the right margin to **15** and the top and bottom margins to **10**. Click **OK**

Simple Grillage



A summary of results will be added to the graphics window showing the loadcase name, diagram component, maximum and minimum diagram values, and element numbers in which the maximum and minimum moments occurs.

- Select the annotation by clicking over any piece of text and then drag it the summary text to an appropriate location on the plot.



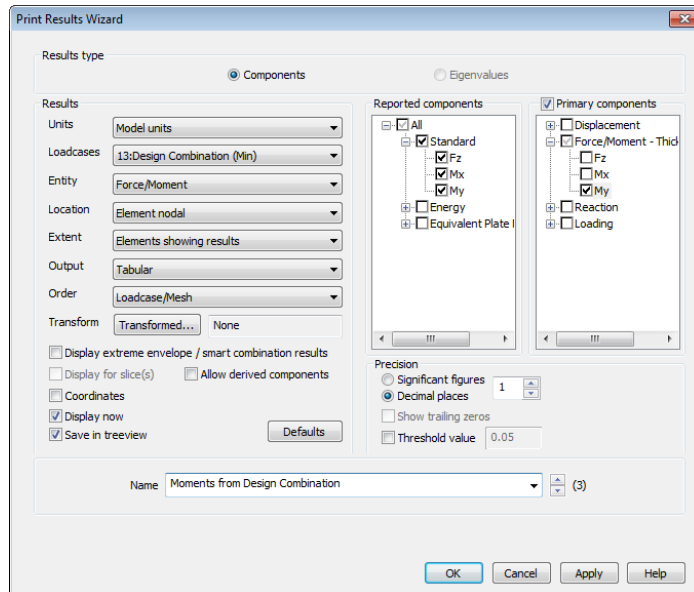
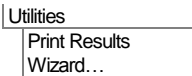
Note. The location of any model feature, element or node can be found by using the Advanced Selection facility. This can be used to find the location of the maximum and minimum results values since the element number is output in the window summary text.

As well as creating a results plot, results can be printed for saving or copying to a spreadsheet using standard Windows copy and paste.

Tabular results from the Print Results Wizard

Results from a selected loadcase, combination or envelope

Results values may be output to the screen in a tabular listing format for the active loadcase or for any selected loadcase.




- Ensure **Design Combination (Min)** is chosen from the Loadcases drop-down.
- Select Entity **Force/Moment**
- Select Location **Element Nodal**

- Select Output **Tabular**
- Ensure the **Primary components** box is checked, and expand the **Force/Moment - Thick Grillage** treeview entry and ensure that the primary results component **My** is checked.



Note. By checking (ticking) the Primary checkbox only the primary component(s) selected is/are enveloped and all other reported components selected will be coincident effects.

- In the Precision panel set the number of decimal places to **1**
- Ensure that the **Display now** and **Save in Treeview** boxes are both checked
- Leave all other Results panel settings as their default values.
- Enter the Name '**Moments from Design Combination**' and click the **OK** button

The results are displayed. A Print Results Wizard entry is also added to the Utilities  Treeview, allowing the same results to be re-assembled without using the dialog next time.

Element	Node	Fz[kN]	Mx[kN.m]	My*[kN.m]	
1	1	-385.5	39.3	-27.9	
2	1	-335.3	37.8	-748.6	
3	2	-311.4	35.2	-754.4	
4	2	-259.6	35.2	-1325.4	
5	3	-232.9	28.9	-1330.6	
6	3	-181.0	28.9	-1744.4	
7	4	-151.6	19.2	-1748.4	
8	4	-99.8	19.2	-1999.8	
9	5	-68.2	6.8	-2002.1	
10	5	-16.4	6.8	-2086.6	
11	6	16.4	-6.8	-2086.6	
12	6	68.2	-6.8	-2002.1	
13	7	99.8	-19.2	-1999.8	
14	7	151.6	-19.2	-1748.4	
15	8	181.0	-28.9	-1744.4	
16	8	232.9	-28.9	-1330.6	
17	9	259.6	-35.2	-1325.4	
18	9	311.4	-35.2	-754.4	
19	10	335.3	-37.8	-748.6	
20	10	385.5	-39.3	-27.9	
21	11	7.4	6.0	-6.5	
22	11	-34.1	-27.9	-39.3	
23	21	11	34.1	27.9	-39.3
24	21	22	-7.4	-6.0	-6.5
25	22	12	-457.6	55.9	-7.1
26	22	13	-405.8	55.9	-870.5
27	23	13	-381.6	52.0	-872.0
28	23	14	-329.8	52.0	-1583.4
29	24	14	-304.8	42.5	-1584.7
30	24	15	-253.0	42.5	-2142.5

In the table, when the active loadcase is an envelope or smart combination, the results printed will show the primary component (**My** in this case) marked with an asterisk. Hovering over the contents of a cell will display a datatip showing location information and an associated value.



Note. Print Results Wizard data can be added to a Model Report by pressing the **Add to Report** button at the top of the results listing. When done, each time the model

report is generated the results included will be for the current state of the model at that time.

Saving tabular results to a spreadsheet

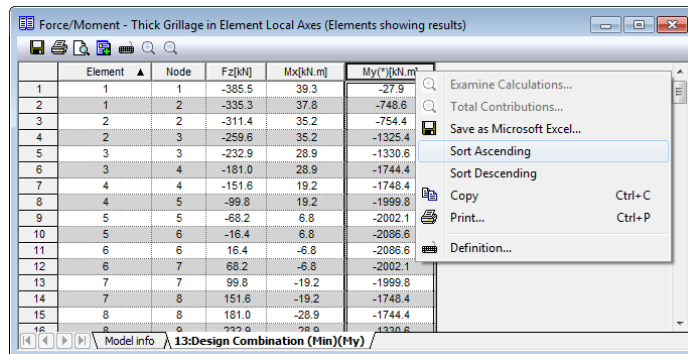
When the Print Results window is shown a context menu can be displayed allowing the printed results to have their number of significant figures or decimal places changed, be sorted in ascending or descending order, be saved to a spreadsheet or copied for pasting elsewhere.

- Right-click inside the Printed Results window and select **Save as Microsoft Excel...**
- Enter a file name of **grillage_results**
- Ensure the save option **All tabs** is selected and click **Save**.

Note that Microsoft Excel may impose limitations on the length of tab name permitted.

Examining calculations in combination or envelope results

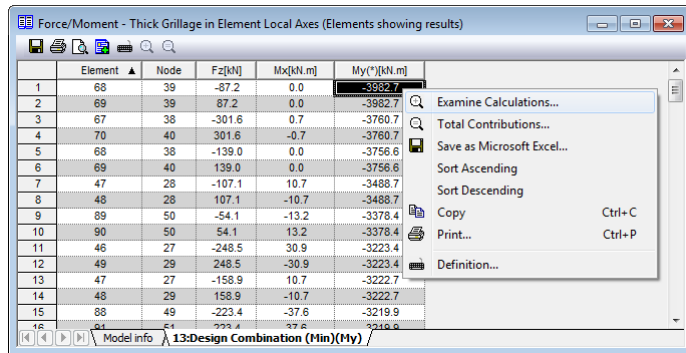
- Right-click on the **My(*)[kN.m]** column and choose **Sort ascending** from the context menu. This brings the most negative (the largest sagging moment) to the top of the table.



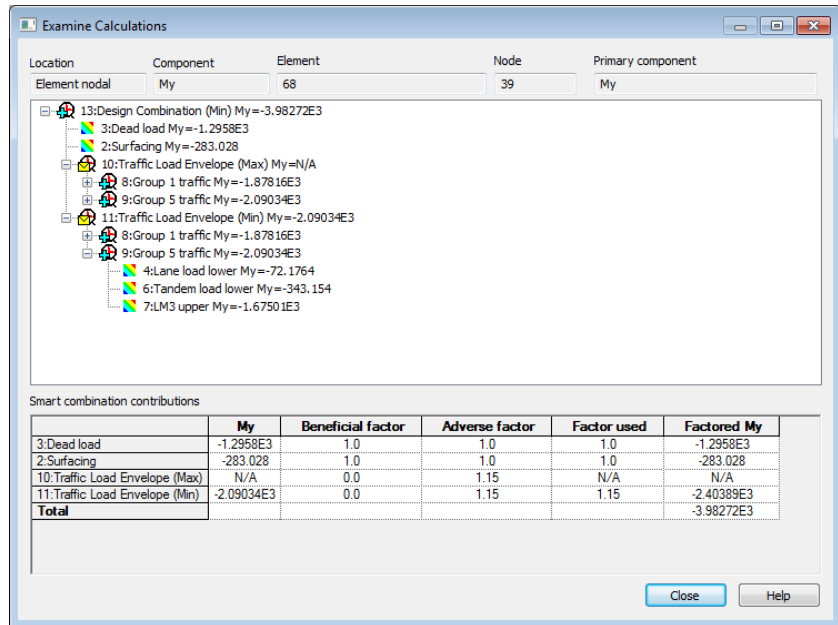
The screenshot shows a software window titled "Force/Moment - Thick Grillage in Element Local Axes (Elements showing results)". It contains a table with columns for Element, Node, Fz[kN], Mx[kN.m], and My(*)[kN.m]. A context menu is open over the My(*) column, with options like "Examine Calculations...", "Save as Microsoft Excel...", "Sort Ascending", and "Sort Descending".

Element	Node	Fz[kN]	Mx[kN.m]	My(*)[kN.m]
1	1	-385.5	39.3	-27.9
2	1	-335.3	37.8	-748.6
3	2	-311.4	35.2	-754.4
4	2	-259.6	35.2	-1325.4
5	3	-232.9	28.9	-1330.6
6	3	-181.0	28.9	-1744.4
7	4	-151.6	19.2	-1748.4
8	4	-99.8	19.2	-1999.8
9	5	-68.2	6.8	-2002.1
10	5	-16.4	6.8	-2086.6
11	6	16.4	-6.8	-2086.6
12	6	68.2	-6.8	-2002.1
13	7	99.8	-19.2	-1999.8
14	7	151.6	-19.2	-1748.4
15	8	181.0	-28.9	-1744.4

- Right-click on the row 1 value of **My=-3981E3** and select **Examine calculations**.



The Examine Calculations dialog appears.



The grid at the bottom of the dialog shows the unfactored My contributions, the factors used, the factored My effects and the total.

By expanding loadcases and entries of interest in the treeview, loadcases, components and values contributing to loadcases and components higher up the treeview can be identified.

- With the tree expanded for '11: Traffic Load Envelope (Min)' the Group 1 traffic effect can be seen to be **-1878kNm**, whereas the Group 5 traffic effect is **-2090kNm**, which was therefore selected from the envelope, since it was more onerous.

- Expanding the tree further the Group 5 traffic effect can be seen to be made up of contributions from Lane Load Lower, Tandem Load Lower and LM3 Upper



Note. Selection of the ‘Total contributions’ option (instead of the Examine calculations option) for a selected results value in a printed results table, reduces the contributing envelopes and combinations to a single table of loadcases and factors in use. For a smart combination, a loadcase can be created, which includes all the loading assignments and factors which gave a particular result (at a particular node).

Save the model



Save the model file.



Note. If the model file is saved after results processing, all load combinations, envelopes, and graph datasets, if defined, are also saved and therefore do not have to be re-created if the model is amended and a re-analysis is done at a later date.

This completes the example.

File _____
Save _____