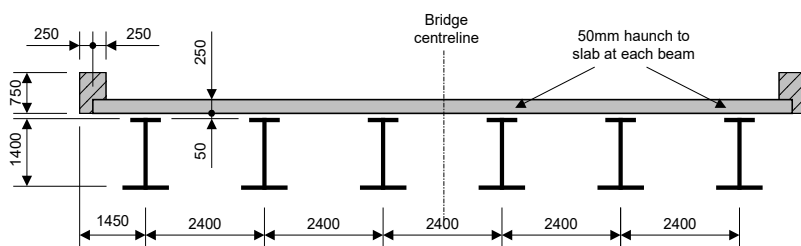


Grillage Load Optimisation

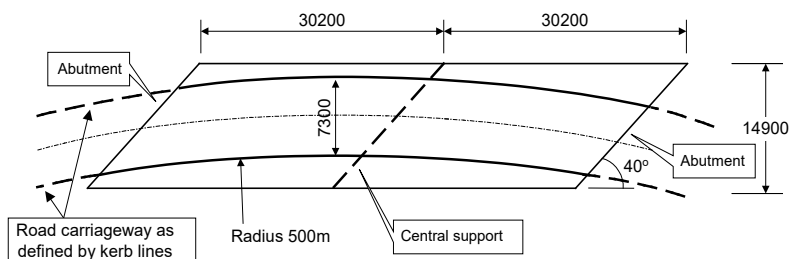
For LUSAS version:	22.0
For software product(s):	LUSAS <i>Bridge</i> and LUSAS <i>Bridge Plus</i>
With product option(s):	Vehicle Load Optimisation
Note: The example exceeds the limits of the LUSAS Teaching and Training Version.	

Description

This example illustrates how the vehicle load optimisation facility for highway traffic is applied to grillage models. An example Code of Practice is used, together with the reciprocal method for generating suitable influence surfaces.



Cross Section



Carriageway Definition

The structure consists of steel plate girders supporting an in-situ reinforced concrete deck slab. Two spans of 30.2m each (skew span measurement) span between the centreline of the support bearings.

In such a structure, parapet beams may be cast after the deck slab has cured. While this would notionally make the edge beam properties different from the internal beams, the parapet beams will be treated as non-structural since:

- Practically they are usually made discontinuous to avoid unsightly cracking under service conditions.
- Including them in the stiffness of the structure would mean that any damage to those parapet beams arising from – for example – vehicular impact, would invalidate the analysis assumptions, potentially leading to overstress in main members.

This assumption makes the internal and edge beam dimensions the same, with a top slab width of 2.4m each.



The structure will be analysed for traffic loads to EN1991-2 using the UK National Annex and Special Vehicle SV196 in Load Model 3. The method used is, however, relevant for other traffic loading Codes Of Practice. No other loading is considered.

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

Objectives

The required output from the analysis consists of:

- ☐ Characteristic (ULS) and Frequent (SLS) values for selected locations on the structure.

Keywords

2D, Grillage, Wizard, Vehicle Load optimisation, Load visualisation, Influence surface, Basic Load Combination, Smart Load Combination, Envelope.

Associated Files

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



- ☐ **grillage_load_modelling.lvb** carries out the modelling of the example up to the point where vehicle loading is applied.

Vehicle load optimisation in LUSAS

Vehicle load optimisation (VLO) uses influence surfaces to identify the most onerous highway traffic loading patterns according to a chosen Code of Practice and to apply these loading patterns to LUSAS models.

In this example, the grillage model is loaded according to Eurocode EN1992-1 and the UK National Annex. The example could be modified or re-run for a different Code of Practice, National Annex or other traffic loading option as required.

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.

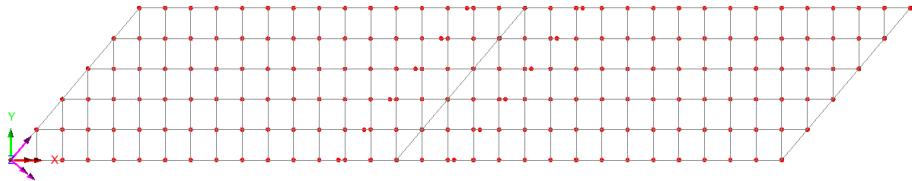
Creating a new model

- Enter a File name of **grillage_load_optimisation**
- 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 **Two span grillage analysis with load optimisation**
- Click the **OK** button.

Feature Geometry

The grillage geometry will be created using the Grillage Wizard.

- Select the **Set Default** button.
- Select **Spaced beam and slab deck** and ensure the **cracked section** is specified to be **15%**. Click **Next**
- The grillage is **Straight** with **40** degrees skew. Click **Next**.
- Select an **Orthogonal** grillage arrangement and click **Next**
- Ensure **Evenly spaced** beams are being defined and enter the width of grillage as **12** and the number of longitudinal beams (including edge beams) as **6**. Click **Next**
- Change the number of spans to **2**
- For each span enter the length of span as **30.2** and change the number of internal transverse beams to **9**
- Click **Finish** to generate the grillage model.



Geometric Attributes required

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

- ☐ **Two ‘girder with top slab’ grillage attributes**, with each incorporating a custom plate girder section.
- ☐ **One ‘slab’ grillage attribute**.
- ☐ **One rectangular solid (concrete) section**, to represent the internal and end skew diaphragm beams.

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, considering 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 the plate girder sections

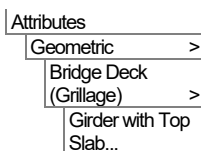
Bridge Deck (Grillage) attributes are used to model the plate girder sections. Two girder section types are required, both with overall depth of 1.4m, but with differing plate sizes. Whilst only two Bridge Deck (Grillage) geometric section attributes are created for this example, many more may be required to model the changing sections in a larger, more complex bridge deck analysis.



Note. Model units of metres are to be input. Other units may be converted during entry by using the drop-down unit converter in each numerical input cell.

Defining a span girder section

The span girder and top slab is defined by selecting the tabs on the dialog.



- Change the Bridge deck type to **Steel I-girder**
- With the **Girder** tab selected, define a **Constant section**
- Click on the  image in the Section cell to allow a section to be chosen from the library.
- On the resulting 'Enter Section' dialog, and from the droplists on the right, select **Parametric Sections**, then **I sections**, then **New...**
- On the resulting I section dialog select **I (Unequal flange)**
- Enter the values shown in the following image, then click in the radius (r) cell to cause the section properties to be calculated but don't enter a value for this cell.

I Sections

Type: I (Unequal Flange)

Dimensional data

☒ D: 1.4
☐ d: 1.35
Bt: 0.4
Bb: 0.65
tft: 0.02
tfb: 0.03
tw: 0.012
r: [dropdown]

Calculated properties

A: 0.0437
Ixx: 0.0140445
Iyy: 0.793424E-3
Ixy: 0.0
J: 7.54643E-6
Asy: 0.0160889
Asx: 0.020549

Section details...

Name: Sct1 (new)

OK Cancel Apply Help

The attribute name of Sct1 is generated automatically.

- Click **OK** to return to the Geometric line dialog (this creates a utility of the name stated in the  Treeview, then click **OK** to return to the 'Girder with Top Slab' dialog.
- Next, select the **Slab** tab.

Bridge Deck (Grillage) Geometric Attribute - Girder with Top Slab

Analysis category: 2D Grillage/Plate

Girder Slab Girder & Slab

☒ Constant dimensions ☐ Dimensions vary linearly between stated distances

☐ Specify effective widths

	b	t	ys	bh	th	Effective width	Distance
1	2.4	0.25	0.0	0.5	0.05	2.4	na

☒ Include haunch in section properties

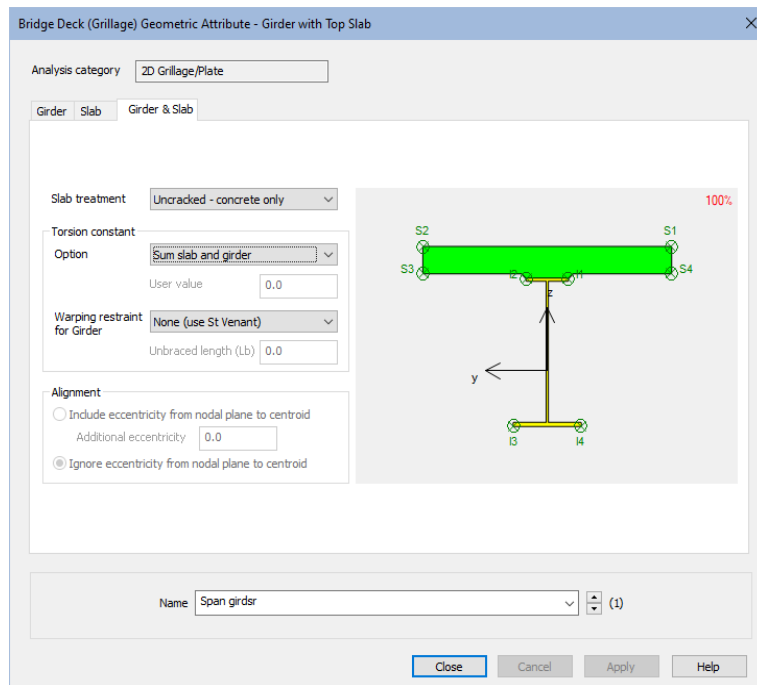
Reinforcement

	Bar size (Ø)	Spacing (s)	Cover (c)
Top	0.0	0.0	0.0
Bottom	0.0	0.0	0.0

Name: Span girder (1)

Close Cancel Apply Help

- 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 width (b) of **2.4** and a thickness (t) of **0.25**
- The slab 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**
- Enter the width of the haunch (thickening) of the concrete slab above the top flange (bh) to be **0.5** and the height of the haunch (th) to be **0.05**
- Ensure that the **Include haunch in section properties** checkbox is checked (ticked) to include the haunch properties in the section property calculation.
- Lastly, select the '**Girder & Slab**' tab.



- For the slab treatment ensure that **Uncracked (concrete only)** is selected. (The alternative ‘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’; ‘None – 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 **Span girder** and press **Apply**. This creates an attribute in the Treeview and retains the existing data allowing it to be modified to create the next attribute.

Defining a pier girder section

- On the current **Girder & Slab** tab, type a new name of **Pier girder**
- Switch to the **Girder** tab and define a **Constant section**
- Click on the  image in the Section cell to allow a section to be chosen from the library.

- On the resulting 'Enter Section' dialog, and from the droplists on the right, select **Parametric Sections**, then **I sections**, then **New...**
- On the resulting I sections dialog select **I (Unequal flange)**
- Enter the values shown in the following image, then click in the radius (r) cell to cause the section properties to be calculated but don't enter a value for this cell.

The screenshot shows the 'I Sections' dialog box with the following data:

Dimensional data		Calculated properties	
Type	I (Unequal Flange)	A	0.06961
☒ D	1.4	Ixx	0.024094
☐ d	1.335	Iyy	2.27926E-3
Bt	0.65	Ixy	0.0
Bb	0.8	J	21.8191E-6
tft	0.025	Asy	0.0213179
tfb	0.040	Asx	0.0390594
tw	0.016		
r	0.0		

Section details...
Name: Sct2 (new)
Buttons: OK, Cancel, Apply, Help

The attribute name of Sct2 is generated automatically.

- Click **OK** to return to the Enter Section dialog (this creates a utility in the Treeview), then click **OK** to return to the 'Girder with Top Slab' dialog.
- Next, select the **Slab** tab.
- Change the width (bh) of the haunch (thickening) to be **0.75** to account for the wider top flange.
- In the reinforcement panel set the top and bottom bar diameters to be **0.012**, the spacings to be **0.15** and the top and bottom cover values to be **0.04** and **0.05** respectively as shown below.

The reinforcement details are used when calculating cracked section properties (the slab being in tension for 15% either side of the interior support).

Girder Slab **Girder & Slab**

☒ Constant dimensions ☐ Dimensions vary linearly between stated distances

☐ Specify effective widths

	b	t	ys	bh	th	Effective width	Distance
1	2.4	0.25	0.0	0.75	0.05	2.4	na

☒ Include haunch in section properties

Reinforcement

	Bar size (Ø)	Spacing (s)	Cover (c)
Top	0.012	0.15	0.04
Bottom	0.012	0.15	0.05

Insert Delete Flip

- Lastly, select the **Girder & Slab** tab.
- Specify the slab to be **Cracked in tension**

Girder Slab **Girder & Slab**

Slab treatment: Cracked in tension

Torsion constant Option: Sum slab and girder

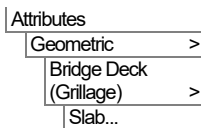
User value: 0.0

100%

- Finally, click the **OK** button to create a Pier girder attribute in the Treeview.

Defining a 'slab' grillage attribute

'Slab' grillage attributes are required to represent the transverse slab members.



- Select a **Solid** slab type and enter a width (b) of **2.013** and 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**.

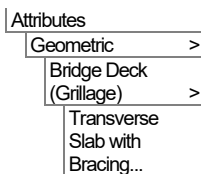


Note. Cracked properties are calculated according to ACI318 clause 10.10.4.1 (b) and (c)), which essentially takes the cracked bending stiffness to be 25% of the gross uncracked stiffness. Alternatively, if the reinforcement dimensions are known, the 'From reinforcement' option may instead be used, and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

Defining a 'slab with bracing' grillage attribute

'Slab with bracing' grillage attributes are required to represent those transverse slab members with bracing between the girders.

In this model the bracing will be provided between pairs of beams.



- Select a **Torsional** bracing arrangement.
- Enter a Slab width (b) of **2.013** and thickness (t) of **0.25**
- Ensure the slab is treated as **Cracked in flexure** with the Cracked properties calculated according to **Approximate (ACI318)**.
- Enter a distance (h1) of **0.2** and a distance (h2) of **0.95**
- Click in the Bracing sections grid for row 1, then click the grid down arrow . On the resulting 'Enter Section' dialog, and from the droplists on the right, select **UK Sections**, then **Equal Angles (Advance)**, then **150X150X12 UKEA**
- Repeat for row 2.
- Girder dimensions can be determined by clicking the **From library** button and on the resulting 'Enter Section' dialog, and from the droplists on the right, select

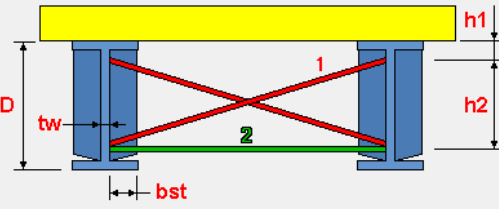
Parametric Sections, then **I sections**, then **Sct1**. Values of D and t_w are then automatically populated.

- Stiffener details are defined for **Both side of web** with an outstand of **0.3** and a thickness of **0.03**
- On completion the dialog should look this:

Bridge Deck Geometric Attribute - Transverse Slab with Bracing

Analysis category: 2D Grillage/Plate

Bracing arrangement: Torsional



☒ Crossing members connected

h1: 0.2
h2: 0.95

Girder dimensions: From library...
D: 1.4
tw: 0.012

Bracing sections	
1	150x150x12 UKEA
2	150x150x12 UKEA

Stiffener details
Position: Both sides of web
Outstand (bst): 0.3
Thickness (tst): 0.03

Slab
b: 2.013 t: 0.25

Slab treatment: Cracked in flexure
Cracked properties: Approximate (ACI318)

Reinforcement details (top & bottom)
Bar size: 0.0
Spacing: 0.0
Cover: 0.0

☐ Specify torsional constant
Torsional constant (J): 0.0

Visualise...

Name: Transverse slab with bracing (new)

OK Cancel Apply Help

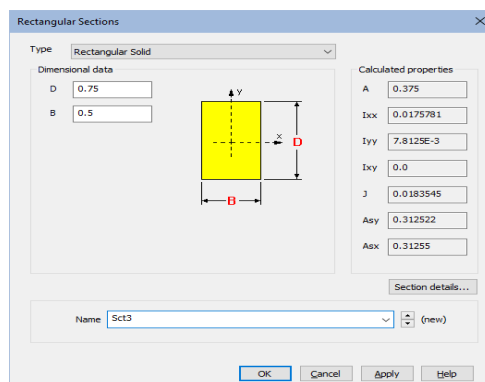
- Name the attribute as **Transverse slab with bracing** and click **OK**.

Defining the internal and end skew diaphragm beams

The internal and end skew diaphragm beams will be represented using a rectangular solid section:

Attributes
Geometric >
Section calculators

- On the 'Enter section' dialog select **Parametric Sections**, then **Rectangular**, then **New...**
- For a rectangular solid section enter a depth (D) of **0.75** and a width (B) of **0.5**
- Accept the default attribute name of **Sct3** that is created and click **OK** to return to the Geometric line dialog (this creates a utility of the name stated in the  Treeview).



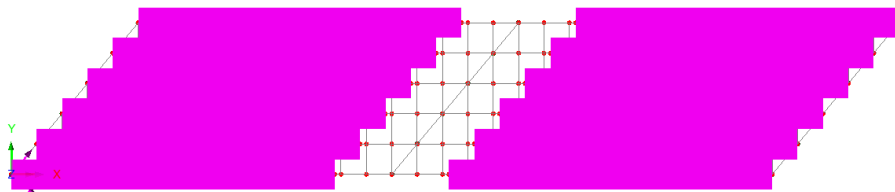
- On the Geometric line dialog name the attribute to be **Diaphragms** and click **OK** to add the section to the  treeview.

Assigning the various geometric properties

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

Span girder (Uncracked) properties

- In the  Treeview, select the '**Span girder**' attribute and click on the Copy  button.
- In the  Treeview, select the **Longitudinal Uncracked Section** group and click on the Paste  button to assign the span girder properties to those members.
- An assignment dialog clarifies that this assignment is to lines in Analysis 1.



Pier girder (Cracked) properties

- In the  Treeview, select the '**Pier girder**' attribute and click on the Copy  button.

- In the  Treeview, select the **Longitudinal Cracked Section** group and click on the Paste  button.

Transverse slab properties

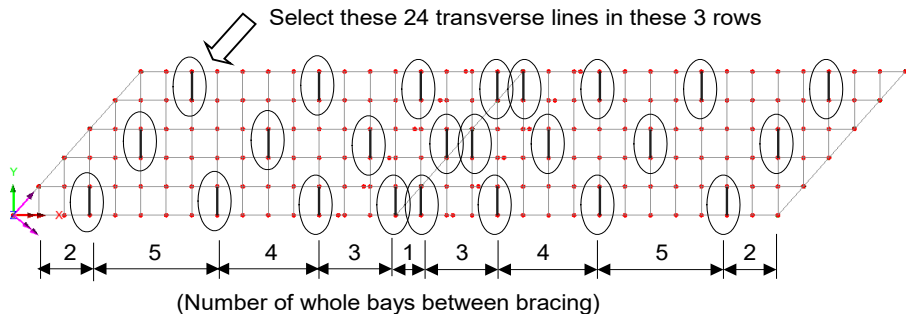
- In the  Treeview, select the '**Transverse slab**' attribute and click on the Copy  button.
- In the  Treeview, select the **Transverse Skew Beams** and click on the Paste  button.
- Repeat to paste the Transverse slab attribute onto the **Transverse Intermediate Beams** and **Transverse Central Beams** entries.



Note. The **Transverse slab with bracing** attribute is to be assigned to some of the Transverse members above. However, the easiest way to do this is to assign the most common attribute to all transverse members first (the transverse slab in this case) and then individually select those members to which other assignments will override the initial assignment made.

Define the transverse slab bracing members

- Turn off the fleshing using the  toolbar button
- Select the 24 members shown in the following image.



-  Press the group button to create a group for these items and name the group **Bracing members**
- In the  Treeview, select the **Transverse slab with bracing** attribute and click on the Copy  button.

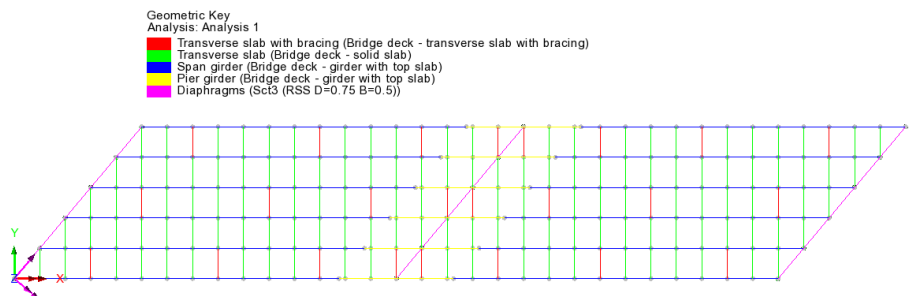
- In the  Treeview, select the **Bracing members** and click on the Paste  button.

Diaphragm properties

- In the  Treeview, select the **'Diaphragms'** attribute and click on the Copy  button.
- In the  Treeview, select the **Internal Diaphragms**, hold down the **Ctrl** key and also select the **End Diaphragms** groups and click on the Paste  button.

Checking for correct geometric assignment

- In the  Treeview, right click on the **Mesh** layer and select **Display** to turn off the meshing display.
- In the  Treeview, double click on the **Geometry** layer and from the **Colour by** drop down list, select **Assignment**.
- Click the **Set** button and select **Geometric** from the Attribute Type drop down list.
- Ensure the **Generate key** option is selected and 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, in the  Treeview double click on **Geometry**, and from the Colour by drop down list select **Own Colour** and click **OK** to update the display.

Defining the Materials

Bridge Deck (Grillage) members

The materials needed for the Bridge Deck (Grillage) geometric attributes are defined using a corresponding “Bridge Deck (Grillage)” material attribute, which references slab, girder and rebar materials in one attribute.

On the dialog make the following selections:

Slab

- Click in the material cell and choose **New...**
- On the material library dialog select material type **Concrete**, region **Europe**, standard **EN1992-2:2005**, and grade **C40/50**
- Leave the material name as the supplied **Iso1** and click **OK**.
- Back on the Bridge Deck (Grillage) Material dialog ensure that the **Stiffness** is included by checking (ticking) that box, but the Mass is excluded.

Girder

- Click in the material cell and choose **New...** and in a similar fashion to defining the concrete material, make the following selections: **Steel-Structural**, **Europe**, **EN1993-1-1:2005**, and **Ungraded**
- Leave the material name as the supplied **Iso2** and click **OK**.
- Back on the Bridge Deck (Grillage) Material dialog ensure that the **Stiffness** is included by checking (ticking) that box, but the Mass is excluded.

Rebar

- Click in the material cell and choose **New...** and in a similar fashion make the following selections: **Steel-Reinforcing bar**, **Europe**, **EN1992-2:2005**, and grade **500B**
- Name the material **Rebar** and click **OK**.

Attributes	
Material	>
Bridge Deck (Grillage)...	

Bridge Deck (Grillage) Material

Sections

Ref	Material	Stiffness	Mass
Slab	1:Iso1 (C40/50 Concrete EN1992-2:2005)	☒	☐
Girder / Bracing	2:Iso2 (Ungraded Steel - Structural EN1993-1-1:2005)	☒	☐

Reinforcement (for cracked sections and design)

Ref	Material
Rebar	3:Iso3 (500B Steel - Reinforcing bar EN1992-2:2005)

Name: (new)

OK Cancel Apply Help

- Enter a bridge deck grillage material name of **Short term properties** and click **OK** to add the material dataset to the Treeview.

Three material entries now appear in the Treeview, one for each slab, girder and rebar isotropic material defined, together with one compound material entry (Short term properties) that references those used. Only the compound material attribute should be assigned to appropriate lines in the model.



Note. A grillage model with slab stiffness included should not be used to distribute self-weight, since this is carried by girders alone, while the concrete slab is wet. Hence, a second analysis using a ‘clone’ attribute with mass check boxes ticked and the stiffness of the slab unchecked would be suitable for a dead load analysis.



Note. As well as a material to suit the dead load analysis, a third material and analysis, where a long-term concrete material (typically having approximately one third of the elastic modulus of the short-term modulus) is included may be used to assess the distribution of long-term loads such as surfacing, deck furniture, and support settlements.

In this example, the “short term properties” are all that are required.

- With the whole model selected (press the **Ctrl** and **A** keys) drag and drop the material dataset **Short term properties** from the Treeview onto the selected features. Click **OK** on the dialog that appears.

Diaphragms

The diaphragms at the supports do not have Bridge Deck (Grillage) geometric attributes assigned, so cannot have Bridge Deck (Grillage) Material properties assigned. For these members, the isotropic 'Concrete' material that is present in the  Treeview must be assigned instead. To do this:

- In the  Treeview right-click on **Concrete C40/50 | Concrete EN1992-2:2005** and select **Copy**.
- In the  Treeview treeview right-click on **End Diaphragms** and select **Paste**.
- In the  Treeview treeview right-click on **Internal Diaphragms** and select **Paste**.

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.

Defining Influence Attributes

In order to use the vehicle load optimisation facility the locations at which the optimised load combinations are to be calculated and the parameter to be computed must be chosen. 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. An influence surface is generated for each of the chosen locations and parameters.

For this example, three influence surfaces will be defined using Reciprocal Method influence attributes.

Influence Point 1

The first influence attribute will be used to investigate the reaction at the left hand abutment.

- Select a **Reaction** influence type for a **Vertical axis** influence direction for a **Positive** displacement direction. Enter the influence attribute name as **Reaction Support 1** and click **OK**.

The influence attribute will be added to the  Treeview.

Attributes	
Influence	>
Reciprocal	
Theorem...	

Influence Point 2

The second influence attribute will be used to investigate mid span bending in the first span.

Attributes	
Influence	>
Reciprocal Theorem...	

- Select a **Moment** influence type **About Transverse** axis direction for a **Negative** displacement direction (because we are interested in the maximum sagging moment). Enter the influence attribute name as **Bending Span 1** and click **OK**.

Influence Point 3

The third influence surface attribute will be used to investigate the hogging moments at the central pier.

Attributes	
Influence	>
Reciprocal Theorem...	

- Select a **Moment** influence type **About Transverse** axis direction for a **Positive** displacement direction (because we are interested in the maximum hogging moment). Enter the influence attribute name as **Bending Internal Support** and click **OK**.



Note. Longitudinal and Transverse directions are specified using the  **Direction definition** object in the  Treeview. In this example the longitudinal direction is defined to be the Global X direction, and the transverse direction is always assumed to be orthogonal to the longitudinal direction.

Turning loading visualisation on and off



At any time during this example any applied loading can, for reasons of clarity, be turned on and off by selecting the Loading on /off button.

Assigning Influence Attributes

Influence attributes can be assigned to nodes or points. When modelling grillages created by the grillage wizard there will always be a node at a point on the model, so influence attributes could be assigned to points. But, in general usage, and for slab analysis especially, influence attributes are better assigned to nodes.

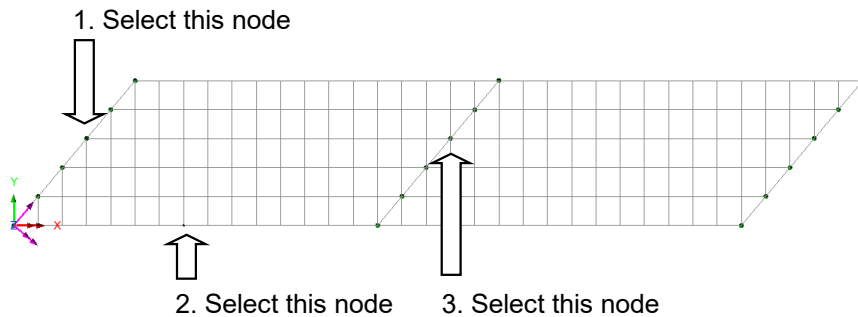


Caution. Reciprocal influence attributes can be assigned to as many nodes or points on the model as necessary but should only be assigned to those nodes or points for which they are applicable. For example, the Bending span 1 influence attribute (defined for a negative displacement direction) should only be assigned to those nodes in the first span of the bridge for which a negative influence shape is of interest. For nodes where a positive influence shape is of interest, a Reciprocal influence attribute with a positive displacement direction should be assigned. In general, when using Reciprocal influence attributes, the influence shapes for nodes of interest should be viewed after a solution is obtained to ensure that they have been defined correctly for the influence shape seen. Note that Direct Method influence attributes can be assigned to the whole

model regardless of whether nodes lie in a positive or negative region of an influence shape.

For this example, and for speed and clarity reasons, only one node will be selected for each influence attribute.

- Click in a blank area of the view window to deselect any previously selected items.
- In the  Treeview turn off the display of the **Geometry** layer.
- In the  Treeview turn on the display of the **Mesh** layer,



The first influence attribute will investigate the reaction near the centre of the left hand abutment.

- First, select only the node on the end diaphragm as shown on the preceding image.
- Drag and drop the influence attribute **Reaction Support 1** from the  Treeview onto the selected feature.



Note. Where a node coincides with its parent geometry the geometry name (in this case a Point) will appear as part of the influence assignment name in the  Treeview.

The second influence attribute will investigate edge mid-span bending in span 1.

- Now select only the mid-span node at the edge of span 1 as shown on the preceding image.
- Drag and drop the influence attribute **Bending Span 1** from the  Treeview onto the selected feature.

The third influence attribute will investigate hogging near the centre of the internal support.

- Lastly, select only the node at the internal support as shown on the preceding image.

- Drag and drop the influence attribute **Bending Internal Support** from the  Treeview onto the selected feature.

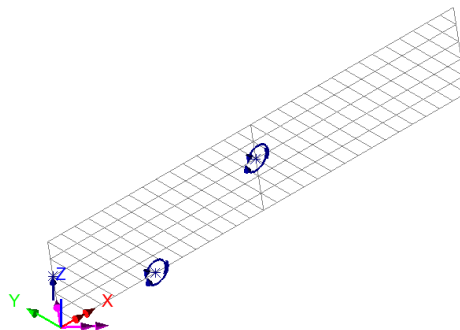
Visualising the Defined Influence Points

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.



Preparing to use the Vehicle Load Optimisation facility

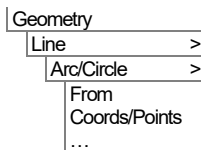
Before invoking the Vehicle Load Optimisation Wizard the kerb lines need to be defined. This can be done by specifying coordinates for the kerbs via the Carriageway Definition dialog of the Wizard but is best done by defining kerb lines on the model.



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

- In the  Treeview turn on the display of the **Geometry** layer.

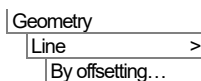
Defining Kerb Lines



- In the first row, specify **(-5, 1)**. For the second row select the **Direction** option and enter **(35, 20)** for that row. For the third row specify an end point of **(75, 1)**.
- Enter that a **Minor** arc type is selected and enter a radius of **500**
- Click **OK** to define the first kerb line.

The other kerb line will be drawn parallel to the first:

- Select the line just drawn.
- Specify an offset distance (carriageway width) of **7.3** and ensure that an offset direction of **Away from reference position** is selected.
- Click the **OK** button to define the second kerb line.





Note. Lines can be offset **towards** or **away from** either a previously selected point, or from a specified reference coordinate, or be created on **both sides** of a selected line or set of lines. See the online help for this dialog for more information.

Save the model



Save the model file.

File
Save



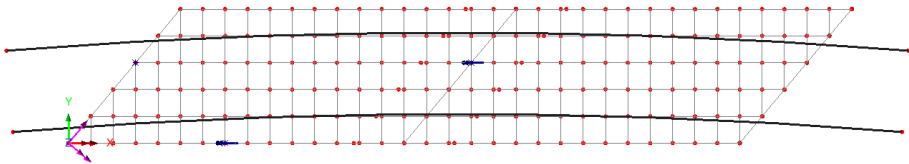
Rebuilding a model after a previous failed analysis

If a previous analysis of this example has failed, return to this point to continue after having run the supplied file stated.

Carrying out Vehicle Load Optimisation

The vehicle load optimiser automates the creation of load datasets in accordance with the chosen loading code for the locations and effects specified. Prior to defining vehicle loading the two lines representing the kerbs at the extent of the carriageway both need to be selected.

- Select both kerb lines (use Shift to add to the initial selection).



- Select **Europe** from the Country drop down list.
- Select **EN1991-2 UK 2009** from the Design code drop-down lists.
- Ensure the **View onerous effects table** is not selected.
- Ensure the **Create loading patterns** option is selected.
- **VLO Analysis 2** will be automatically entered for the Analysis entry in the Treeview.

Bridge
Vehicle Load
Optimisation
VLO Run

The screenshot shows the 'Vehicle Load Optimisation' dialog box. It has several sections: 'Loading options' with 'Country' set to 'Europe' and 'Design code' set to 'EN1991-2 UK 2009'; 'Solution process' with 'View onerous effects table' unchecked, 'Create loading patterns' checked, and 'Create envelopes' unchecked; and 'Analysis' with 'VLO Analysis 2' selected. There are also buttons for 'Optional code settings...', 'Optional loading parameters...', 'Set influence surfaces...', and 'Define carriageways...'. At the bottom, there are 'OK', 'Cancel', and 'Help' buttons.

(Note that an alternative name can be entered by selecting the New option from the drop-down list).

- Enter the VLO run Name as **VLO Run 1**
- Select the Optional code settings button.

The EN1991-2 UK 2009 optional code settings will appear.

- Deselect **Combination (psi0)**
- For Group 5 – LM3, select the  button to deselect the **SV80** vehicle and select the **SV196** vehicle instead.
- Click **OK** to set the selected options and return to the main vehicle load optimisation dialog.
- Select the **Optional loading parameters** button to display the optional loading parameters dialog.
- Since **SV196** is unsymmetrical, ensure that vehicle direction is set to **Both** and click **OK** to set the options and return to the main vehicle load optimisation dialog.

The rationalised placement method is selected by default. When selected, adverse loadable areas are considered when placing a vehicle-type traffic load, ignoring the relieving areas.



Note. By selecting ‘Edit Advanced Code Options’ on the Optional code settings dialog and clicking the Advanced button some of the lesser used loading options may be modified.

Next, the influence surfaces to be utilised must be defined.

- Select the **Set influence surfaces** button.
- Ensure **Include all influence surfaces** is selected.
- Ensure that each loaded influence area is set to **Positive** in the grid. This will solve for the worst positive effects on each of the specified influence surfaces.
- Click **OK** to set the influence data and return to the main dialog.

Finally, the carriageway positions need to be defined. In this example the carriageway positions are defined from the selected kerb Lines.

- Select the **Define carriageways** button.
- Ensure the **Kerbs from selection** option is selected and click **Apply** to define the carriageways and return to the main dialog.

The vehicle load optimisation parameters are now set up.

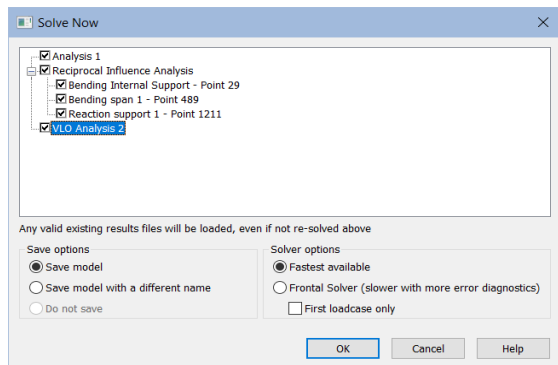
- Click **OK** on the main VLO dialog to carry out the load optimisation.

LUSAS Modeller will run an analysis using the Solver to generate each of the three influence surfaces and then run the Vehicle Load Optimisation software for the design code chosen to compute the worst position of vehicles for each influence surface and generate the corresponding loadcases in LUSAS Modeller. Once calculations have completed, these will be seen in the  Treeview.

Running the analysis

Finally, the model needs to be solved using those critical loadcases:

- Press the **Solve Now**  button and on the dialog ensure every analysis is selected and click the **OK** button to perform the analysis.



If the analysis is successful...

Analysis loadcase results are added to the  Treeview.

If the analysis fails...

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

Failing to assign geometric properties to all required lines is an easy mistake to make when working through this example. If this has been done, note that double-clicking on a line number stated in the text output window will provide options to allow you to see where in the model the line resides, and to make any correction as necessary.

Rebuilding the Model

If errors have been made in defining the model that for some reason you cannot correct, a file is provided to re-create the model information correctly. You may download this file from the user area of the LUSAS website.



- **grillage_load_modelling.lvb** carries out the modelling of the example up to the point of using the vehicle load optimisation facility.

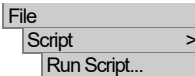


Start a new model file. If an existing model is open Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **grillage_load_optimisation**
- Use the default **User-defined** working folder.
- Ensure an Analysis type of **2D Grillage/Plate** is set.
- Click the **OK** button.



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



To recreate the model, select the file **grillage_load_modelling.lvb** that was downloaded and placed in a folder of your choosing.



Rerun the analysis to generate the results.

-  If present turn off the display of the fleshed members.

Now return to the section entitled **Vehicle Load Optimisation** earlier in this example and continue from that point.

Viewing the Results

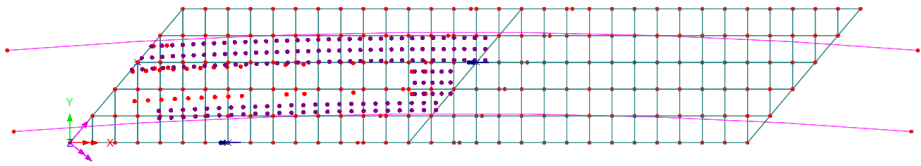
Analysis loadcase results for VLO Run 1 are present in the  Treeview.

Load Visualisation

If it is required to visualise the load pattern for a particular loadcase, this can be done by setting the results loadcase to be active.

If the **Attributes** layer is present in the  Treeview and the  Loading on/off button is depressed (signifying it is 'on') the loading patterns for the various results loadcases can be displayed:

- In the  Treeview expand **VLO Run 1** branch, expand the results folder  **Reaction support 1 - Positive - Characteristic** then right-click on the model loadcase  **Reaction support 1 - Point <n> - Positive - Characteristic** and select the **Set Active** option to display the load pattern.



Reactions at Influence Point 1 (end support)

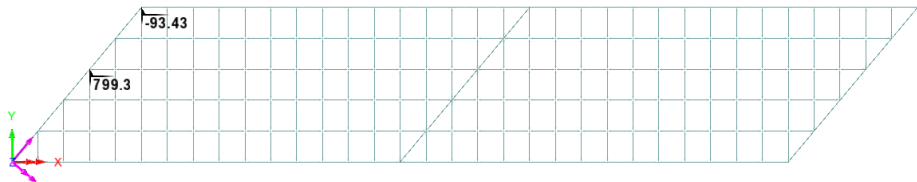
A plot showing the reactions is to be displayed.

- Turn off the display of the **Geometry** and **Attributes** layers in the  Treeview.
- With no features selected, click the right-hand mouse button in a blank part of the view window and select the **Values** option to add the values layer to the  Treeview.

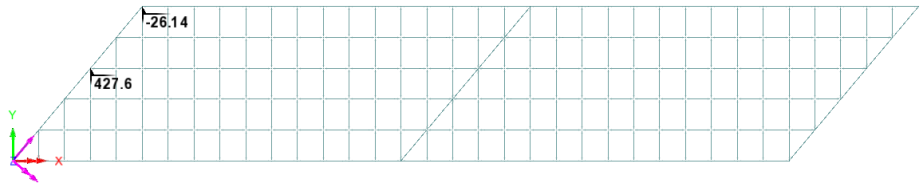
The values properties will be displayed.

- Select entity **Reaction** results for the component vertical reaction **FZ**
- Select the **Values Display** tab.
- Plot the top 1% of the **Maxima** and **Minima** values.
- Set the Number of significant figures to 4
- Select the **Choose Font** button.
- Select a font **Arial** style **Bold** size of 12
- Click **OK** to return to the values properties dialog and **OK** again.

The design reaction at the fixed bearing (previously selected for influence surface analysis) for the current loadcase **Reaction support 1 - Point <n> - Positive - Characteristic** will be displayed.



- In the  Treeview, right-click on the loadcase  **Reaction support 1 - Point <n> - Positive - Frequent** loadcase and select the **Set Active** option to display the design reaction value.

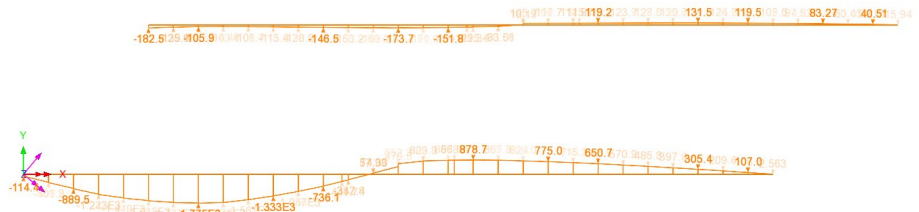


Moments at Influence Point 2 (lower edge beam span 1)

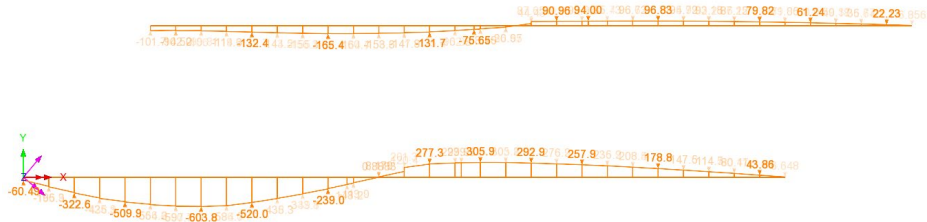
To view the bending moments in an edge beam the groups facility will be used in conjunction with the Diagrams layer.

- In the  Treeview right-click on **Edge Beams** group and select the **Set as Only Visible** option.
- Turn off the display of the **Values** layer in the  Treeview.
- In the  Treeview, right-click on the **Bending span 1 - Point <n> - Positive Characteristic** loadcase and select the **Set Active** option.
- Add the **Diagrams** layer to the  Treeview.
- Select entity **Force/Moment** results for the bending moment **My**
- Select the **Diagram Display** tab.
- With the **Label values** option selected, select the **Orientate flat to screen/page** option, and set the significant figures to **4**. Click **OK** to finish.

The peak bending moment for the characteristic case is 1775kNm.



By changing the active results loadcase to the Positive Frequent loadcase, the corresponding peak bending moment at midspan of span 1 can be found to be -603.8kNm.

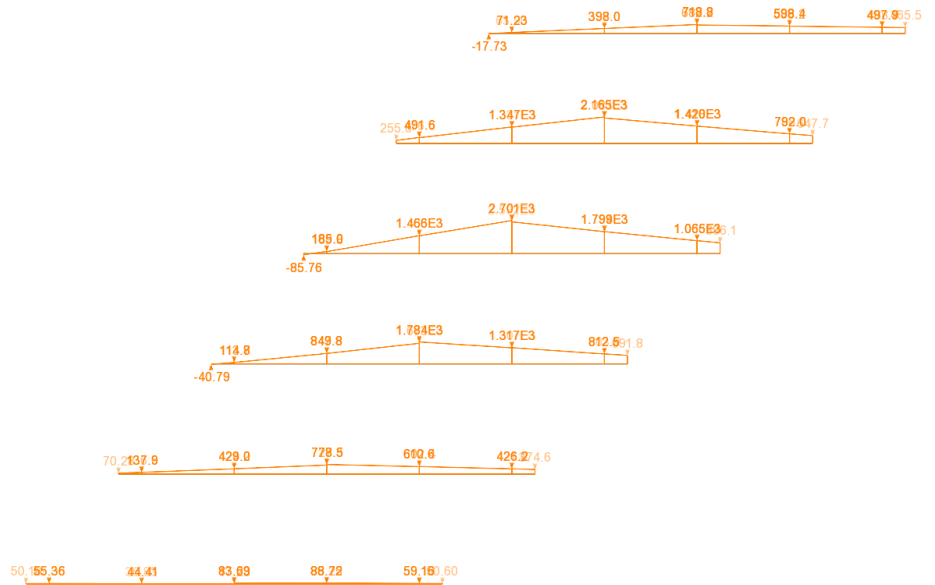


Note. If the values layer is used to display bending moments at junctions in the grillage, unaveraged values are displayed for all the members that meet at the connection. Use of the visibility options and the diagrams layer are generally more convenient.

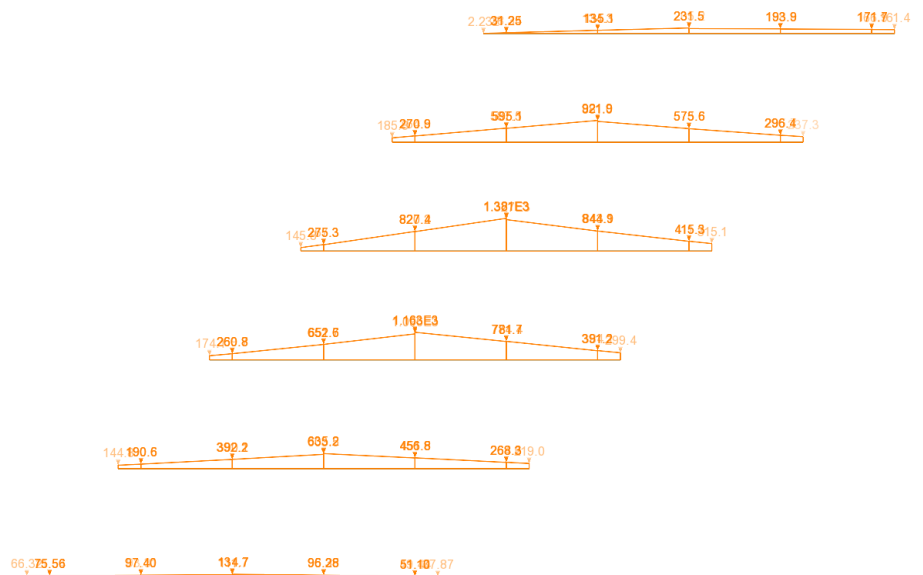
Moments at Influence Point 3 (at a central support)

- In the  Treeview, right-click on **Longitudinal Cracked Section** group and select the **Set as Only Visible** option.
- In the  Treeview, right-click on the loadcase  **Bending Internal Support - Point <n> - Characteristic** loadcase and select the **Set Active** option.

The peak bending moment for the characteristic case is 2701kNm.



By changing the active results loadcase, the corresponding peak bending moment for the frequent case can be found to be 1381kNm.



This completes the example.

Additional Notes

When modelling grillages either Reciprocal Method influence attributes (as used in this example), Direct Method attributes, or Direct Method Influence Envelope attributes may be used.

- ❑ For the **Reciprocal Method**, the influence type may be a Shear force, a Reaction, a Moment or a Displacement only. For each influence type the influence direction and either a positive or negative displacement direction needs to be specified.
- ❑ For the **Direct Method**, the influence type may be set by defining any Entity of interest (such as a Reaction, or a Force/Moment, or a Stress), an influence direction (such as an axis of a member, or a path along a structure, or a material direction) and a Component of interest (such as M_y).
- ❑ **Direct Method Influence Envelope** attributes differ from Direct Method Influence attributes in that they allow selection of all the components that are to be optimised (as opposed to only being allowed to only select one in the Direct Method). They also differ in that they are assigned to point, line or surface features in a model (as opposed to just nodes or points) and they provide the optimised load effect of interest (e.g. M_y) at all nodal locations in the assignment. If an option to include the coincident effects is also chosen, all the coincident effects (e.g. F_x , F_y , F_z , M_x , M_z) at all assigned locations will also be provided for each selected component (as opposed to having to select which ones to include individually).

When a limited number of influences are required, the reciprocal theorem can be very computationally efficient. The Direct method is often more convenient. The Direct Method Influence Envelope method can provide “whole structure” results, but at an increased computational cost.

Different worked examples are available which show these methods.