Grillage Load Optimisation

For LUSAS version:	19.0
For software product(s):	LUSAS *Bridge* and LUSAS Bridge *Plus*
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.
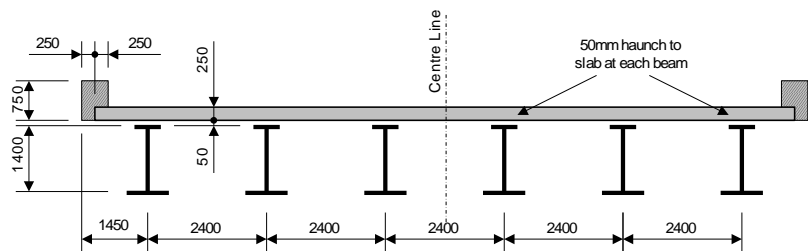
Cross Section
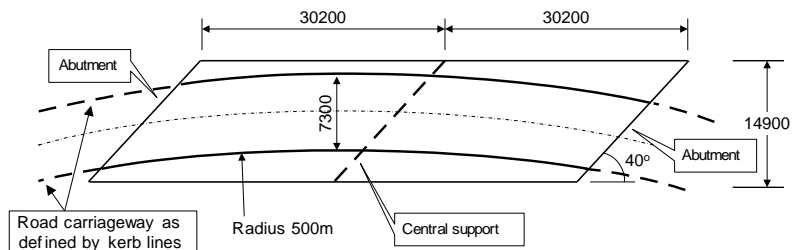
Carriageway Definition

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.

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, Vehicle Load optimisation, Load visualisation, Influence surface, Basic Load Combination, Smart Load Combination, Envelope.

Associated Files

❑ **spaced_beam_geometry.vbs** carries out the definition of the section properties.

❑ **spaced_beam_modelling.vbs** carries out the modelling of the example without the loading 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 Examples Manual Introduction.

Note. This example is written assuming a new LUSAS Modeller session has been started. If continuing from an existing Modeller session select the menu command **File>New** to start a new model file. Modeller will prompt for any unsaved data and display the New Model dialog.

Creating a new model

- Enter a File name of **spaced_beam**

- 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.

🖉 **Note.** It is useful to save the model regularly as the example progresses. This allows a previously saved model to be re-loaded if a mistake is made that cannot be corrected easily by a new user.

Feature Geometry

The grillage geometry will be created using the Grillage Wizard.

<table>
<tr><td>Bridge</td></tr>
<tr><td>Grillage Wizard...</td></tr>
</table>

- Select the **Set Default** button.

- Select **Spaced beam and slab deck** and ensure the cracked section is **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.
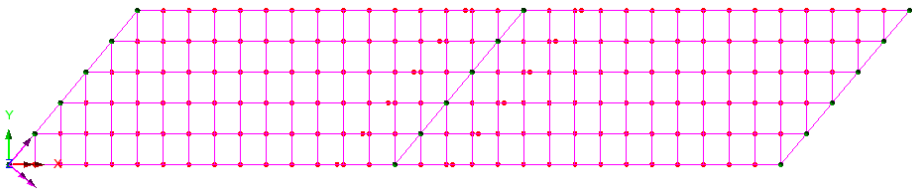
Geometric Attributes required

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.

4

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.
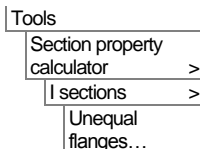
Defining the plate girder sections

Prior to defining the 'girder with top slab' grillage attributes the plate girder sections must be defined because these are referenced by the attributes. Section properties for the steel plate girders are to be calculated using the standard Section Property Calculator for 'I' sections with unequal flanges, and omitting the radius value. Both girders have an overall depth of 1.4m but their plate sizes differ.

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 section:

Tools
  Section property
  calculator   >
    I sections   >
      Unequal
      flanges…

- Enter the values shown in the following dialog. The attribute name is generated automatically.

- Ensure **Add to local library** is selected and click **Apply**

Defining a pier section:

- Update the values to be as shown in the following dialog:


- On completion click **OK**:

Note. Whilst just two Bridge Deck (Grillage) geometric attributes are created for this example, many more may be required to model the changing sections in a larger, more complex bridge deck analysis.

Defining the two 'girder with top slab' grillage attributes

Attributes
Geometric   >
Bridge Deck
(Grillage)   >
Girder with Top
Slab...

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

Defining a span girder

- With the **Girder** tab selected, define a **Constant section**.

- Click on the ![icon] 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 section **I D=1.4 Bt=0.4 Bb=0.65**…(etc.). Then click **OK** to return to the previous dialog.
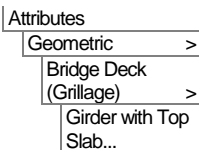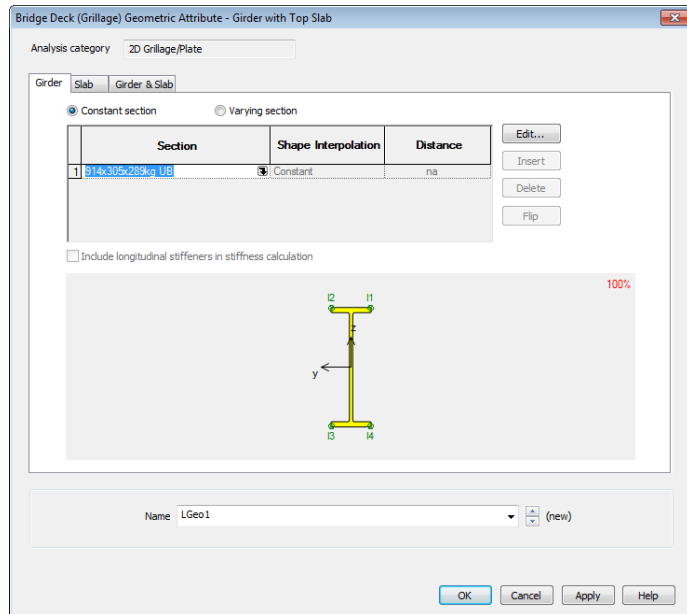
- Next, select the **Slab** tab.
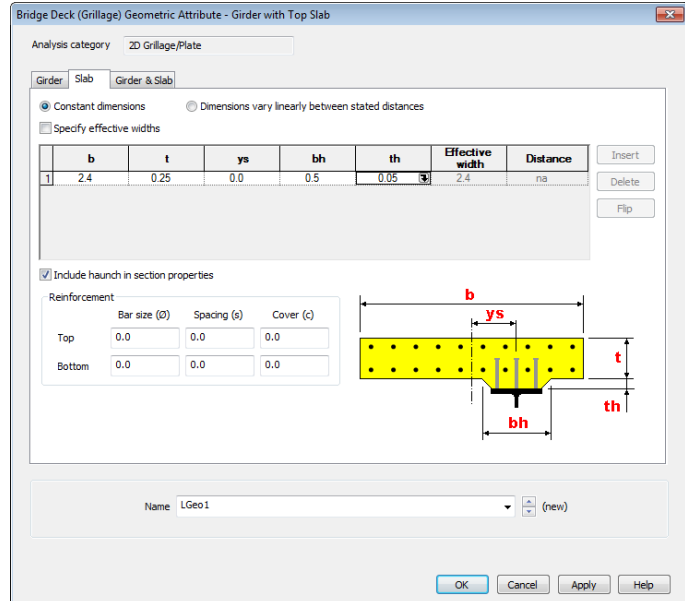
- Ensure **Constant dimensions** is selected. In this example, the reduction in slab width to account for shear lag is being neglected (as is customary in some Codes of Practice e.g. AASHTO). In practice, if this is to be considered, the 'Specify

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 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**.

- Check (tick) the **Include haunch in section properties** checkbox to include the haunch properties in the section property calculation.

- Lastly, select the '**Girder & Slab**' tab.
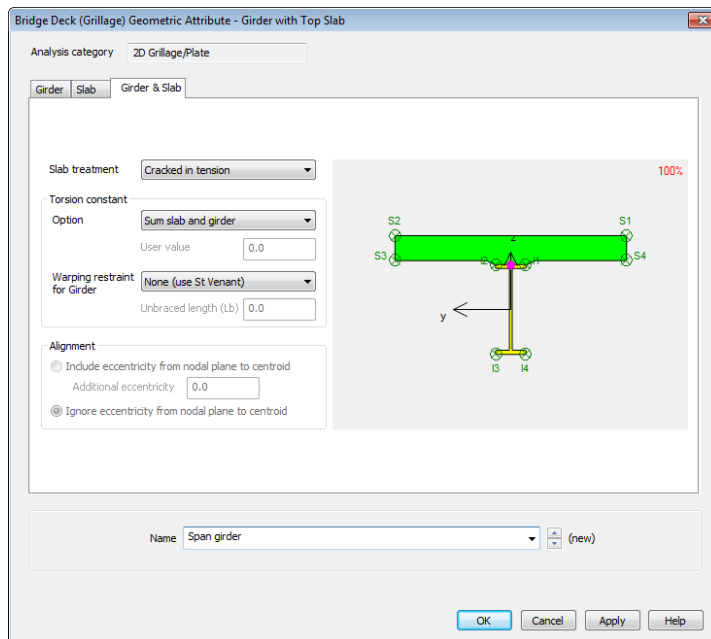
- For the slab treatment ensure that **Uncracked (concrete only)** is selected. (The 'Uncracked (gross section)' option uses an increased stiffness to account for any reinforcing bars specified, a refinement often felt unnecessary by practicing engineers.)
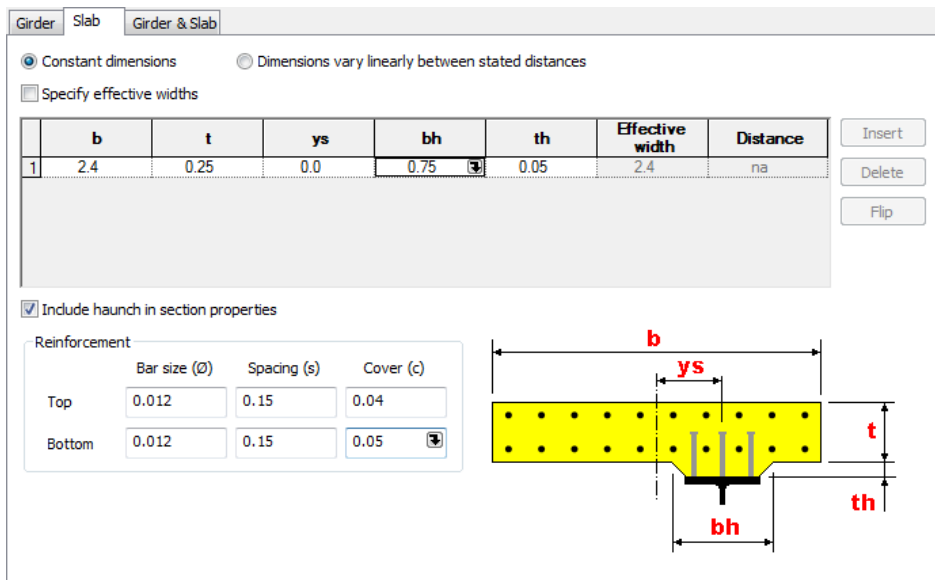
The torsional constant calculations settings are left as their default settings ('Sum slab and girder'; 'no warping restraint – use St Venant'). The warping restraint options include those from the NCHRP Report 725 (Section 3.3.2) and have been shown to be superior, particularly for cases of high skew and/or curved girders.

- Name the geometric attribute as **Span girder** and press **Apply**. This saves and retains the existing data, and allows modification to create the next attribute.

Defining a pier girder

- On the **Girder & Slab** tab, type a new name of **Pier girder**.

- Select the **Girder** tab and click on the ⬇ image in the Section cell.

- On the 'Enter Section' dialog select **User sections** from the **Local library** and choose the other girder section **I D=1.4 Bt=0.65 Bb=0.8**…(etc.). Then click **OK** to return to the previous dialog.

- Select the **Slab** tab and change the width 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).

- Lastly, select the **Girder & Slab** tab.

- Specify the slab to be **Cracked in tension**.

- Click the **OK** button to save the **Pier girder** geometric attribute.

Defining a 'slab' grillage attribute

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

Attributes
Geometric >
Bridge Deck
(Grillage) >
Slab...

- 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 the internal and end skew diaphragm beams

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

Tools
Section property
calculator >
Rectangular
Sections >
Solid…

- Enter a depth (D) of **0.75** and a width (B) of **0.5**

- Ensure that **Add to local library** is selected.

- Accept the default attribute name of **RSS D=0.75 B=0.5** that is created, and click **OK**.

To call the section back from the library:

Attributes
Geometric
Section library >

- On the 'Geometric Line' dialog select **User sections** from the **Local library** and choose the section **RSS D=0.75 B=0.5**

- Name the attribute to be **diaphragms** and click **OK**.

This adds the section to the Attributes ⚛ treeview.

Assigning 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.

- 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,** hold down the Ctrl key and also select **Transverse Intermediate Beams** and **Transverse Central Beams** groups and click on the Paste 📋 button. Click **OK** for each dialog that appears.

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. Click **OK** for each dialog that appears.

Checking for correct geometric assignment

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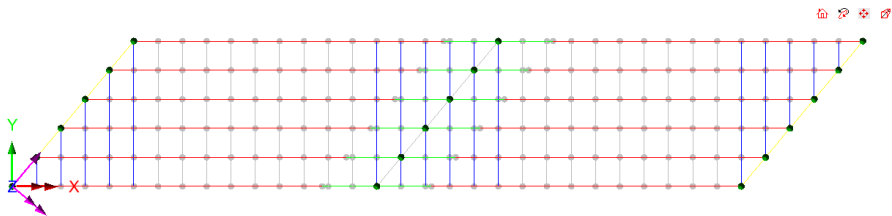
- To remove the 'Colour by Attribute' assignment , in the ⬚ Treeview double click on **Geometry**, and from the Colour by drop down list select **Own Colour** and click **OK** to update the display.
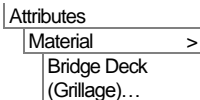
Defining the Materials

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:

Attributes	
Material	>
Bridge Deck (Grillage)…	

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**

- Ensure that the **Stiffness** is included by checking (ticking) that box, but the mass is excluded.

- Name the material **Concrete** and click **OK**.

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**, and **Ungraded**

- Ensure that the **Stiffness** is included by checking (ticking) that box, but the mass is excluded.

- Name the material **Girder** and click **OK**.

Rebar

- Click in the material cell and choose **New**… and in a similar fashion make the following selections: **Steel-Reinforcing bar**, **Europe**, **EN1992-2**, and grade **500B**

- Name the material **Rebar** and click **OK**.

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

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.
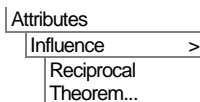
Defining Influence Attributes

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.

> Attributes
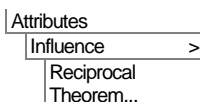>> Influence   >
>>> Reciprocal
>>> Theorem...

- 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.

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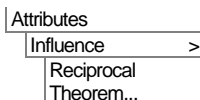
- Select a **Moment** influence type **About Transverse** axis direction for a **Positive** displacement direction (because we are interested in the maximum hogging moment). Enter the influence attribute name as **Bending Internal Support** and click **OK**.

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.
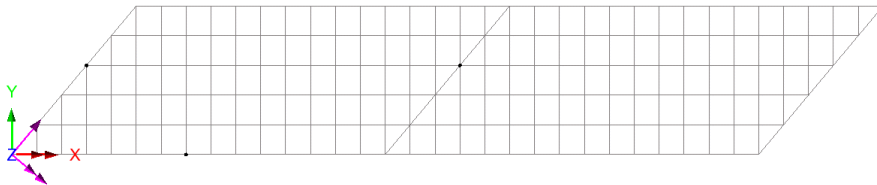
Assigning Influence Attributes

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, <u>but should only be assigned to those nodes or points for which they are applicable</u>. 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.

- Turn off the display of the **Geometry** layer in the ⬚ Treeview.

- Turn on the display of the **Mesh** layer in the ⬚ Treeview

- If necessary turn off any loading visualisation using the ⬚ Loading button.

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

- Select 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.

- Select 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.

- Select 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
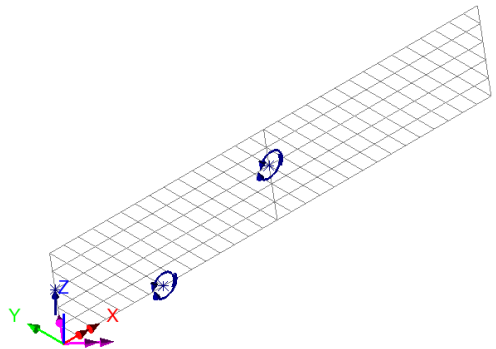
Using the Vehicle Load Optimisation facility to calculate the worst case loading patterns.

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

Before invoking the Vehicle Load Optimisation Wizard the kerb lines need to be defined. This can be done by specifying coordinates for the kerbs via the Carriagewway 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

Geometry	
Line	>
Arc/Circle	>
From	
Coords/Points	
…	
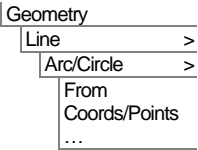
- Specify the first point as **(-5, 1)** the second point as **(75, 1)**. Define the third point as **(35, 20)** and select the **Direction** option for the third point.

- Enter the radius as **500** and ensure that a **Minor** arc type is selected.

- Click **OK** to define the first kerb line.

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

- Select the line just drawn.

Geometry	
Line	>
By offsetting…	

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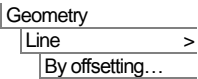
Note. Lines can be offset **towards** or **away from** either a previously selected point, or from a specified reference coordinate, or be created on **both sides** of a selected line or set of lines. See the online help for this dialog for more information

Save the model

| File |
|---|
| Save |

🖫 Save the model file.

Rebuilding a model after a previous failed analysis

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

## Carrying out Vehicle Load Optimisation

Prior to defining vehicle loading the two lines representing the kerbs at the extent of the carriageway both need to be selected.
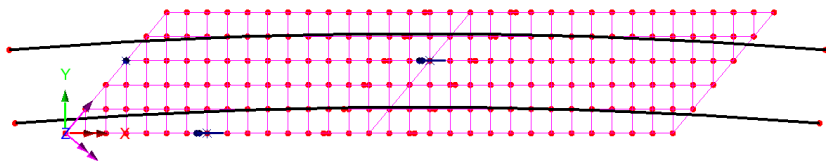

- **Select both kerb lines** (use Shift to add to the initial selection).

Bridge
Vehicle Load
Optimisation

- Select **United Kingdom** 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 <u>not</u> selected.

- Ensure the **Create loading patterns** option is selected.

- **VLO Analysis 2** will be automatically entered for the Analysis entry in the 🕒 Treeview. (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.
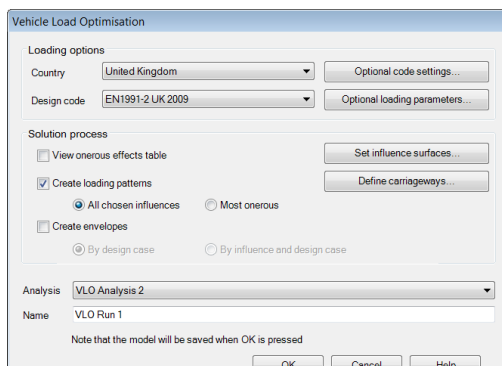
The EN1991-2 UK 2009 optional code settings will appear.

- <u>Deselect</u> **Combination (psi0)**

- For Group 5 – LM3, select the ⋯ button to <u>deselect</u> 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.
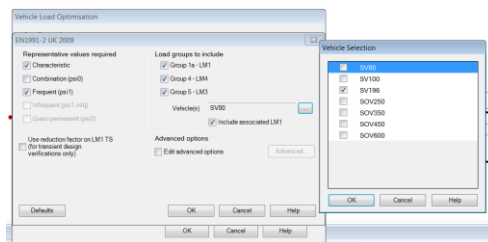
📝 **Note.** By selecting 'Edit Advanced Code Options' on the Optional code settings dialog and clicking the Advanced button some of the lesser used loading options may be modified.

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.
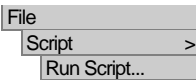
❑ **spaced_beam_modelling.vbs** carries out the modelling of the example up to the point of using the vehicle load optimisation facility.
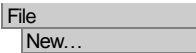
File
   New…

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 **spaced_beam** and click **OK**

File
   Script   >
      Run Script...

To recreate the model up to the point of using the vehicle load optimisation facility, select the file **spaced_beam_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

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 xxxx - Positive - Characteristic** and select the **Set Active** option to display the load pattern.
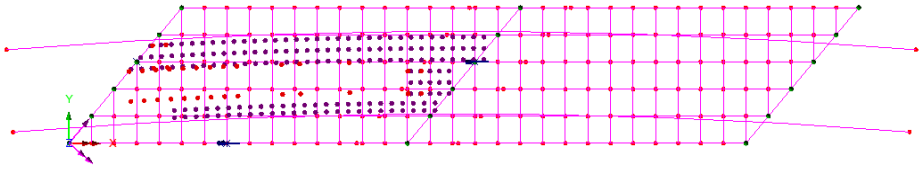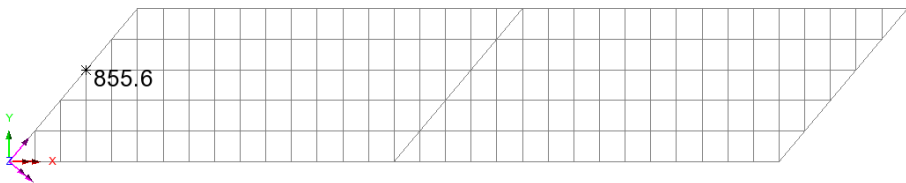
Reactions at Influence Point 1 (end support)

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 **10**% of the maxima 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 xxxx - Positive – Characteristic** will be displayed.

- In the 🕒 Treeview, right-click on the loadcase **⬛ Reaction support 1 - Point xxxx - Positive - Frequent** loadcase and select the **Set Active** option to display the design reaction value.
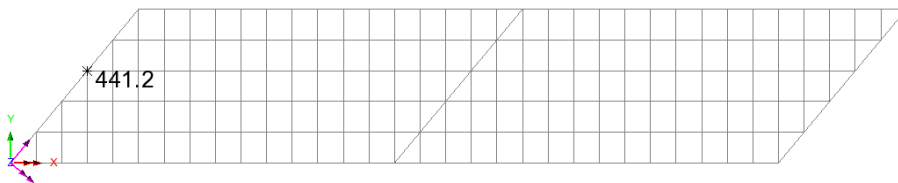
## Moments at Influence Point 2 (lower edge beam span 1)

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 xxxx - 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

- Select the **Label values** and **Peaks Only** options. Deselect the **Label only if selected** option. Select the **Orientate flat to screen/page** option, set the label font to be size **11**, the font Angle to **45** and the significant figures to **4.** Click **OK** to finish.

The peak bending moment for the characteristic case is 1.672kNm.
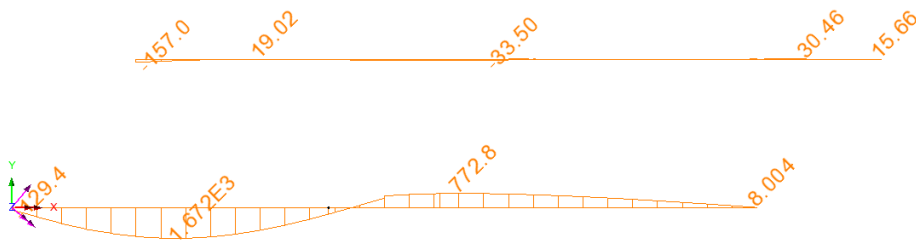
By changing the active results loadcase to the Positive Frequent loadcase, the corresponding peak bending moment at midspan of span 1 can be found to be -545.2kNm
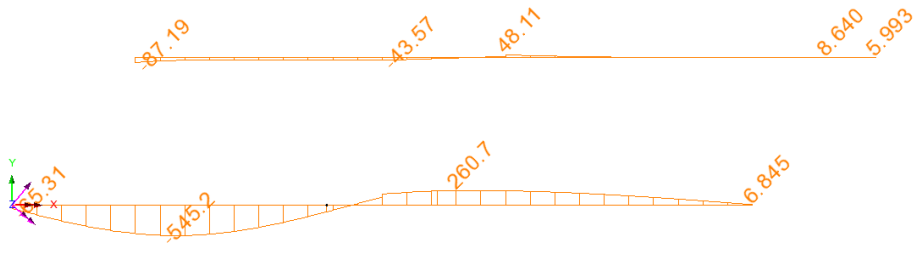
Note. If the values layer is used to display bending moments at junctions in the grillage, unaveraged values are displayed for all the members that meet at the connection. Use of the visibility options and the diagrams layer are generally more convenient.

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 - (*x,y*) - Characteristic** loadcase and select the **Set Active** option.

The peak bending moment for the characteristic case is 2.776kNm.
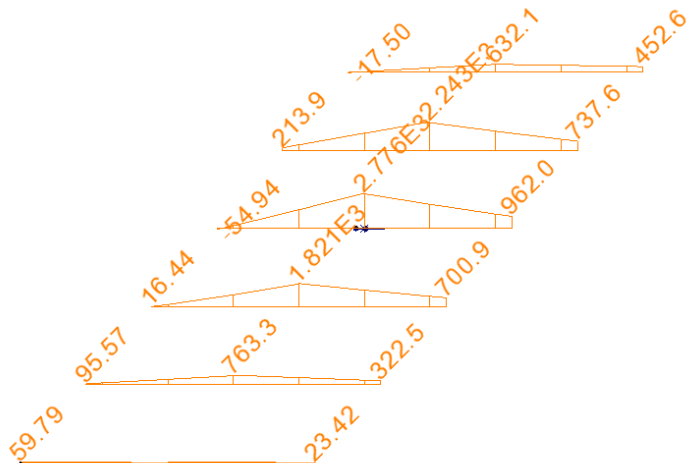
By changing the active results loadcase, the corresponding peak bending moment for the frequent case can be found to be 1430kNm.
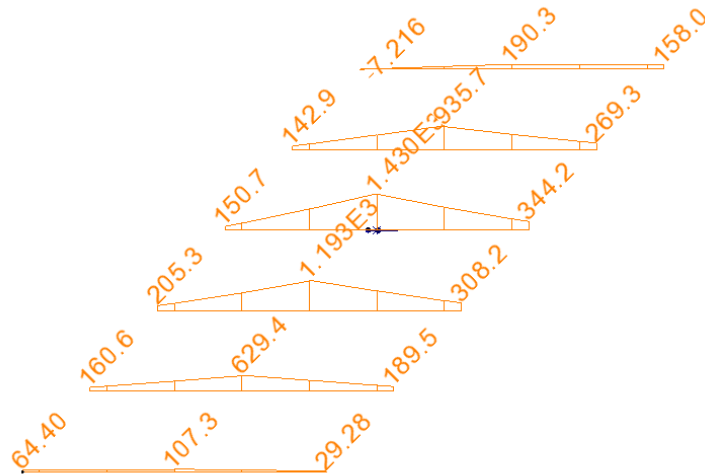
This completes the example.

Additional Notes

When modelling grillages either Reciprocal Method influence attributes (as used in this example) or Direct Method 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 My).

When a limited number of influences are required, the reciprocal theorem can be very computationally efficient. However, the Direct method is often more convenient.