Grillage Load Optimisation

For LUSAS version:	18.0
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 is applied to grillage models. Vehicle load optimisation is used for a typical design code using the reciprocal influence method.
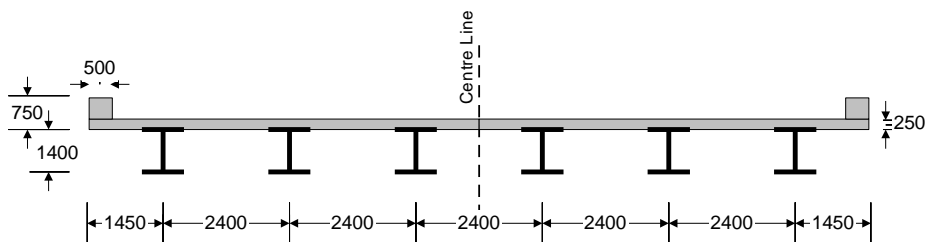
Cross Section
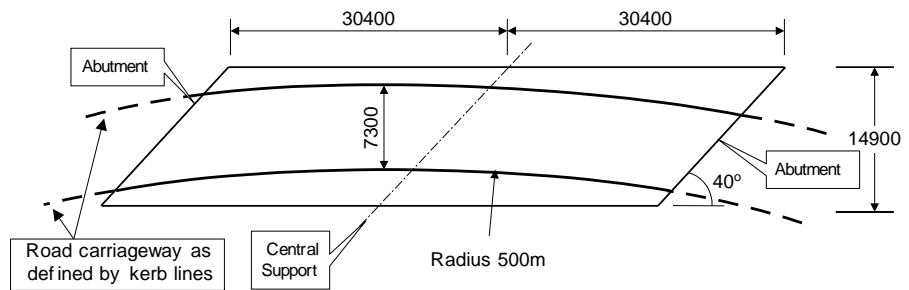
Carriageway Definition

The structure to be analysed consists of steel plate girders supporting an in-situ reinforced concrete deck slab. Two spans of 30.4m each (skew span measurement) span between the centreline of the support bearings.

The parapet beams are cast onto the structure after the deck slab has cured. This will make the edge beam properties different from the internal beams however for simplicity this will be ignored in this example. This assumption makes both the internal and edge beam properties the same. The properties are computed assuming 6 transverse beams in each span.

Note. The section properties stated are for the composite sections ignoring shear lag effects, i.e. taking into account the full width of the slab. The composite sections have been transformed using short term concrete modulii and are stated in "steel" units in the table below.

Section Property Table	**Area m^2 (Asz m$^{2)}$**	**Iyy m^4 (Izz m^4)**	**J m^4**
Edge Beam/Internal Beams (Uncracked)	0.12359	0.03777	0.697E-3
Edge Beam/Internal Beams (Cracked)	0.04151	0.01387	5.366E-6
Transverse Beams	0.1188	0.5012E-3	0.997E-3
Skew Beam	0.0684	0.2886E-3	0.575E-3
Diaphragms	0.05775	0.029E-3	0.107E-3

The structure will be designed to carry full HA loading and 30 units of HB loading as derived from BD37/88. The method used is relevant for other design codes.

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

Objectives

The required output from the analysis consists of:

❑ ULS and 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) makes use of influence surfaces and influence analysis to identify the most onerous vehicle loading patterns on bridges for a chosen design code and to apply these loading patterns to LUSAS models. A vehicle load optimisation wizard provides the means of defining parameters, for a particular design code, to generate the most critical traffic loading pattern for each influence shape under consideration.

In this example, the grillage is loaded according to design code BD37/01. After completing the example as written it could be run-through again (using the similar steps as described) for a different design code.

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.

Bridge
   Grillage Wizard...

- 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 **30.4** and change the number of internal transverse beams to **6**
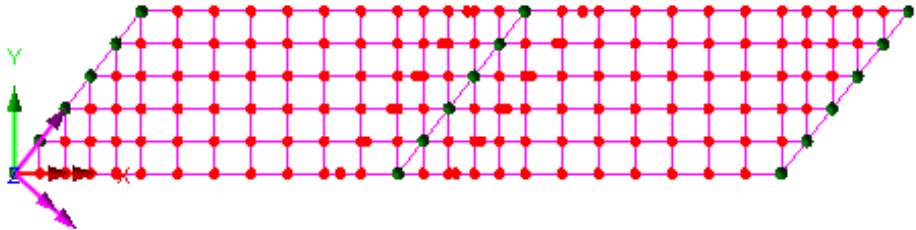
- Click **Finish** to generate the grillage model.

Defining the Geometric Properties

The geometric properties for this model are listed in the section property table at the beginning of this example. However, for ease of use they will be read in using a file that is supplied.

- Open the file spaced_beam_geometry.vbs which is located in the \<**LUSAS Installation Folder**>\Examples\Modeller directory.

Assigning Geometric Properties

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

Edge Beam/Internal Beam (Uncracked) properties

- Select the **Edge Beam/Internal Beams** (**Uncracked**) properties from the 🔵 Treeview and click on the Copy 📋 button.

- In the 🔲 Treeview select the **Longitudinal Uncracked Section** group and click on the Paste 📋 button.

Confirmation of the assignment will appear in the text window.

Edge Beam/Internal Beam (Cracked) properties

- Select the **Edge Beam/Internal Beams** (**Cracked**) properties from the 🔵 Treeview and click on the Copy 📋 button.

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

Transverse Beam properties

- Select the **Transverse Beams** properties from the 🔵 Treeview and click on the Copy 📋 button.

- In the 🔲 Treeview select the **Transverse Intermediate Beams** group and click on the Paste 📋 button.

- In the 🔲 Treeview select the **Transverse Central Beams** group and click on the Paste 📋 button.

Skew Beam properties

- Select the **Skew Beams** properties from the 🔵 Treeview and click on the Copy 📋 button.

- In the ⬡ Treeview select the **Transverse Skew Beams** group and click on the Paste 📋 button.

Diaphragm properties

- Select the **Diaphragms** properties from the ⚛ Treeview and click on the Copy 📋 button.

- In the ⬡ Treeview select the **Internal Diaphragms** group and click on the Paste 📋 button.

- In the ⬡ Treeview select the **End Diaphragms** group and click on the Paste 📋 button.

Checking for correct geometric assignment

- Turn off the display of the **Mesh** layer from the ▭ Treeview.

- Double click on the **Geometry** layer in the ▭ Treeview and select **Assignment** from the Colour by drop down list.

- Click the **Set** button and select **Geometric** from the Attribute Type drop down list.

- With the **Generate key** option selected click the **OK** button to return to the Geometry properties dialog and **OK** again to display the geometry coloured by geometric assignment with the key annotated.
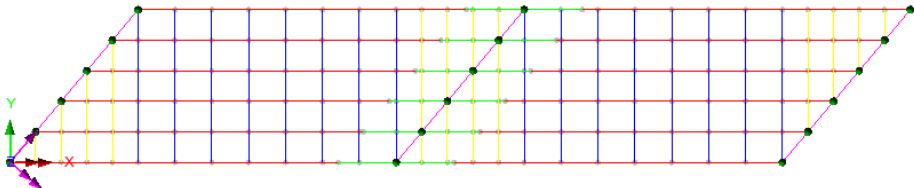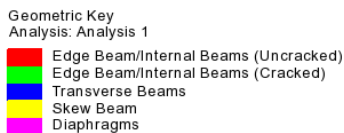
- To remove the 'Colour by Attribute' assignment double click on **Geometry** in the ▭ Treeview, select **Own Colour** from the Colour by drop down list and click **OK** to update the display.

Defining the Material
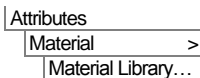
- Select material type **Steel - Structural,** Region **None**, Standard **None**, and of Grade **Mild steel** from the drop-down list and click **OK** to add the material dataset to the 🔴 Treeview.

- With the whole model selected (**Ctrl** and **A** keys together) drag and drop the material dataset **Iso1 (Mild Steel | Steel - Structural)** from the 🔴 Treeview onto the selected features.

Note. Because the geometric properties provided are transformed section properties (as mentioned in the example's description) Mild Steel properties are assigned to the complete model.

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. If in-plane effects such as braking forces are to be considered the grillage elements should be replaced by 3D beam elements and the appropriate properties and supports should be assigned.

Self-Weight
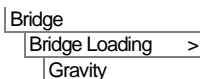
Selecting this menu item adds **Gravity -ve Z** to the 🔴 Treeview.

- With the whole model selected, drag and drop the loading dataset **BFP1 (Gravity - ve Z)** from the 🔴 Treeview onto the selected features. Ensure the **Assign to lines** option is selected and click **OK** to assign to **Loadcase 1**.

The gravity loading will be visualised on the model.

- From the ⏱ Treeview expand **Analysis 1** and then, using the right-hand mouse button, select **Loadcase 1** and select the **Rename** option. Change the loadcase name to **Self Weight**
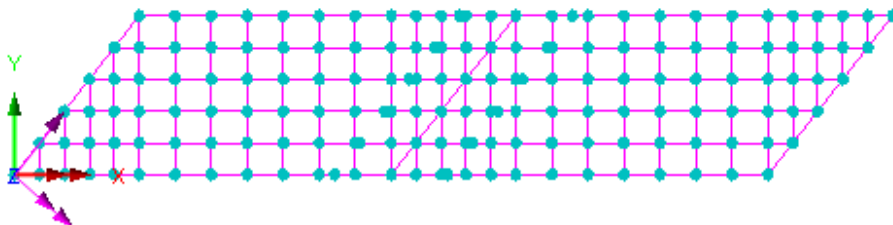
Defining Influence Attributes

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.
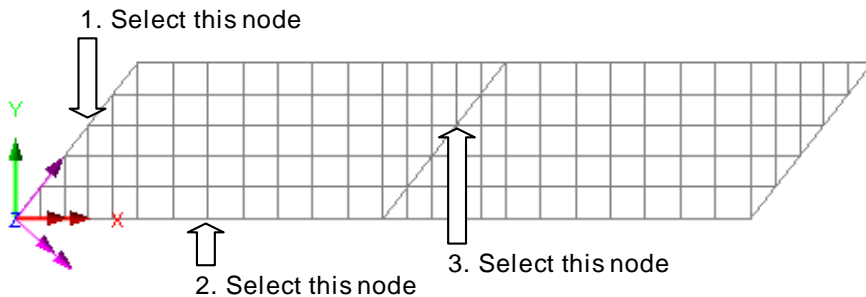
Assigning Influence Attributes

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 applies. For nodes where a positive influence shape exists 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
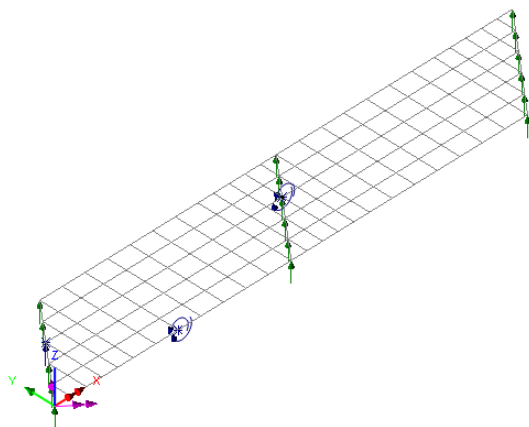
Using the Vehicle Load Optimisation facility to calculate the worst case loading patterns.

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

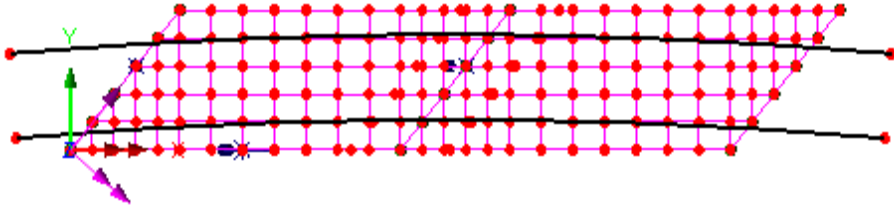| Geometry	
Line	>
Arc/Circle	>
From	
Coords/Points	
…	
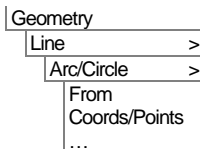
- Specify the first point as **(-5, 1)** the second point as **(75, 1)**. Define the third point as **(35, 20)** and select the **Direction** option for the third point.

- Enter the radius as **500** and ensure that **Minor Arc** is selected.

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

Geometry	
Line	>
Arc/Circle	>
From	
Coords/Points	
…	
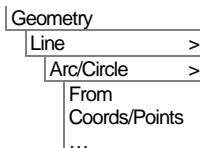
- Specify the first point as **(-5.584, 8.2766)** and the second point as **(75.584, 8.2766).** Define the third point as **(35, 20)** and set the **Direction** option for the third point.

- Enter a radius of **507.**3 and ensure that **Minor Arc** is selected.

- Click the **OK** button to define the second kerb line.

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

🖉 **Note.** It is also possible to generate the upper kerb line by offsetting a line from the lower one by first selecting the lower line and choosing the Geometry > Line > By Offsetting menu item, and the specifying an offset distance equivalent to the width of the carriageway to be generated (7.3m in this case), and generating it away from the reference position (or from 0,0).

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.
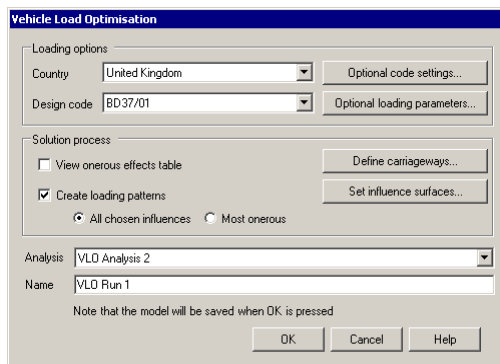
Carrying out Vehicle Load Optimisation

- Ensure the two kerb lines are selected before choosing the next menu item.

| Bridge
| Vehicle Load
| Optimisation

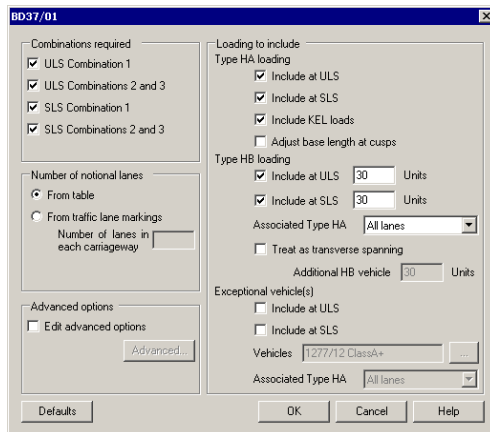With the kerbs selected the vehicle load optimisation can begin.

- Select **United Kingdom** from the Country drop down list.

- Select **BD37/01** 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. (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 BD37/01 optional code settings will appear.

- Press the Defaults button to set all values to their defaults and make the following changes:

- For Type HA loading deselect **Adjust base length at cusps**

- For Type HB Loading change the number of units to include at ULS to **30**

- 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.

- Ensure 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.

- Select the **Kerbs from selection** option and click **Apply** to define the carriageways and return to the main dialog.

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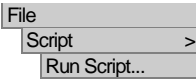
❑ **spaced_beam_modelling.vbs** carries out the modelling of the example up to the point of using the vehicle load optimisation facility.
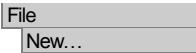
File
    New…

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 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 - ULS Combination 1** then right-click on the model loadcase **N Reaction support 1 - Point xxxx - Positive - ULS Combination 1** and select the **Set Active** option to display the load pattern.
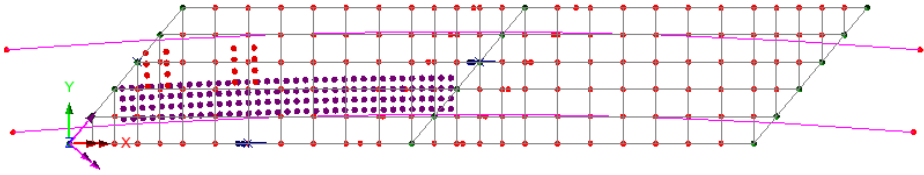
Reactions at Influence Point 1

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 - ULS Combination 1** will be displayed.
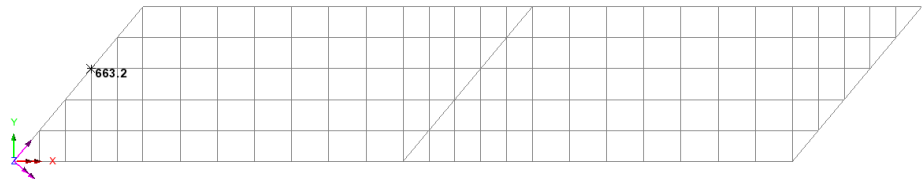
- In the Treeview, right-click on the loadcase **Reaction support 1 - Point xxxx - Positive - ULS Combinations 2 and 3** loadcase and select the **Set Active** option to display the design reaction value.
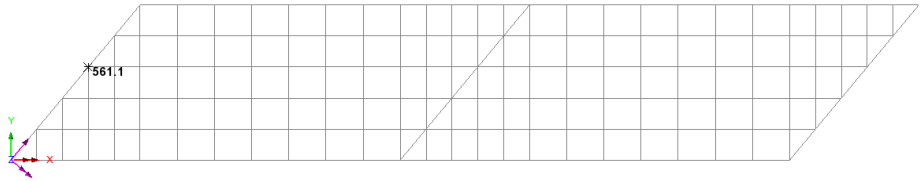
- Select the remaining Reaction Support 1 results loadcases in turn and choose **Set Active** to display the other corresponding SLS reaction values which are shown in the following table:

Location (Group name)	Quantity	ULS1 (Comb 1)	ULS2 (Combs 2 +3)	SLS1 (Comb 1)	SLS2 (Combs 2 +3)
Reaction Support 1 (End diaphragms)	Vertical reaction (FZ)	663.2	561.1	561.1	510.1

Moments at Influence Point 2

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 - ULS Combination 1** 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** option, **Label only if selected** option, **Orientate flat to screen/page** option. Deselect the **Peaks Only** option, set the label font to be size **20**, the font Angle to **45** and the significant figures to **4**
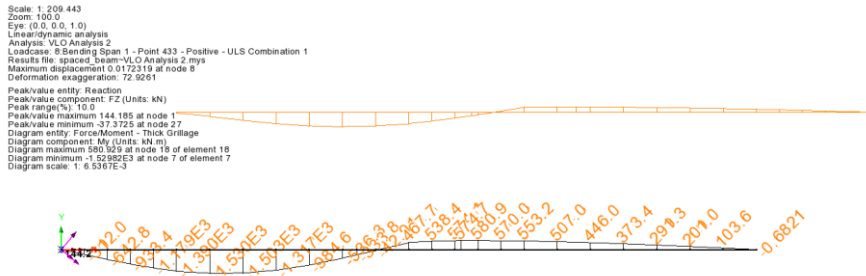
- Click **OK** to finish.

Note. Summary information (as shown in the image that follows) can be added by ensuring the Window Summary check box is ticked. This is found in the panel beneath the 🔲 Treeview.

The window summary shows the maximum and minimum values.

- Drag a box around the beams of interest to view moment values on the beam.

By changing the active results loadcase this table of results for the design of the edge beam can be generated.

Location (Group name)	Quantity	ULS1 (Comb 1)	ULS2 (Combs 2+3)	SLS1 (Comb 1)	SLS2 (Combs 2+3)
Lower edge beam (Edge Beams)	Bending moment (My)	-1529	-1294	-1294	-1176

Note. By selecting **Peaks Only** in the **Diagram Display** tab the labels can be limited only to the maximum and minimum in the selection.

Note. If the values layer was used to investigate bending moments at points in the grillage, care must be taken to ensure that unaveraged values are displayed for any number of grillage elements coming together at a node rather than averaged values. If so done, end shrinkage settings will allow the separate values to be seen in isolation.

Moments at Influence Point 3

- 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*) - ULS1/Positive** loadcase and select the **Set Active** option.

- Drag a box around beams of interest to view hogging moment values.

Scale: 1: 84.3052
Zoom: 100.0
Eye: (0.0, 0.0, 1.0)
Linear/dynamic analysis
Analysis: VLO Analysis 2
Loadcase: 4:Bending Internal Support - Point 29 - Positive - ULS Combination 1
Results file: spaced_beam~VLO Analysis 2.mys
Maximum displacement 9.81545E-3 at node 98
Deformation exaggeration: 51.5342

Peak/value entity: Reaction
Peak/value component: FZ (Units: kN)
Peak range(%): 10.0
Peak/value maximum 825.492 at node 95
Peak/value minimum 23.1984 at node 149
Diagram entity: Force/Moment - Thick Grillage
Diagram component: My (Units: kN.m)
Diagram maximum 1.64066E3 at node 95 of element 173
Diagram minimum –499.36 at node 98 of element 175
Diagram scale: 1: 6.09512E-3

622.6
704.9
1.128E3
1.641E3
655.2
272.2
499.4
825.5
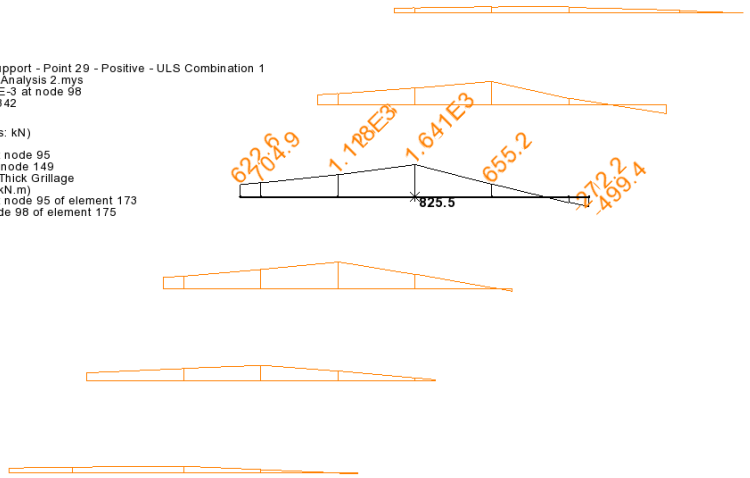
By changing the active loadcase the remaining ULS and SLS results for the design of the longitudinal beam over the support can be generated.

Location (Group name)	Quantity	ULS1 (Comb 1)	ULS2 (Combs 2+3)	SLS1 (Comb 1)	SLS2 (Combs 2+3)
Central pier (Longitudinal Cracked Beams)	Bending moment (My)	1641	1388	1388	1262

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).