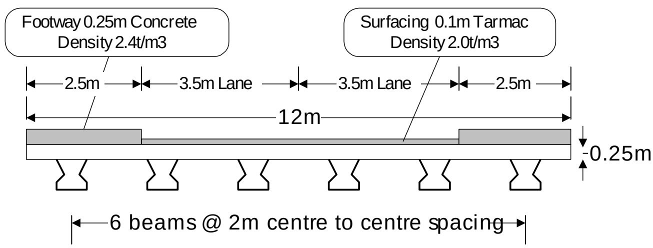
Simple Grillage

For LUSAS version:	18.0
For software product(s):	LUSAS *Bridge.*
With product option(s):	None.
This example can be used with the LUSAS Teaching and Training version by making one change to the entered data where shown.	

Description

A bridge deck is to be analysed using the grillage method. The geometry is as shown below. All members are made of C50 concrete to BS5400. Section properties of the longitudinal beams and diaphragms are to be calculated using the Section Property Calculator facility.



Cross-section through deck

The structure is subjected to four loadcases: Dead load, Superimposed dead load, Lane loads in both lanes (UDL and KEL), and an abnormal load (HB) in the lower notional lane with a lane load (UDL and KEL) in the upper lane.

Units of N, mm, t, s, C are used in calculating the section properties of selected components.

Units of kN, m, t, s, C are used throughout when modelling the grillage.

Objectives

The required output from the analysis consists of:

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

Keywords

2D, Inplane, Y6 Precast Section, Section Property Calculation, Local Library, Grillage/Plate, Grillage, Basic Load Combination, Smart Load Combination, Enveloping, Deformed Mesh, Bending Moment Diagram, Print Results Wizard

Associated Files

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

Modelling

Running LUSAS Modeller

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

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

Before creating the grillage model (requiring an analysis category of 2D Grillage/Plate) the section properties of the longitudinal beams and end diaphragms are to be computed using the section property calculator and stored for future use. Calculation of section properties requires section geometry to be drawn/defined in the XY plane. An analysis category of 2D Inplane is initially used.

Creating the Longitudinal Beam Model

File
New…

Create a new model, and on the dialog:

- Enter a file name of **Y6**
- Use the **Default** user-defined working folder.
- Ensure an Analysis type of **Structural** is set.

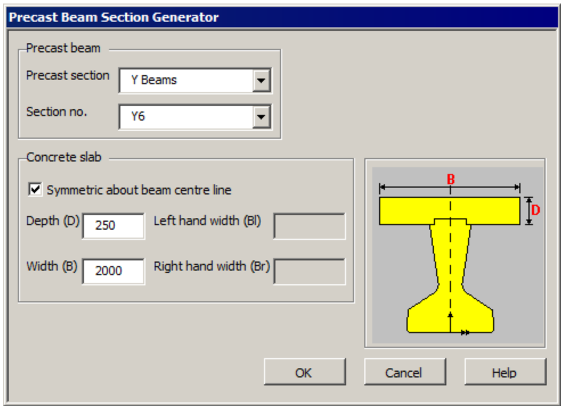
- Select an Analysis Category of **2D Inplane**
- Set Model units of **N,mm,t,s,C**
- Ensure Timescale units are in **Seconds**
- Select a Startup template of **None**
- Ensure the Layout grid is set as **None**
- Enter a Title of **Y6 Precast Beam**
- Click the **OK** button

Defining Longitudinal Beam Geometry

Tools
Section Property Calculator >
Precast Section…

- From the **Y Beams** section series select a **Y6** section.
- Specify a slab depth of **250**
- Enter a slab width of **2000**
- Click the **OK** button.

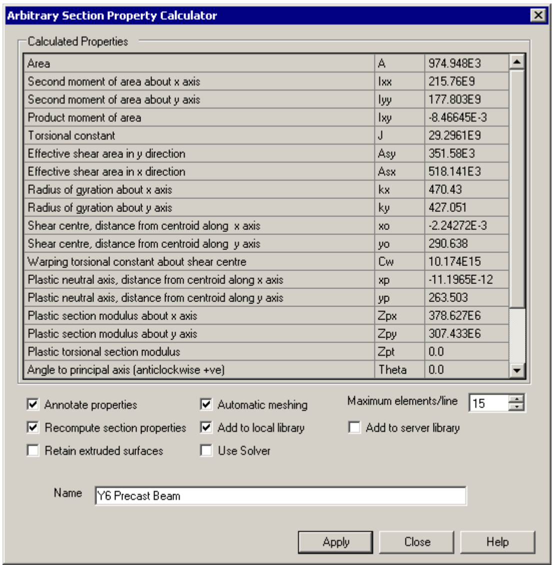
The section will now be drawn.



Calculating Longitudinal Beam Section Properties

Tools
Section Property Calculator >
Arbitrary Section…

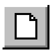
- Press **Ctrl** and **A** to select the two surfaces defining the Y6 section
- Select the option **Add to local library** so the calculated properties will be available from the local library when required.
- Name the section **Y6 Precast Beam**.
- Click the **Apply** button to compute the section properties. These will be displayed in the greyed text boxes on the right hand side of the dialog and written to the local library.



- Click the **Close** button to close the dialog.

Creating the End Diaphragm Model

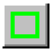
File
New…

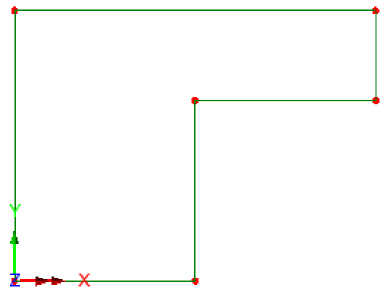

Create a new model and discard the changes to the previous model. On the dialog:

- Enter a file name of **diaphragm**
- Use the **Default** user-defined working folder.
- Ensure an Analysis type of **Structural** is set.
- Select an Analysis Category of **2D Inplane**
- Set Model units of **N,m,kg,s,C**
- Ensure Timescale units are in **Seconds**
- Select a Startup template of **None**
- Ensure the Layout grid is set as **None**
- Enter a Title of **End diaphragm**
- Click the **OK** button

Defining Diaphragm Geometry

Geometry
Surface >
Coordinates…

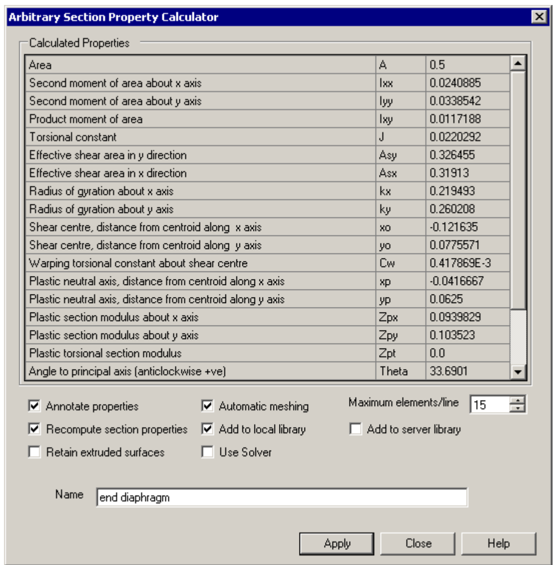

Enter coordinates of **(0,0)**, **(0.5,0)**, **(0.5,0.5)**, **(1,0.5)**, **(1,0.75)**, **(0,0.75)** to define a surface representing the end diaphragm and slab (which is to be represented by the end beam on the grillage model) and click **OK**



Calculation of End Diaphragm Section Properties

Tools
Section Property Calculator >
Arbitrary Section…

- Select the option **Add local library** and name the section **End Diaphragm**.
- Ensure the option **Add to local library** is selected.
- If the LUSAS Teaching and Training version is in use, the **Maximum elements / line** number must be changed from 15 to **10** in order to not exceed the element node limits of this version.
- Click the **Apply** button to compute the section properties. These will be displayed in the greyed text boxes on the right hand side of the dialog and written to the local library file for future use.

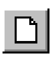


- Click the **Close** button to close the dialog.

Creating the Grillage Model

Now that the beam and diaphragm properties have been calculated the grillage model representing the entire bridge deck can be created.

File
New…


Create a new model and discard the changes to the previous model.

- Enter a file name of **grillage**
- 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**
- Ensure Timescale units are in **Seconds**
- Select a Startup template of **None**
- Ensure the Layout grid is set as **None**
- Enter a Title of **Simple grillage analysis**
- Click the **OK** button

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

Using the Grillage Wizard

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

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

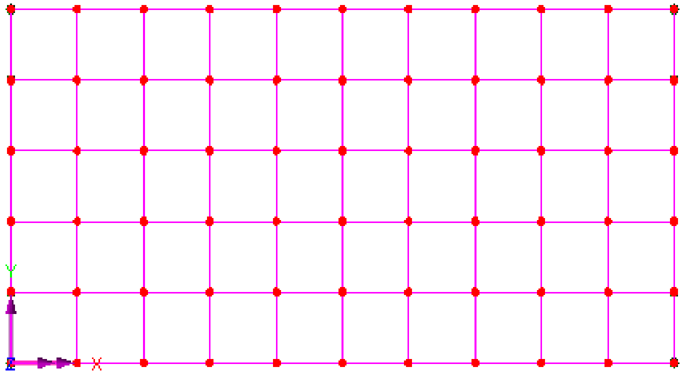
a) Longitudinal beams within the grillage should be coincident with the actual beams within the structure.

b) Transverse beams should have a spacing which is similar or greater than that of the longitudinal beams and the total number of transverse beams should be odd to ensure a line of nodes occur at mid span.

Bridge
Grillage Wizard…

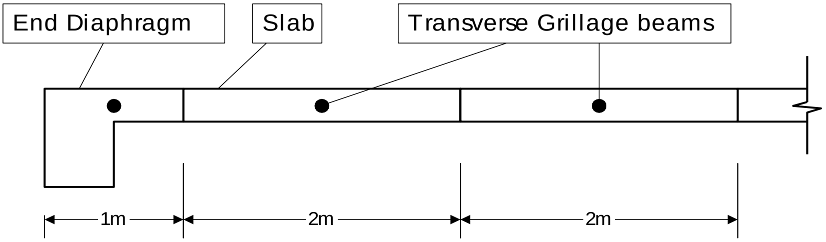
- Select the **Set default** button
- Ensure **Slab deck** is selected and click **Next**
- The grillage is **Straight** with **0** degrees skew so click **Next** again
- Select **evenly spaced** longitudinal beams, and enter the grillage width as **10** and the number of longitudinal (including edge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

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



Calculation of Transverse Beam Section Properties

The internal transverse beams each represent a 2m width of slab so the section properties are computed for an equivalent solid rectangular section, and the section is added to the local library for assignment to the model later in the example.



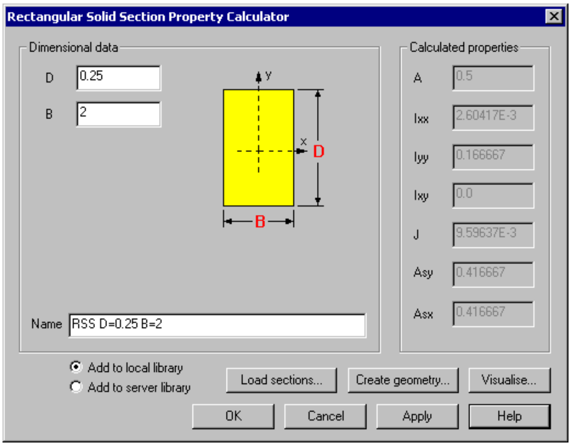
Longitudinal Section

Tools
Section Property Calculator >
Rectangular Sections > Solid

- Enter a depth of D = **0.25**
- Enter a width of B = **2**

The section properties will be displayed in the greyed text boxes on the right hand side of the dialog.

Note the torsion constant (J) is calculated to be 0.0096. This is based on beam theory and is not appropriate to represent a slab in a grillage analysis. It will be adjusted later in the example.



A section name of RSS D=0.25 B=2 is automatically created from the entered dimensions.

- Ensure the **Add to local library** option is selected and click **OK** to add the properties to the local section library.

Modifying Section Properties for Grillage Analysis

When representing an isotropic slab using a grillage model, the effective torsion constant (per unit width) can be shown to be $c = d^3/6$ (per unit width). It is therefore common practice to assume 50% of the value calculated using beam theory for a wide slab-like beam. In this example therefore:

- The transverse members represent only the slab and therefore their torsion constant can be entered as $c = bd^3/6$ (i.e. 50% of the section library value).
- The longitudinal members represent the precast beam and associated width of slab, and therefore this reduction is only applied to the proportion of the torsion constant exhibited by the slab.

When the transverse 0.25m deep 2m wide slab and the Y6 precast (longitudinal) beams are selected from the local section library their section properties will be adjusted accordingly.

Adding Section Library Items to the Treeview

First, add the transverse slab section geometric attribute:

Attributes
Geometric >
Section Library...

- Select the **User Sections** library from the upper-right drop-down list.
- Select the **Local** library type.

- Select the **RSS D=0.25 B=2** entry from the drop down list.
- Select the **Enter Properties** radio button (which permits editing of the calculated section properties for Grillage use) and reduce the torsion constant to **0.005 (**as previously discussed: $c = bd^3/6$).
- Enter an Attribute name of **Transverse Slab** and click **OK** to add the section properties to the Treeview.

Next, add the Y6 beam geometric attribute:

Attributes
Geometric >
Section Library...

- Select the **User Sections** library from the upper drop-down list.
- Select the **Local** library type.
- Select **Y6 Precast Beam** from the name in the drop-down list.
- Select the **Enter Properties** radio button and modify the torsion constant to be **0.020** (The computed value of the precast beam alone, 0.0143, plus the reduced contribution from the slab; $c = bd^3/12 = 0.005$).
- Enter an Attribute name of **Y6 Precast Beam** and click **OK** to add the section properties to the Treeview.

Note. Even though the Y6 beam was defined in millimetres the units can be extracted from the library in metres. The units will be set to metres automatically as these were the units selected on the New Model dialog.

Lastly, define the end diaphragm geometric attribute:

Attributes
Geometric >
Section Library...

- Select the **User Sections** library from the upper drop-down list.
- Select the **Local** library type.
- Select **End diaphragm** from the drop down list.
- Leave the Attribute name of **LGeo3** and click **OK** to add the section properties to the Treeview.

Note. When a section is used from a library without amending both its section properties, or the section's orientation (as in the case of the End diaphragm) the library name is appended automatically to the automatic identifying name given in the dialog. This will be of the type LGeo1, LGeo2 etc, signifying Line Geometric properties.

Assigning Geometric Properties to the Grillage Members

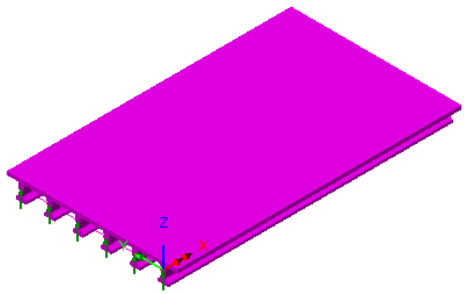
Use the Isometric View button to rotate the model so that the following assignment of the geometric properties can be seen.

Ensure the fleshing button is depressed in the toolbar menu.

Longitudinal members

The Y6 beam section properties are to be assigned to all the longitudinal members.

- In the Treeview select the **Y6 Precast Beam** entry and click on the copy button.
- In the Treeview select the **Edge Beams** group and click the paste button to assign the Y6 beam section properties to the edge beams.



Confirmation of the assignment will appear in the text window, and the edge beams will be visualised in the view window.

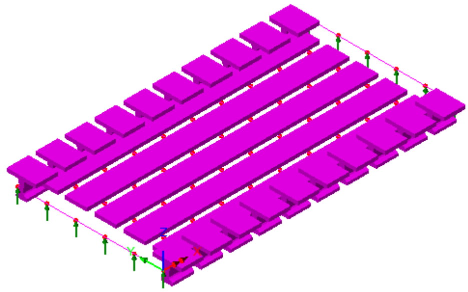
- Now, select the **Longitudinal Beams** group and click the paste button again to assign the Y6 beam section properties.

Transverse slab members

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

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

Diaphragm members



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

- In the Treeview, double click on the **Attributes** name, click the **Geometric** tab and press the **Settings** button. For cross-section end shrinkage select the **Automatic** option, and click **OK** to update the display.

- In the Treeview right-click on the **Longitudinal Beams** entry and select **Invisible** from the context menu. This makes it easier to see the orientation of the end diaphragm members.

The diaphragm section properties are assigned to the end diaphragms in a similar fashion.

- In the Treeview select the **LGeo3 (End diaphragm)** entry and click on the copy button.

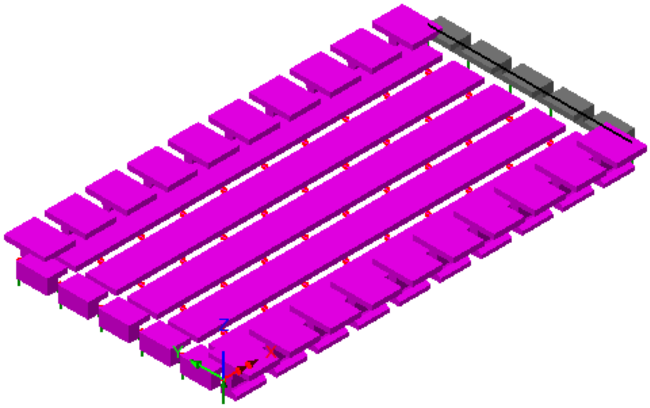
- In the Treeview select the **End Diaphragms** group and click the paste button to assign the slab section properties.

From the fleshed image it can be seen that the end diaphragm members for the far end are incorrectly displayed, and as a result the line directions of the lines to which they have been assigned need to be reversed.

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

Geometry
Line >
Reverse…

This reverses the line directions of the selected lines to give the following image.



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

Turn off the display of the fleshed members

Note. You can also check assignments by right-clicking on a group name in the Treeview and selecting Select Members.

Defining the Material

Note. In this example a single material property will be used. If deflections and rotations are of interest then separate analysis runs with short and long term properties may be appropriate.

Attributes
Material >
Material Library…

- Select material **Concrete BS5400** from the drop down list, and **Short Term C50** from the grade drop down list.
- 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 (Concrete BS5400 Short Term C50)** from the Treeview onto the selected features and assign to the selected Lines by clicking the **OK** button.

Loading

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

Renaming the Loadcases

- In the Treeview expand **Analysis** 1 and right click on **Loadcase 1** and select the **Rename** option.
- Rename the loadcase to **Dead Load** by over-typing the previous name.

Dead Load

Dead load is made up of the self-weight of the structure, which is defined as acceleration due to gravity.

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

Turn off the display of the fleshed members

Bridge
Bridge Loading >
Gravity

- A load dataset named **BFP1 (Gravity -ve Z)** will be added to the Treeview.
- In the Treeview select the **BFP1 Gravity -ve Z** entry and click on the Copy button.
- In the Treeview select the **Edge Beams** group and click the Paste button.
- With the **Assign to lines** and **Single Loadcase** options selected click the **OK** button to assign the gravity loading to the **Dead Load** loadcase.

Loading arrows confirming the assignment of the self-weight dead load on the edge beams will be displayed.

- In the Treeview select the **Longitudinal Beams** group and click the Paste button to gravity loading.
- With the **Assign to lines** and **Single Loadcase** options selected click the **OK** button to assign the gravity loading to the **Dead Load** loadcase.

Loading arrows confirming the assignment of the self-weight dead load on the internal longitudinal beams will be added to the display.

Superimposed Dead Load

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

Bridge
Bridge Loading >
Surfacing…

Specify the surfacing loading for the footway:

- Leave the density as **2.4**
- Change the thickness to **0.25**
- Set the length to **20** and set the width to **2.5**
- Leave the skew angle as **0** and the origin as **Centre**
- Click the **Apply** button to add a **Pch2 (Surfacing 20.0m x 2.5m Skew=0.0deg Thickness=0.25m Density=2.4 t/m^3)** loading attribute to the Treeview.

Now specify the tarmac highway surfacing load:

- Change the density to **2.0**
- Change the thickness to **0.1**
- Leave the length as 20 but change the width to **3.5**
- Click the **OK** button to add a **Pch3 (Surfacing 20.0m x 3.5m Skew=0.0deg Thickness=0.1 Density=2.0t/m^3)** loading attribute to the Treeview.

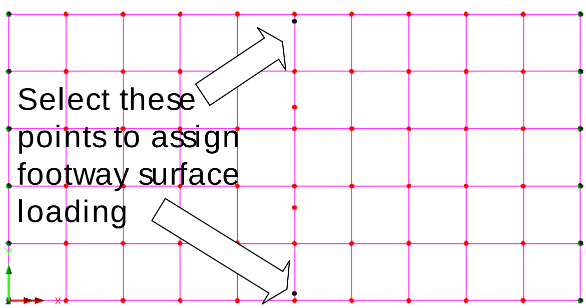
Discrete point and patch loads are positioned by assigning them to points which do not have to form part of the model.

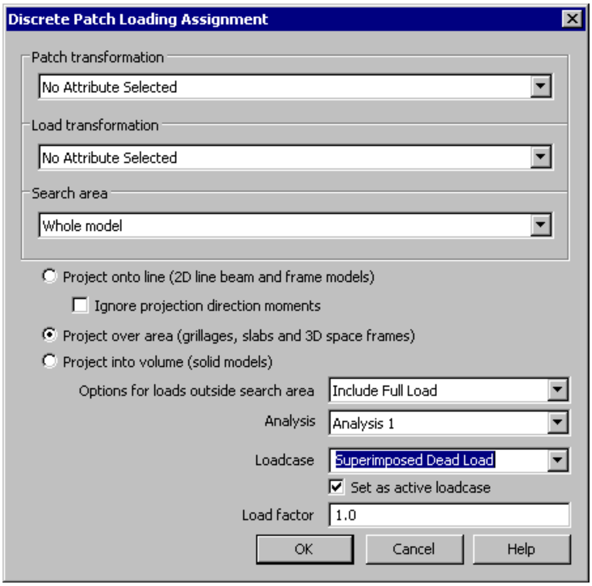
Geometry
Point >
By Coords…

Enter the coordinates of the mid-point of each footway and each notional lane **(10,0.25), (10,3.25), (10,6.75), (10,9.75)** and click **OK**

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

- Select the points at the centre of each footway by holding the **Shift** key down to select points after the initial selection.
- Drag and drop the discrete loading dataset **Pch2 (Surfacing 20.0m x 2.5m Skew=0.0deg Thickness=0.25m Density=2.4 t/m^3)** onto the selected points.





- Select **Include Full Load** from the drop down list. This will ensure the portion of the pavement load which is overhanging the edge of the grillage model is applied to the edge beams.
- Enter **Superimposed Dead Load** as the Loadcase name and click **OK** to assign the loading.

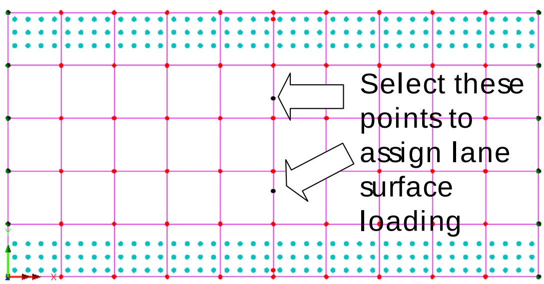
The loading will be visualised.

Patch load divisions

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

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

Now the road surfacing is to be assigned:



- Select the two points at the centre of each notional lane.
- Drag and drop the discrete load dataset **Pch3 (Surfacing 20.0m x 3.5m Skew=0.0deg Thickness=0.1 Density=2.0t/m^3)** from the Treeview onto the selected points.
- Leave the loading option for loads outside the search area set as **Exclude All Load** because for this load type it is irrelevant whether include or exclude is used since the load length, which is positioned centrally, is the same length as the span length

- Select **Superimposed Dead Load** from the Loadcase drop down list.
- Click **OK** to assign the road surfacing load. The loading will be visualised.

Vehicle Load Definition

Bridge
Bridge Loading >
United Kingdom…

HA loading is to be applied to each notional lane in loadcases 3 and 4. These loads are defined using the UK vehicle loading definitions supplied with LUSAS *Bridge*.

Lane load

- Select the **Lane load (HA load)** button.
- Select loading code **BD 37/88** and change the length to **20**
- Click the **OK** button to add the load dataset **Pch4 (HA BD37/88 20.0m x 3.5m Skew=0.0deg (Centre))** to the Treeview.

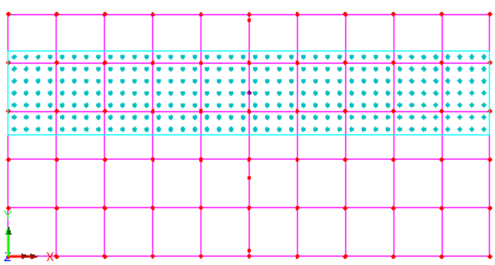
Knife edge load

- On the same dialog, select the **Knife edge load (KEL load)** button.
- Leave the notional width as **3.5** and the intensity as **120**
- Click the **OK** button to add the load dataset **Pch5 (KEL 120kN Width=3.5m Offset=0.0m Skew=0.0deg (Centre))** to the Treeview.

Abnormal load

- On the same dialog, select the **Abnormal load (HB vehicle)** button.
- With the axle spacing set to **6** and **45** units of HB load click the **OK** button to add the load dataset **Pnt6 (HB 6.0m spacing 45.0 units)** to the Treeview.
- Click the **Close** button to close the UK bridge loading dialog.

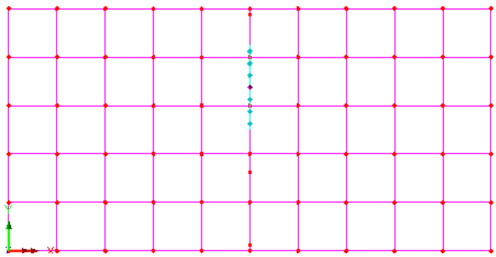
Assigning HA Loading



- Select the point defined at the centre of the upper notional lane.
- Drag and drop the dataset **Pch4 (HA BD37/88 20.0m x 3.5m Skew=0.0deg (Centre))** from the Treeview.
- Enter **HA upper** as the Loadcase, leave other values as their defaults, and click **OK.** The loading will be visualised.

- Select the point defined at the centre of the lower notional lane.
- Drag and drop the dataset **Pch4 (HA BD37/88 20m x 3.5m Skew=0 (Centre))** from the Treeview.
- Enter **HA lower** as the Loadcase, leave other values as their defaults, and click **OK**

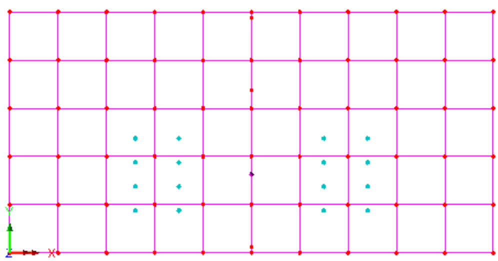
Assigning KEL Loading



- Select the point defined at the centre of the upper notional lane.
- Drag and drop the dataset **Pch5 (KEL 120kN Width=3.5m Offset=0.0m Skew=0.0deg (Centre))** from the Treeview.
- Enter **KEL upper** as the Loadcase and click **OK**
- Select the point defined at the centre of the lower notional lane.
- Drag and drop the dataset **Pch5 KEL 120kN Width=3.5m Offset=0 Skew=0 (Centre)** from the Treeview.
- Enter **KEL lower** as the Loadcase, leave other values as their defaults, and click **OK**

Assigning Abnormal HB Loading

For this example, abnormal HB loading is only assigned to the lower lane.



- Select the point defined at the centre of the lower notional lane.
- Drag and drop the dataset **Pnt6 (HB 6.0m spacing 45.0 units)** from the Treeview onto the selected point.
- Enter **HB lower** as the Loadcase, leave other values as their defaults, and click **OK**

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

Save the model

File
Save

Save the model file.

Running the Analysis

With the model loaded:

Open the Solve Now dialog, ensure **Analysis 1** is selected and press **OK** to solve.

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

If the analysis is successful...

Analysis loadcase results are added to the Treeview.

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

- **grillage.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- **grillage.mys** this is the LUSAS results file which is loaded automatically into the Treeview to allow results processing to take place.

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.

Rebuilding a Model

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

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

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 **grillage** and click **OK**

File
Script >
Run Script...

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

Rerun the analysis to generate the results.

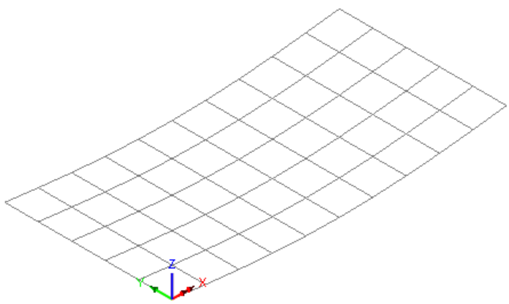
Viewing the Results

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

Deformed Mesh and Summary Plot

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

- In the Treeview right-click on **Dead load** and select the **Set Active** option.
- If present, turn off the display of the **Geometry, Attributes** and **Mesh** layers in the Treeview.
- The **Deformed mesh** layer should be turned on by default in the Treeview.. Double click its name and select the **Specify factor** option and enter **300** Click the **OK** button to display the deformed mesh for loadcase 1.



If necessary use the Isometric View button to rotate the model.

- It is good practice to step through each of the loadcases in the Treeview using the **Set Active** option to check each deformed shape looks correct for the supposed loading.

Defining Envelopes and Combinations

The design combination will consist of all dead loads and an envelope of all live loads factored by the appropriate adverse or relieving factor.

According to BS5400 part 1 two safety factors should be applied to adverse loading. γ_{f1} accounts for the uncertainty in the applied loading and γ_{f3} is a safety factor to allow for modelling inconsistencies / inaccuracies.

Loadcase name	**Adverse Factor**		**Relieving Factor**
	γ_{f1}	γ_{f3}	
Dead Load	1.15	1.10	**1.0**
Super Dead Load	1.75	1.10	**1.0**
HA alone	1.5 (*)	1.10	**0**
HA with HB	1.3 (*)	1.10	**0**

Table 1

(*) When designing to BD 37/88 the HA lane loading factors also include additional lane factors. For a two lane structure these are noted in the tables which follow.

Defining a Basic Load Combination 1

Analyses
Basic Combination…

A basic load combination to investigate HA and Knife Edge loads will be defined.

On the Basic Combination dialog:

[>>] Add loadcases (**3) HA upper, (4) HA lower, (5) KEL upper, (6) KEL lower**

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

Each loadcase selected then needs a corresponding lane factor to be specified.

- Update the **Factor** for each of the included loadcases to be as shown below.

Loadcase name	Load Factor		Lane Factor	Lane Factor to be used
	γ_{f1}	γ_{f3}		
HA upper	1.5	1.10	0.956	**1.6**
HA lower	1.5	1.10	0.956	**1.6**
KEL upper	1.5	1.10	0.956	**1.6**
KEL lower	1.5	1.10	0.956	**1.6**

Table 2

- Change the combination name to **HA + KEL both lanes**
- Click **OK** to save the combination definition.

Defining a Basic Load Combination 2

A basic load combination to investigate HA, HB and Knife Edge loads will also be defined.

Analyses
Basic Combination…

On the Basic Combination dialog:

>> Add result loadcases (**3) HA upper, (5) KEL upper, (7) HB lower**

Each loadcase selected needs the factor to be specified.

- Update the **Factor** for each of the included loadcases to be as shown below.

Loadcase name	Load Factor		Lane Factor	Lane Factor to be used
	γ_{f1}	γ_{f3}		
HA upper	1.3	1.10	0.956	**1.4**
KEL upper	1.3	1.10	0.956	**1.4**
HB lower	1.3	1.10	0.956	**1.4**

Table 3

- Change the combination name to **HA + KEL upper, HB lower**
- Click **OK** to save the combination definition.

Enveloping the Basic Live Load Combinations

Analyses
Envelope

On the Properties dialog:

>> Add combinations **(8) HA+KEL both lanes** and **(9) HA+KEL upper, HB lower**

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

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

Defining a Smart Combination

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

Note. For new models, by default, the smart combinations dialog is presented to allow input in terms of Beneficial/Adverse factors rather than Permanent/Variable factors.

Historically, for this example, Permanent and Variable factors have been entered as opposed to entering Beneficial and Adverse factors, so:

- In the treeview double-click the **Combination and envelope options** object.

- On the Combinations and Envelope Options dialog, uncheck the **Display beneficial/adverse factors** check box, and Click **OK**.

Analyses
Smart Combination…

On the Smart Combination dialog:

>> Add loadcase **(1)Dead load** and (**2)Superimposed DL** to the Included panel.

>> Add **(10)Live Load Envelope (Max)** and **(11)Live Load Envelope (Min)** to the Included panel.

Each loadcase/envelope selected needs the permanent and variable factors to be specified.

- Update the **Permanent Factor** for the Live Load Envelopes to be **0** and ensure the **Variable Factor** for all loadcases are as shown in the table.

Loadcase name	**Variable Factor**		**Permanent (relieving) Factor**	**Variable Factor to be used**
	γ_{f1}	γ_{f3}		
(1)Dead Load	0.15	0.10	**1.0**	**0.265**
(2)Superimposed DL	0.75	0.10	**1.0**	**0.925**
(10)Live Load Envelope (Max)	-	-	**0**	**1.0**
(11)Live Load Envelope (Min)	-	-	**0**	**1.0**

Table 4

Caution. In LUSAS, when inputting permanent and variable load factors:

- ❑ **Permanent load factor = Beneficial load factor**
- ❑ **Variable load factor = Adverse load factor - Permanent load factor.**

Note. In the preceding table the permanent factor is based upon the relieving factor from Table 1. The variable factor for Dead Load and for Superimposed Dead Load is based upon the product of the adverse factors for both from Table 1 minus the permanent (relieving) factor. The live load envelopes have already been factored in previous load combinations (Tables 2 and 3) and, as a result, only a unity factor is applied as a variable factor.

- Change the combination name to **Design Combination**

- Click **OK** to save the smart combination.

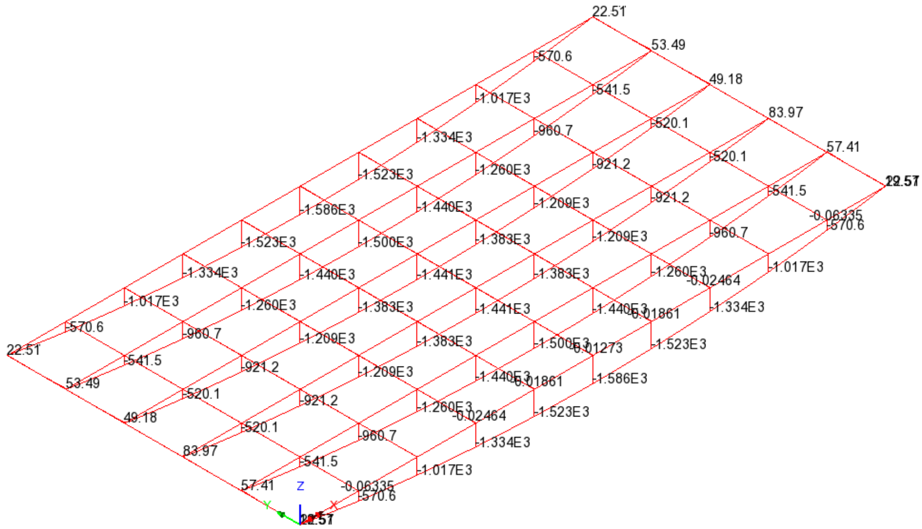
Bending Moment Diagram

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

- In the Treeview right-click on **Design Combination (Max)** and select the **Set Active** option.
- Select entity **Force/Moment** results of component bending moment **My**
- With no features selected, click the right-hand mouse button in a blank part of the graphics window and select the **Diagrams** option to add the diagram layer to the Treeview.

The diagram properties will be displayed.

- Select entity **Force/Moment** results of component bending moment **My**
- In the envelopes and combinations region, choose the option to **Show maximum only**
- Select the **Diagram Display** tab
- Select the **Label values** option.
- <u>Deselect</u> the **Use for labels** too option, so that labels are drawn in black.
- Set the **Number of significant figures** to **4**
- Click the **OK** button to display the bending moment diagram initially for the currently active Dead load loadcase.

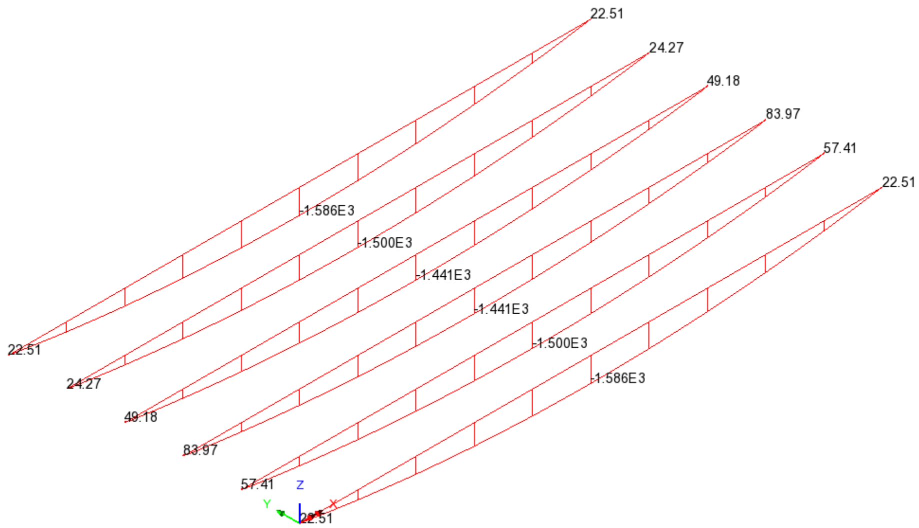


Note. When activating a smart combination the selected component is used to decide if the variable factor should be applied. (The variable component is only applied if the resulting effect is more adverse) Viewing results for a component other than the selected component will result in display of the associated values (coincident effects). When the results of an envelope or smart combination are printed the column used to compute the combination or envelope is denoted with an asterisk in the column header.

Selecting Members for Results Processing

Results are to be plotted for selected longitudinal members of the grillage only. The grillage wizard automatically creates groups which are useful in the results processing.

- In the Treeview select the **Longitudinal Beams** with the right hand mouse button and pick **Set as Only Visible**
- In the Treeview select **Edge Beams** with the right hand mouse button and pick **Visible**



Note. By default diagrams are drawn orientated according to element axes. As an alternative (if this model was ever to be viewed along Z-axis) they can be drawn 'flat to screen/page' by selecting this option on the Diagrams properties dialog

Page layout view

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

View
Page Layout Mode

- Switch to page layout view.

Tools
Annotation >
Window border

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

File
Page Setup

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

Tools
Annotation >
Window
Summary

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

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

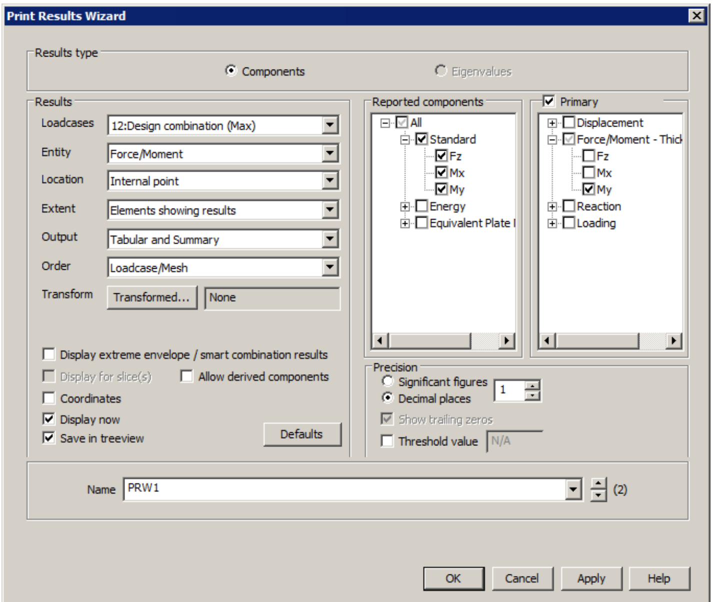
Note. The location of any model feature, element or node can be found by using the Advanced Selection facility. This can be used to find the location of the maximum and minimum results values since the element number is output in the window summary text.

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

Printing results for the current loadcase

Utilities
Print Results Wizard…

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

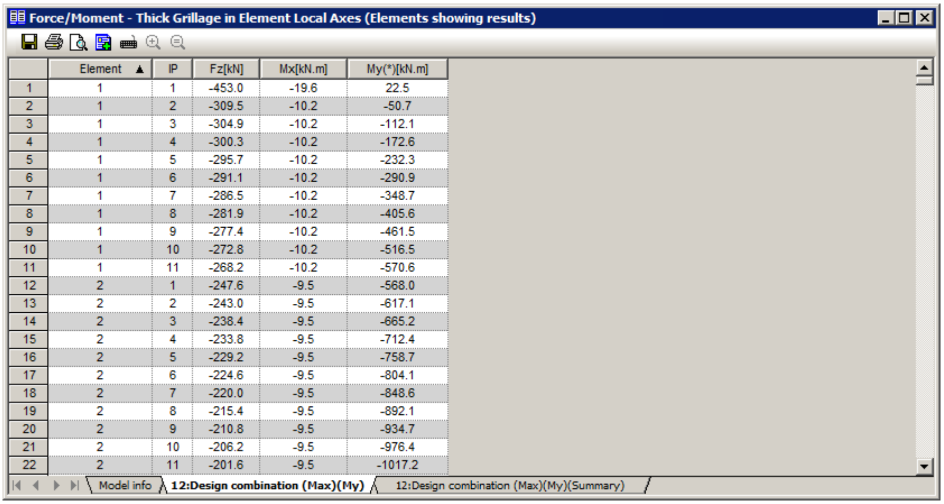


- Ensure **Design Combination (Max)** is chosen from the Loadcases drop-down.
- Select entity **Force/Moment**
- Select location **Internal Point**
- Leave all other Results panel settings as their default values (see dialog).

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

- In the Precision panel set the number of decimal places to **1**

- Ensure that the **Display now** check box is selected and click the **OK** button to display the results. A Print Results Wizard entry is also added to the Utilities Treeview.


Force/Moment - Thick Grillage in Element Local Axes (Elements showing results)

	Element ▲	IP	Fz[kN]	Mx[kN.m]	My(*)[kN.m]
1	1	1	-453.0	-19.6	22.5
2	1	2	-309.5	-10.2	-50.7
3	1	3	-304.9	-10.2	-112.1
4	1	4	-300.3	-10.2	-172.6
5	1	5	-295.7	-10.2	-232.3
6	1	6	-291.1	-10.2	-290.9
7	1	7	-286.5	-10.2	-348.7
8	1	8	-281.9	-10.2	-405.6
9	1	9	-277.4	-10.2	-461.5
10	1	10	-272.8	-10.2	-516.5
11	1	11	-268.2	-10.2	-570.6
12	2	1	-247.6	-9.5	-568.0
13	2	2	-243.0	-9.5	-617.1
14	2	3	-238.4	-9.5	-665.2
15	2	4	-233.8	-9.5	-712.4
16	2	5	-229.2	-9.5	-758.7
17	2	6	-224.6	-9.5	-804.1
18	2	7	-220.0	-9.5	-848.6
19	2	8	-215.4	-9.5	-892.1
20	2	9	-210.8	-9.5	-934.7
21	2	10	-206.2	-9.5	-976.4
22	2	11	-201.6	-9.5	-1017.2

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

Note. Table data can be sorted by selecting a column and choosing a sort option from the context menu for the selection.

Note. Print Results Wizard data can be added to a Model Report by pressing the **Add to Report** button at the top of the results listing. When done, each time the model report is generated the results included will be for the current state of the model at that time.

Saving printed results to a spreadsheet

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

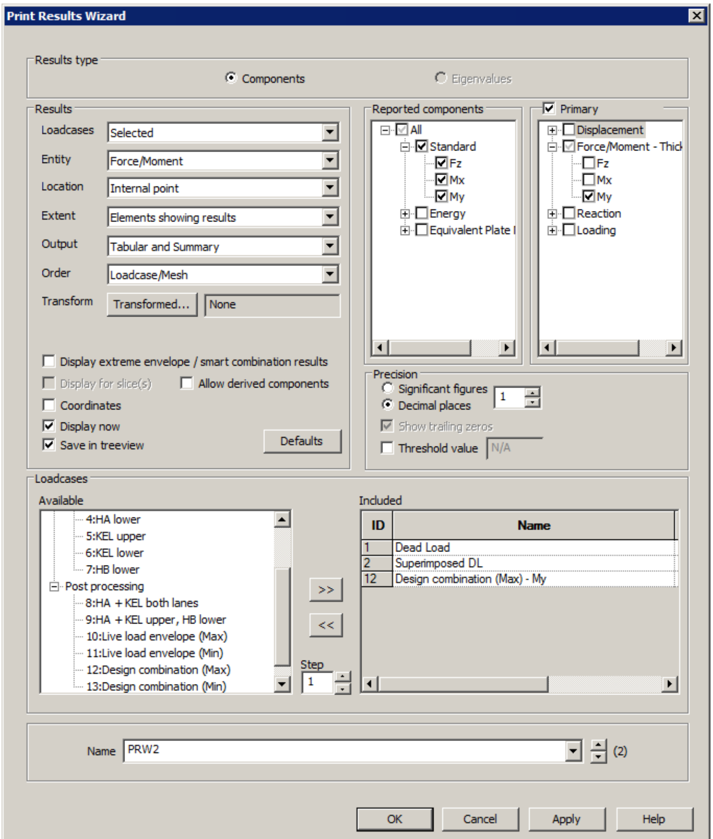
- Right-click inside the Printed Results window and select **Save as Microsoft Excel…**
- Enter a file name of **grillage_results**
- Ensure the save option **All tabs** is selected and click **Save**.

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

Printing results for selected loadcases

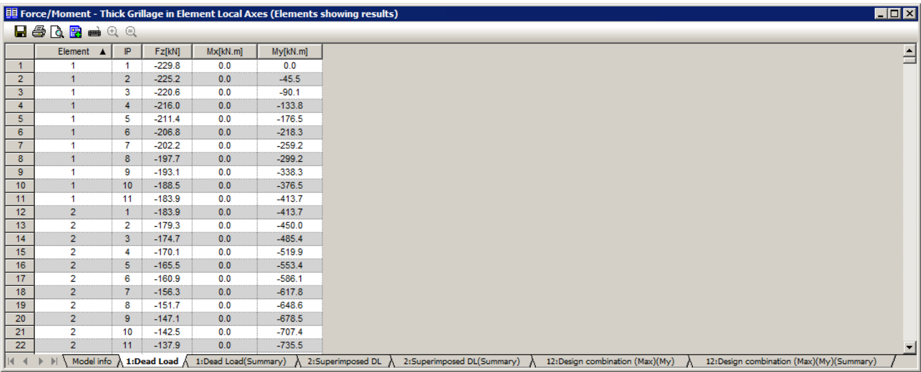
Utilities
Print Results Wizard…

Results values may be printed for more than one loadcase. To illustrate this use:



- Ensure **Selected** is chosen from the Loadcases drop-down.
- Select entity **Force/Moment**
- Select location **Internal Point**

- Leave all other Results panel settings as their default values.
- Ensure the **Primary** components check box is selected, expand the Force/Moment - Thick Grillage treeview entry and ensure that **My** is the primary results component selected.
- In the Precision panel set the number of decimal places to **1**
- In the Loadcases available panel select **Dead load**, **Superimposed DL**, and **Design Combination Max** loadcases and press the **Add to** button to add them to the Included panel
- Ensure that the **Display now** check box is selected and click the **OK** button to display the results.



	Element ▲	IP	Fz[kN]	Mx[kN.m]	My[kN.m]
1	1	1	-229.8	0.0	0.0
2	1	2	-225.2	0.0	-45.5
3	1	3	-220.6	0.0	-90.1
4	1	4	-216.0	0.0	-133.8
5	1	5	-211.4	0.0	-176.5
6	1	6	-206.8	0.0	-218.3
7	1	7	-202.2	0.0	-259.2
8	1	8	-197.7	0.0	-299.2
9	1	9	-193.1	0.0	-338.3
10	1	10	-188.5	0.0	-376.5
11	1	11	-183.9	0.0	-413.7
12	2	1	-183.9	0.0	-413.7
13	2	2	-179.3	0.0	-450.0
14	2	3	-174.7	0.0	-485.4
15	2	4	-170.1	0.0	-519.9
16	2	5	-165.5	0.0	-553.4
17	2	6	-160.9	0.0	-586.1
18	2	7	-156.3	0.0	-617.8
19	2	8	-151.7	0.0	-648.6
20	2	9	-147.1	0.0	-678.5
21	2	10	-142.5	0.0	-707.4
22	2	11	-137.9	0.0	-735.5

Save the model

File
Save

Save the model file.

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

This completes the example.