Wood-Armer Bridge Slab Assessment

For LUSAS version:	21.1
For software product(s):	LUSAS *Bridge*.
With product option(s):	None.

Description

A Wood Armer slab assessment is to be carried out on a 15m long, 9.8m wide single span, reinforced concrete bridge deck. The deck has a skew angle of 11.3 degrees and is 0.9m deep.

A load combination comprising self weight, upper and lower lane loads, and upper and lower knife edge is to be created from the results obtained. Slab reinforcement is to be arranged orthogonally and has capacities of 1700 and 300 kNm in the chosen Mx and My directions.

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

Note. In order to create a generic example, no reference is made to any particular design code or loadcase type.

Keywords

2D, Simple Slab, **Skew Angle**, **Wood-Armer Reinforcement**, **Wood-Armer Assessment**, **Safety Factors**, **Self Weight, General Loading**, **Knife Edge Loading**, **General Patch Loading**, **Combination**, **Contour Plotting**, **Display Peak Values.**
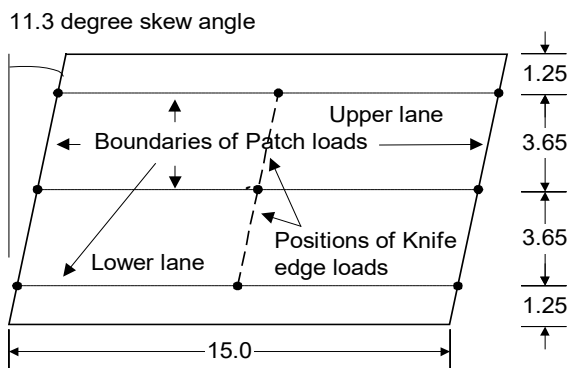
Associated Files

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

❑ **wood_armer_modelling.lvb** carries out the modelling of the example.

Modelling

Note. Model and results images included in this example were captured in an earlier version and may differ in appearance from those seen and obtained from this version.

Running LUSAS Modeller

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

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

Creating a new model

- Enter the file name as **wood_armer**

- Use the default User-defined working folder.

- Select Analysis type to be **Structural**

- Select Analysis Category to be **2D Grillage/Plate**

- Select model units of **kN,m,t,s,C**.

- Ensure the timescale units are **Seconds**

- Select a Startup template of **2D Plate**

- Ensure the Layout grid is set as **None**

- Enter the title as **Wood Armer Slab Example** and 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 was done.
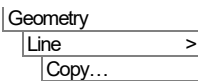
Feature Geometry
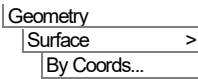
Enter x and y coordinates of (**0, 0**), **(15, 0**), (**16.96, 9.8**) and (**1.96, 9.8**) and click **OK** to define the slab as a single Surface.

Next, the edge of carriageway and lane widths will be defined by copying the lower Line of the main span.

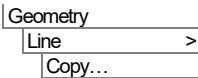- Select the lower horizontal Line of the model.

Enter the **X** translation as **0.25** [derived from 1.25*tan(11.3°]

- Enter the **Y** translation as **1.25** metres and click the **OK** button.

The edge of the lower carriageway will be drawn.
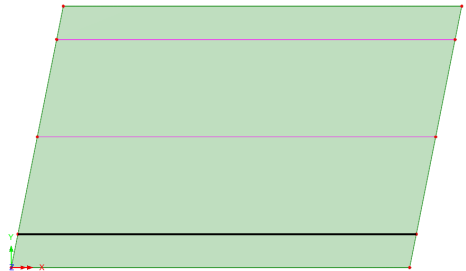
To define 2 notional lane widths of 3.65 metres:
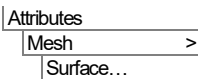
- Select the Line just drawn.

Enter the **X** translation as **0.73** [derived from 3.65*tan{11.3°}]

- Enter the **Y** translation as **3.65** metres.

- Enter the number of copies as **2**

- Click the **OK** button to finish.

Meshing

- Select **Thick plate**, **Quadrilateral**, elements with **Linear** interpolation.

- Deselect the automatic divisions option and enter **10** divisions in the local **X** direction and **7** divisions in the local **Y** direction.

- Enter the attribute name as **Thick Plate** and click the **OK** button to finish.

LUSAS will add the Surface mesh attribute to the Treeview.

- Select the whole model using the **Ctrl** and **A** keys together**.**

- Drag and drop the Surface mesh attribute **Thick Plate** from the Treeview onto the selected features.

Note. At any time the mesh (and other features) displayed in the model view window may be turned on or off by right-clicking on the layer name in the Treeview and selecting the Display option.

- Turn off the display of the **Mesh** as just described.

Geometric Properties

To illustrate how units can be converted on-the-fly:

Attributes
  Geometric   >
    Surface…

- Open the Units Convertor and Variation Utility from the Thickness textbox.

- Convert from **900**, selecting **mm** to convert 900mm to model units. Press **OK** to return.

- Enter the attribute name as **Thickness 0.9**. Click **OK** to add surface geometry attribute to the Treeview.
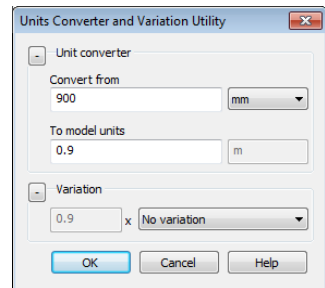
- Select the whole model (press **Ctrl + A** keys)

- Drag and drop the geometry attribute **Thickness 0.9** from the Treeview onto the selected features. Click **OK**.

Select the fleshing on/off button to turn-off the geometric visualisation. If at any time during the example you wish to visualise the geometry select this button.

Material Properties

Because a startup template of 2D Plate was chosen earlier, an ungraded concrete material is present in the Treeview.

- With the whole model selected, drag and drop the material attribute **Concrete (Ungraded | Concrete)** from the Treeview onto the selected features, ensuring that **Assign to surfaces** is selected click **OK**
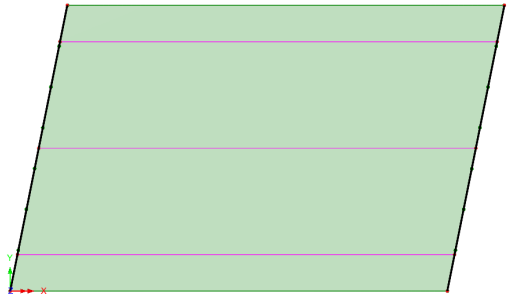
Supports

- Select the **Fixed** for **translation in Z** option, enter the name as **Fixed in Z** and click **OK** to add the support attribute to the 🔴 Treeview.

Assigning the Supports

- Select the 2 inclined Lines at either end of the slab. (hold the Shift key down to add to the first selection).

- Drag and drop the support attribute **Fixed in Z** from the 🔴 Treeview onto the selected features.

- Ensure **Assign to lines** and **All analysis loadcases** are selected and click **OK**

The supports will be visualised.

Loading

Five loadcases will be considered, with an additional two combinations of these loads created at the results processing stage.

Modifying the Geometry to assign loading

To position knife edge loads additional Points are required at mid-span positions.

- Select the 3 horizontal Lines that define the traffic lane boundaries. (Hold the **Shift** key down to add to the selection)

- Select **Use same divisions for all lines** and enter the number of divisions required for all as **2**

- Ensure the **Delete original geometry after splitting** option is selected and click **OK**
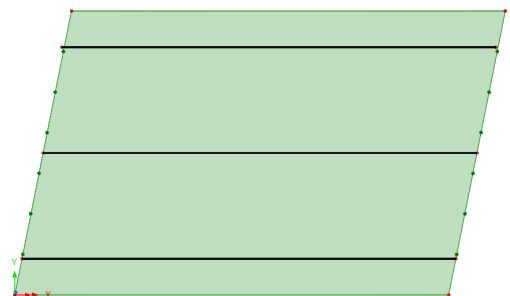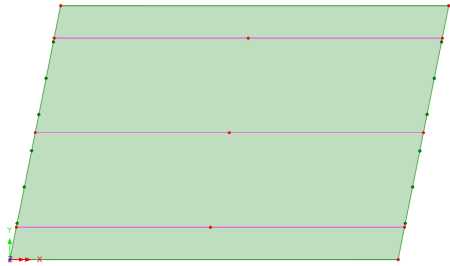
Additional Points will be created at mid-span and existing Lines will be broken into 2 new Lines. The new Points will be used later to define the position of the mid-span knife edge loads for each traffic lane.


Loadcase 1 - Self weight

- Select load type **Displacement, velocity, acceleration, body force** option

- Then select **Body Force** and click **Next**

- Define the linear acceleration due to gravity in the **Z** direction as **-9.81**

- Enter the attribute name as **Self Weight** and click **Finish**

- With the whole model selected, drag and drop the loading attribute **Self Weight** from the 🔵 Treeview onto the selected features ensuring the loading is assigned to Surfaces as **Loadcase 1** and click **OK**

The self weight loading will be displayed.

- In the 🔲 Loadcases Treeview expand **Analysis 1** then right-click on **Loadcase 1** and select the **Rename** option.

- Rename **Loadcase 1** to **Self Weight**
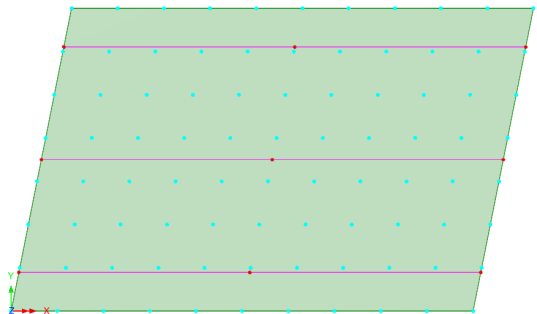
Loadcase 2 - Lane load (lower lane)

- Select the first point, hold down the **Shift** key and select point 2, 3 and 4 in the order shown to define the patch area.

- Select load type **Discrete loads**, then select **Discrete patch** and click **Next**

3. Select this Point

4. Select this Point

1. Select this Point   2. Select this Point

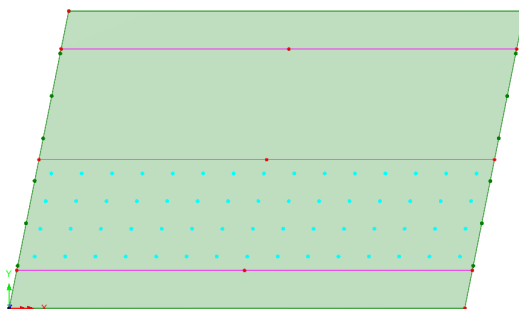The coordinates of the Points selected will be inserted into the coordinate fields.
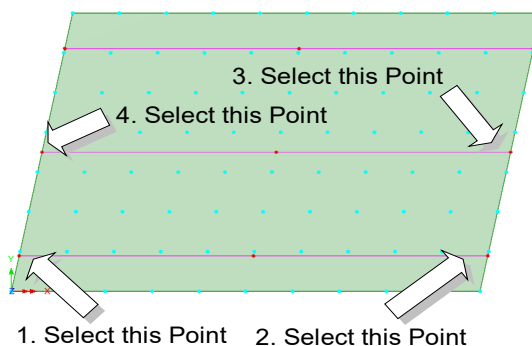
- Enter a load value of **-15** (kN/m^2) for each coordinate.

- De-select the 'Use default' button in the Patch load divisions panel and enter **15** for the number of divisions in the x direction and **0** for the number of division in the y direction, so that the number of loading points in the patch y direction will be computed automatically from the aspect ratio of the patch.

- Enter the attribute name as **Lane load (lower)** and click **Finish**

Note. The order in which the Points are selected determines the local x and y directions of the patch load. The local x direction is from the first Point to the second Point selected. The local y direction is from the second to the third Point.

- Select the Point at the origin 0,0 of the structure.

- Drag and drop the loading attribute **Lane load (lower)** from the Treeview onto the selected Point.

- Ensure **Options for loads outside search area** is set to **Exclude all load**

- Enter the loadcase as **Lane load (lower)** with a load factor of **1** and click **OK**

The loading will be displayed.

Note. Because the coordinates of the patch load were taken from the coordinates of the Points, the patch load must be assigned to the Point at the origin of the structure.

Loadcase 3 - Lane load (upper lane)

- Select the Points in the order shown to define the patch area for the upper lane.

- Select load type **Discrete loads**, then select the **Discrete Patch** option and click **Next**

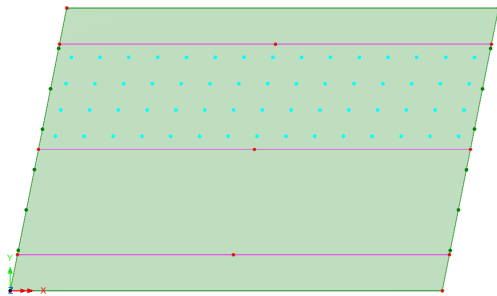The coordinates of the Points selected will be inserted in the coordinate fields.
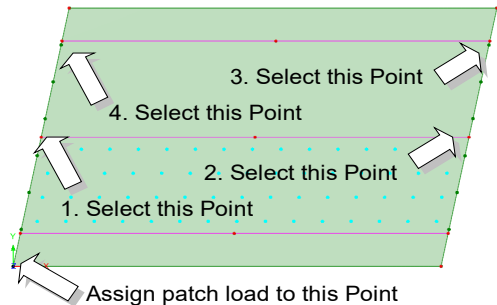
- Enter a load value of **-15** (kN/m^2) for each coordinate.

- De-select the Use default button in the Patch load divisions panel and enter **15** in the x direction and **0** in the y direction.

- Enter the attribute name as **Lane load (upper)** and click **Finish**

- With the Point at the origin of the structure selected, drag and drop the loading attribute **Lane load (upper)** from the Treeview onto the selected Point.

- Ensure **Options for loads outside search area** is set to **Exclude all load**

- Enter the loadcase as **Lane load (upper)** with a load factor of **1** and click **OK**
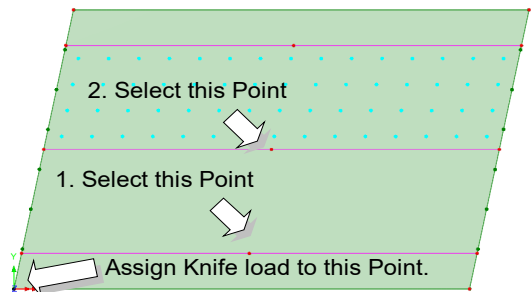
Loadcase 4 - Knife edge load (lower lane)

- Select the Points in the order shown to define the knife edge load position for the lower lane.

- Select load type **Discrete loads,** then select the **Discrete Patch** option and click **Next**

- Select the **Straight** option.

The coordinates of the Points selected will be inserted in the coordinate fields.
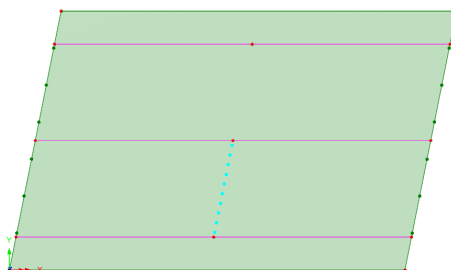
- Enter a load value of **-32.24** (kN/m) for each coordinate.

- De-select the Use default button in the Patch load divisions panel and enter **15** in the x direction.

- Enter the attribute name as **Knife edge load (lower)** and click **Finish**

Note. The order in which the Points are selected determines the local X direction of the knife edge load. The Local X direction is from the first Point to the second Point selected.
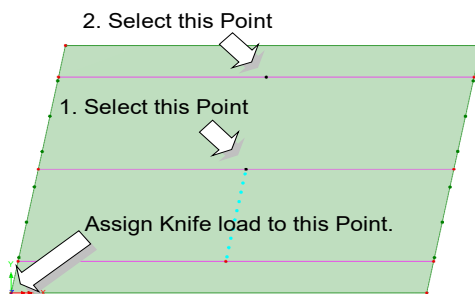
- With the Point at the origin selected, drag and drop the loading attribute **Knife edge load (lower)** from the Treeview onto the selected Point.

- Enter the loadcase as **Knife edge load (lower)** with a load factor of **1** and click **OK**

Load case 5 - Knife edge load (upper lane)

- Select the Points in the order shown to define the knife edge load position for the upper lane.

- Select load type **Discrete loads,** then select the **Discrete Patch** option and click **Next**

The coordinates of the Points selected will be inserted in the coordinate fields.

Attributes
  Loading…

2. Select this Point

1. Select this Point

Assign Knife load to this Point.

- Enter a load value of **-32.24** (kN/m) for each coordinate.

- De-select the Use default button in the Patch load divisions panel and enter **15** in the x direction.

- Enter the attribute name as **Knife edge load (upper)** and click **Finish**

- With the Point at the origin selected, drag and drop the loading attribute **Knife edge loading (upper)** from the Treeview onto the selected Point.

- Enter the loadcase as **Knife edge load (upper)** with a load factor of **1** and click **OK**

Visualising Loadcases

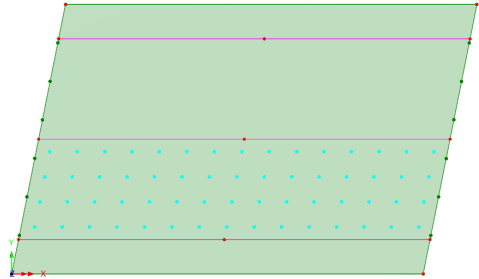Loadcases can be visualised at any time by activating each loadcase in the 🕐 Treeview.

- From the 🕐 Treeview select the **Lane load (lower)** loadcase, click the right-hand mouse button and select the **Set Active** option.

Saving the model

File
Save

💾 Save the model file.

Running the Analysis

With the model loaded:

▤ Press the **Solve Now** button to run the analysis.

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, these files will be created in the LUSASFiles32\<model_name> folder:

- ❑ **wood_armer.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- ❑ **wood_armer.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. You may download this file from the user area of the LUSAS website.

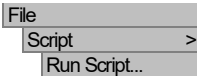❑ **wood_armer_modelling.lvb** carries out the modelling of the example.
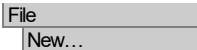
File
New…

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 **wood_armer**

- Use the default **User-defined** working folder.

- Ensure an Analysis category of **2D Inplane** is set.

- Click the **OK** button.

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

File
Script >
Run Script...

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

Rerun the analysis to generate the results.

Viewing the Results

Combinations

A combination will be created to apply the ultimate limit state (ULS) factors and lane factors to loadcases 1 to 5. Typical ULS and lane factors can be found in relevant design codes.

Dead load factor ULS combination

- ❑ Dead load factor $\gamma_{fl} = 1.15$
- ❑ Additional factor $\gamma_{f3} = 1.1$

Loadcase	Factor Calculation	Factor
Self weight	$\gamma_{fl} \times \gamma_{f3}$	**1.265**
Lane load (lower)	$\beta_1 \times \gamma_{fl} \times \gamma_{f3}$	**1.65**
Lane load (upper)	$\beta_2 \times \gamma_{fl} \times \gamma_{f3}$	**1.65**
Knife edge load (lower)	$\beta_1 \times \gamma_{fl} \times \gamma_{f3}$	**1.65**
Knife edge load (upper)	$\beta_2 \times \gamma_{fl} \times \gamma_{f3}$	**1.65**

Live load factors for ULS combination

- ❑ Lane factors $\beta_1 = 1.0$ and $\beta_2 = 1.0$
- ❑ Live load factor $\gamma_{fl} = 1.5$
- ❑ Additional factor $\gamma_{f3} = 1.1$

Defining a Combination

Combinations can be created to view the combined effects of multiple loadcases on the structure. A combination dataset **Combination 1** is to be created in the 🔍 Treeview

Analyses
> Basic Combination

The combination properties dialog will appear. All 5 loadcases should be included in the load combination panel.

- Select loadcase **Self Weight** hold the **Shift** key down, scroll down, and select **Knife edge load (upper).**

- Click the ⟫ button to add the loadcases to the combination dataset.

✏️ **Note.** To obtain the correct effect from the combined loads in this example the Combination should only include one occurrence of each loadcase.

Assigning load factors

- In the Included panel on the Combination dialog, select the **Self Weight** loadcase and enter load factor of **1.265** in the load factor text box.

- For each of the other 4 loadcases, select each loadcase in turn and enter a factor of **1.65**

- Ensure the **Method** drop list is set to **Factored**

- Click **OK** to create the combination dataset.

Selecting Loadcase results

- In the 🔍 Treeview right-click on **Combination 1** and select the **Set Active** option.

- If present in the 🔲 Treeview turn off the **Geometry** and **Attributes** layers.

- Turn on the display of the **Mesh** layer in the 🔲 Treeview.

Contour Plots

- With no features selected click the right-hand mouse button in a blank part of the graphics window and select the **Contours** option to add the contours layer to the Treeview.

The contour properties dialog will be displayed.

Moments in the X direction

- Select **Force/Moment** contour results.

- Select the **MX** component.

- Click the **OK** button to display contours of MX for Combination 1.

Marking Peak Values

- With nothing selected click the right-hand mouse button in a blank part of the graphics window and select the **Values** option to add the values layer to the 🔲 Treeview.

The properties dialog of the values layer will be displayed.
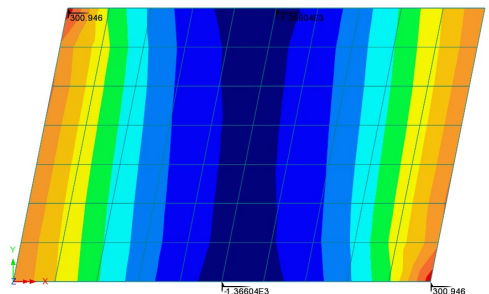
- Select the **Force/Moment** entity.

- Select the **MX** component.

- Select the **Values Display** tab and set **Maxima** and **Minima** values to display the moments for the top **0.1**% of results.

- Click the **OK** button to redisplay the contours with peak values marked.

Moments in the Y direction

- In the ▣ Treeview double-click on the **Contours** layer.

- Select the **Force/Moment** entity.

- Select the **MY** component.

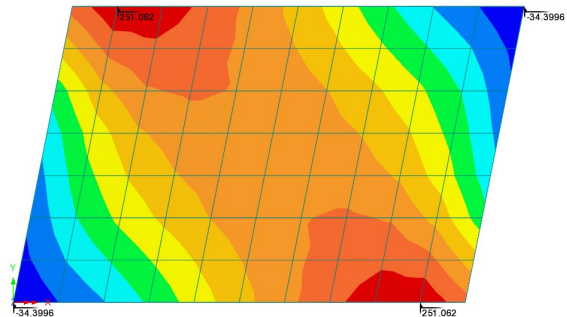- Click the **OK** button to display contours of MY for Combination 1.
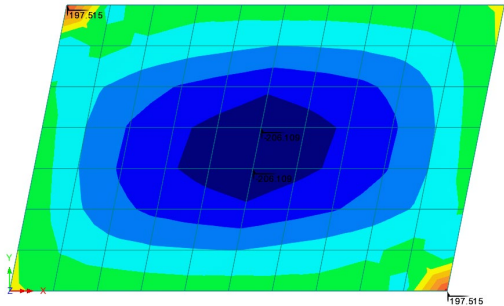
Note. The values layer is currently displaying results for MX.

- Select **Yes** to change the values layer results to match those of the contours layer.

Moments in XY direction

- In the ▣ Treeview double-click on the **Contours** layer.

- Select the **Force/Moment** entity.

- Select the **MXY** component. Click the **OK** button.

- Select **Yes** to change the values layer results to match those of the contours layer.

Wood-Armer Reinforcement Moments

Note. For the following section, it is assumed that you are familiar with the theory of Wood-Armer as an explanation is beyond the scope of this example. If additional information is required consult the LUSAS Theory Manual. Initially, Wood-Armer reinforcement moments Mx(B) in the bottom surface of the slab will be calculated.

Define and assign a Wood-Armer attribute

The Wood Armer attribute dialog allows data required to calculate Wood Armer moments to be defined and assigned to the model.

Attributes
Wood Armer…

- Leave all the default settings in the dialog and click OK to add the Wood Armer attribute to the ⊙ Treeview.

- In the ▭ Treeview turn back on the geometry layer

- With the whole model selected, drag and drop the Wood Armer attribute **WdAmr1** from the ⊙ Treeview onto the selected features ensuring the attribute is assigned to Surfaces and click **OK**

- A new entity "Wood Armer – Thick Plate" is now added to the results entities.

- Click in a blank area of the view window to deselect the features

Wood-Armer Moments in X direction of the bottom of the slab

- In the ▭ Treeview double-click on the **Contours** layer.

- Select the **Wood Armer** entity.

- Select the **MX(B)** component to view contour of Wood Armer moments in the X direction for the bottom of the slab. Click the **OK** button.

- Select **Yes** to change the values layer results to match those of the contours layer.

From the resulting Wood-Armer plot it can be seen that the maximum value of MX(B) is 1543 kNm. This value is compared with the maximum slab capacity (1700 kNm) to determine the safe load carrying capacity. In this case the slab passes the assessment.
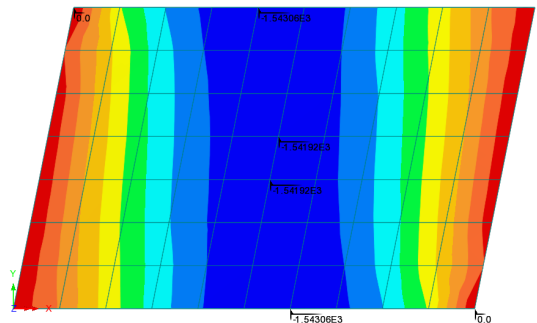
### Wood Armer Moments in Y direction of the bottom of the slab

The Wood Armer moments in the direction of the reinforcement on the bottom Surface MY(B), are now to be examined.

- In the 🔲 Treeview double-click on the **Contours** layer.

- Select the **Wood-Armer** entity.

- Select the **MY(B)** component to view contours of Wood Armer moments in the Y direction for the bottom of the slab.

- Click the **OK** button.

- Select **Yes** to change the values layer results to match those of the contours layer.

The plot indicates a maximum applied sagging Wood Armer moment MY(B) of 390 kNm This exceeds the slab capacity of 300 kNm and therefore this slab fails assessment.

✏️ **Note.** Normally the analysis would be continued to calculate the reduction in live load that would be necessary to obtain a situation where the applied moment MY(B) equals the capacity of the slab MY*(B). This would result in a restriction in the load carrying capacity of the slab.
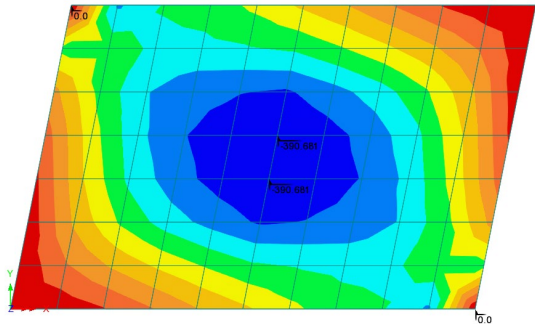
Wood-Armer - Discussion

It is generally accepted that the Wood-Armer equations when used in the assessment of the load carrying capacity of slabs provide a safe but conservative estimate of the structural capacity. It is possible to obtain significant improvements using alternative equations based on the fundamental principles of Wood-Armer.

✏️ **Note.** A detailed explanation of the modified equations is beyond the scope of this example however the user may find it beneficial to consult the following references:

- ❑ **Concrete Bridge Design to BS5400.** L.A Clark. Published by Construction Press. Appendix A. Equations For Plate Design.
- ❑ **The Assessment of Reinforced Concrete Slabs.** S.R Denton, C.J Burgoyne. The Structural Engineer. Vol. 74. No. 9. May 7.

In outline, the method adopts a safety factor approach, where the user inputs the slab capacity of the reinforcement and the reinforcement skew angle. Since the slab can potentially fail in flexure about any axis in the plane of the slab, the method examines

the applied moment field (Mx, My, Mxy) against the moment capacity field (Mx*, My*, Mxy*) for all possible reinforcement skew angles.

The method returns minimum safety factors for top (hogging) and bottom (sagging) reinforcement for each nodal position.

Note. Top and bottom safety factors are possible at any single position due to the application of mixed moment fields.

Wood-Armer Assessment

- In the 🔵 Treeview double click the WdArmr1 attribute.

- Check on the option to **Display assessment utilisations**

- Enter the Resistive moment in the x direction (top) as **1700**

- Enter the Resistive moment in the y direction (top) as **300**

- Enter the Resistive moment in the x direction (bottom) as **1700**

- Enter the Resistive moment in the y direction (bottom) as **300**

- Click the **OK** button to save the attribute.

- In the 🔲 Treeview double-click on the **Contours** layer and select the **Wood Armer** entity.

- Select the **UtilM(B)** component to select contours of Utility factors in the bottom of the slab.

- Click the **OK** button to display contours of safety for the moment capacities entered.

Note. Wood Armer skew angles are measured anticlockwise from the x-axis towards the y-axis.

Combination 1
Entity: Wood-Armer - Thick Plate
Component (Averaged nodal): UtilM(B)
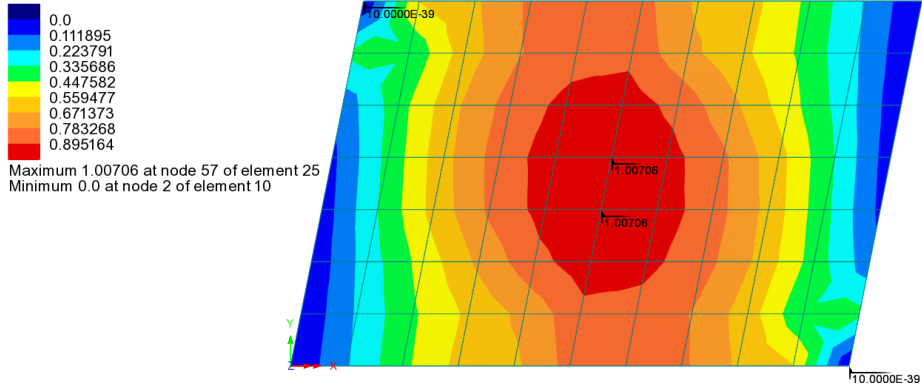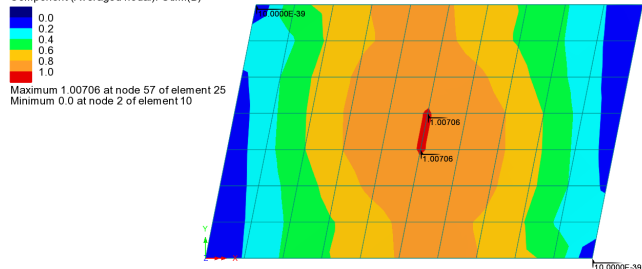
Changing the Contour range

If a utility factor plot shows values of less than 1.0, the slab is deemed to satisfy the reinforcement criteria and no live load reduction is necessary.

To help clarify the region of the slab that has failed the assessment the contour range will be modified.

- In the ⊡ Treeview double-click on the **Contours** layer and select the **Appearance** tab.

- For the **Classic** contour type, press the **Set** button and set the contour **Interval** to be **0.2** and the **Value to pass through** to be **1**

- Click the **OK** button to finish.

Contours of utility factors from 0 to 1 will be displayed showing a small region in the centre of the slab that has failed the assessment according to the moment capacities used.

This completes the example.