Simple Building Slab Design

For LUSAS version:	18.0
For software product(s):	LUSAS *Civil & Structural* or LUSAS *Bridge.*
With product option(s):	None.
Note: The example as written exceeds the limits of the LUSAS Teaching and Training Version. However, by not increasing the default mesh density to 8 divisions per line (where shown) the analysis can be run using a default of 4 divisions per line.	

Description

Four panels of a concrete slab supported by a wall, columns and a lift shaft are to be analysed, with reinforcement areas computed for bending moment only. Shear and displacement checks need to be carried out separately. The geometry of the slab is as shown.

The slab is subjected to self-weight and a live load.

The units of the analysis are N, m, kg, s, C throughout.

Objective

❑ To produce areas of reinforcement under ultimate loads and check crack widths under service loads

Keywords

2D, Plate, Slab Design, Holes, Reinforcement, Wood-Armer, RC Design, Steel Area, Load Combinations, Smart Combinations, SLS, ULS, Beneficial, Adverse, Cracking

---
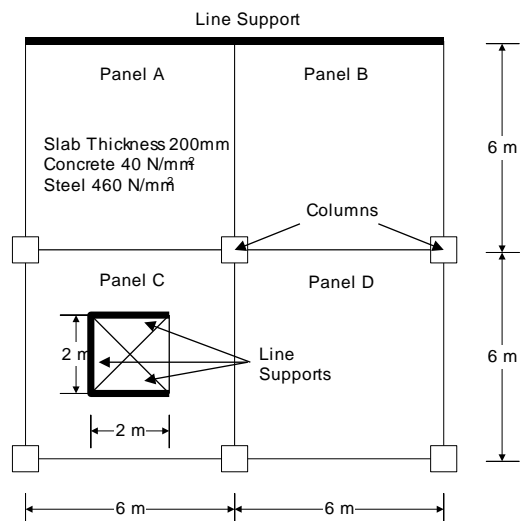
Associated Files

❑ **slab_design_modelling.vbs** carries out the modelling of this 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.

Creating a new model

• Enter the file name as **Slab_Design**

• Enter a file path for the working folder.

• Set the Analysis type to be **Structural**

• Set the Analysis Category to be **2D Grillage/Plate**

• Select model units of **N,m,kg,s,C**

• Ensure the timescale units are **Seconds**

• Select the Startup template **None** from those available in the drop-down list.

• Enter the title as **Slab Design Example**

• Leave Layout grid set to **None** and click the **OK** button.

The Attributes 🔗 treeview will be populated with potentially useful basic line mesh, geometric and support entries appropriate to the settings made on the new model dialog.

Note. The Analysis category determines which elements will be available for modelling. Here we are only interested in the out-of-plane bending effects in a flat slab and therefore we can approximate this behaviour with 2D plate elements.

Note. Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.
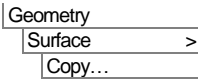
Feature Geometry
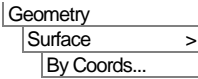
Enter coordinates of (**0, 0**), to define the lower left-hand area of slab, and using the Tab key to move to the next entry field enter coordinates of (**6, 0**), (**6, 6**) and (**0, 6**).

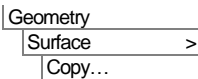- With all the coordinates entered click the **OK** button to create a surface.
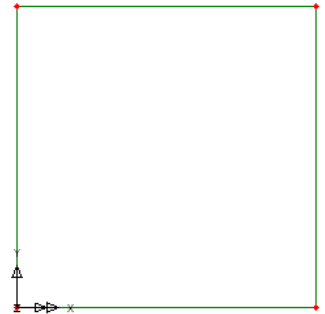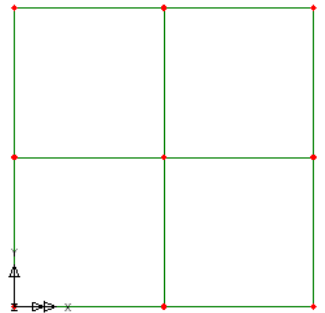
- Select the newly created Surface.

Copy the selected surface with an X translation of **6**.

- Click the **OK** button to create the new Surface.

- Use **Ctrl + A** keys together to select the whole model.

Copy the selected surfaces with a Y translation of **6**.

- Click the **OK** button to define two new Surfaces.

Defining a Hole in a Slab

Next, a hole representing the lift shaft needs to be defined. This is done by first creating a surface representing the extent of the lift shaft and then selecting both the surrounding and inner surface to define the hole.
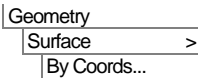
Enter coordinates of (**2, 2**), (**4, 2**), (**4, 4**) and (**2, 4**) and click **OK** to define the extent of the hole.

- Select the lower left Surface and then the Surface representing the lift shaft. (Use the **Shift** key to pick the second Surface to add to the selection)

- Ensure **Delete geometry defining holes** is selected A new singular Surface will be created.
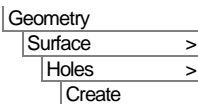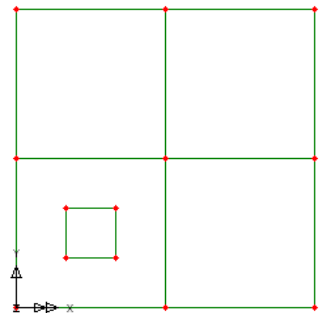
This new surface containing the hole can be seen /

checked by clicking in a blank part of the Graphics Window and then re-selecting the lower-left hand surface.

Note. It is normally good practice to ensure that the orientation of surface axes (and hence mesh element orientation) is consistent throughout the model. However, plate elements, as used in this example, produce results based upon global axes and as such ignore inconsistent element axes.

Meshing

By default lines normally have 4 mesh divisions per line. If you are using the Teaching and Training version this default value should be left unaltered to create a surface element mesh within the limit available. This will give a coarser mesh and correspondingly less accurate results will be obtained. Otherwise, for this example, and to give greater accuracy, 8 divisions per line will be used.

File
  Model Properties…

- Select the **Meshing** tab and set the default number of divisions to **8** and click the **OK** button.

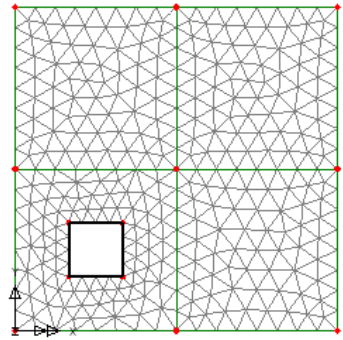Attributes
  Mesh   >
    Surface...

- Select the element type as **Thick Plate**, the element shape as **Triangle** and the interpolation order as **Quadratic**. Select an **Irregular mesh** and ensure the **Element size** is deselected. This forces the number of default line mesh divisions to be used when meshing the surfaces. Enter the attribute name as **Thick Plate** and click the **OK** button.

- With the whole model selected drag and drop the mesh attribute **Thick Plate** from the Treeview onto the selected Surfaces

In the vicinity of the lift core less elements are required.
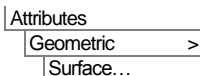
Attributes
  Mesh   >
    Line...

- Select the element type as **None** and enter the Number of divisions as 4. Enter the attribute name as **Divisions=4** and the **OK** button

- Select the 4 lines defining the lift core and drag and drop the line mesh attribute **Divisions=4** from the Treeview onto the selected Lines.

In this manner the mesh density on slabs can be varied according to the levels of detail required.

Geometric Properties

- Enter a thickness of **0.2**. Enter the attribute name as **Thickness 200mm** and click the **OK** button.

- With the whole model selected drag and drop the geometric attribute **Thickness 200mm** from the 🔧 Treeview onto the selected Surfaces.

Geometric properties are visualised by default.

- In the 🔲 Treeview drag and re-order the layers so that the **Attributes** layer is at the top of the Treeview, followed by the **Mesh** layer, followed by the **Geometry** layer. Delete the Utilities and Deformed mesh layers if present by right clicking and selecting **Delete**.

⚡ Select the fleshing on/off button to turn-off the geometric visualisation.

Material Properties
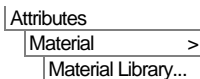
- Select material type **Concrete**, region **Europe**, standard **EN1992-1-1:2004/2014** and grade **C40/50** and click **OK** to add the material attribute to the 🔧 Treeview.

- Select the whole model and drag and drop the material attribute **Iso1 (C40/50 Concrete EN1992-1-1:2014)** from the 🔧 Treeview onto the selected Surfaces.
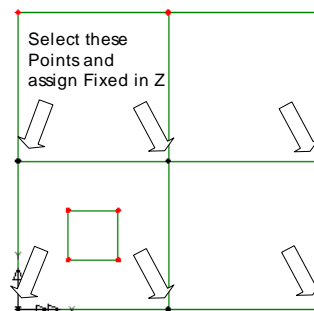
Supports
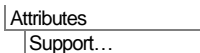
- Select **Fixed** for **Translation in Z** and enter the attribute name **Fixed in Z** and click **OK**.

For clarity the mesh layer is not shown on these diagrams.

- Select the 6 points where the columns are located.

- Assign the supports by dragging and dropping the support attribute **Fixed in Z** from the 🔧 Treeview and assign to **All analysis loadcases** by clicking the **OK** button.

Select these Points and assign Fixed in Z

- Select the 2 Lines representing the line support and drag and drop the support attribute **Fixed in Z** from the 🔵 Treeview and assign to **All analysis loadcases** by clicking the **OK** button.
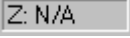
- Select only the top, bottom, and left-hand lines defining the lift shaft and drag and drop the support attribute **Fixed in Z** from the 🔵 Treeview and assign to **All analysis loadcases** by clicking the **OK** button.

Select these 2 Lines and assign Fixed in Z

Select these 3 Lines and assign Fixed in Z
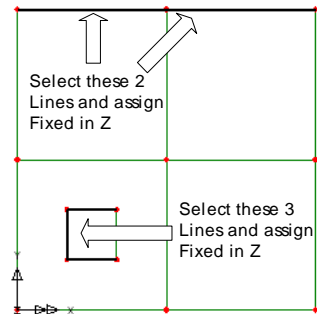
⊕ Select the isometric view button to check that all supports have been assigned correctly.

Z: N/A Click this part of the status bar to view the model from the Z direction again.

Loading
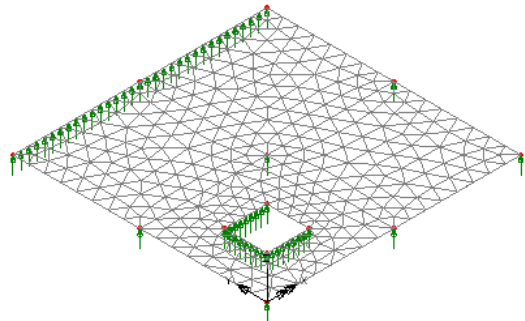
Attributes
  Loading…

- Select **Body Force** and click **Next**.

- Enter an acceleration of **-9.81** in the Z direction.

- Enter an attribute name of **Self Weight** and click the **Finish** button.

Attributes
  Loading…

- Select the **Global Distributed** radio button and click **Next**.

- Select the **Per unit area** option.

- Enter **-5000** in the **Z** Direction.

- Enter the attribute name as **Imposed Load 5kN/m2** and click the **Finish** button.

Now the dead and live loading needs to be assigned to each slab panel of the building. This is done using separate load cases so that the loadcases can be combined to determine the most adverse effects.
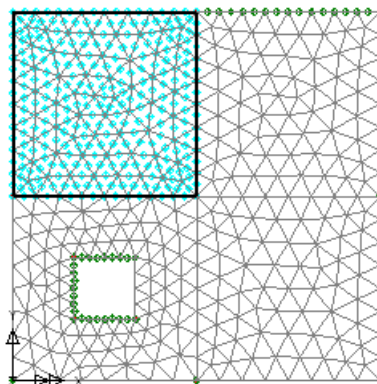
- In the Analyses 🕒 Treeview, right click on Analysis 1 > **Loadcase 1**, select the **Rename** option and change the loadcase name to **Panel A Permanent**

- Select the surface representing the top left-hand panel (Panel A).

- Assign the dead loading by dragging and dropping the attribute **Self Weight** from the 🔧 Treeview onto the selection. The loading assignment dialog will be displayed. Select the **Single Loadcase** option, the analysis **Analysis 1** and loadcase **Panel A Permanent**, then click the **OK** button.

✏️ **Note.** The loading assigned to each panel will be visualised for each loadcase as it is assigned to the model

- Drag and drop the dataset **Imposed Load 5kN/m2** from the 🔧 Treeview onto the selection, accepting all defaults, and change the loadcase name to **Panel A Variable** and click the **OK** button.

- Select the top right-hand panel (Panel B).

- Assign the dead loading by dragging and dropping the attribute **Self Weight** from the 🔧 Treeview onto the selection, accepting all defaults, and change the loadcase name to **Panel B Permanent** Leave the load factor as 1 and click the **OK** button.

- Drag and drop the attribute **Imposed Load 5kN/m2** from the 🔧 Treeview onto the selection and change the loadcase name to **Panel B Variable** and click the **OK** button.

- Select the bottom-left panel (Panel C).

- Assign the dead loading by dragging and dropping the attribute **Self Weight** from the 🔧 Treeview onto the selection. Change the loadcase name to **Panel C Permanent** and click the **OK** button.

- Drag and drop the attribute **Imposed Load 5kN/m2** from the 🔧 Treeview onto the selection. Change the loadcase name to **Panel C Variable** and click the **OK** button.

- Select the bottom right-hand panel (Panel D).

- Assign the dead loading by dragging and dropping the attribute **Self Weight** from the 🔵 Treeview onto the selection. Change the loadcase name to **Panel D Permanent** and click the **OK** button.

- Drag and drop the attribute **Imposed Load 5kN/m2** from the 🔵 Treeview onto the selection. Change the loadcase name to **Panel D Variable** and click the **OK** button.

The 🕒 Treeview should now contain eight loadcases consisting of a Permanent and Variable loadcase for each panel.

Saving the model

File
Save

🖫 Save the model file.
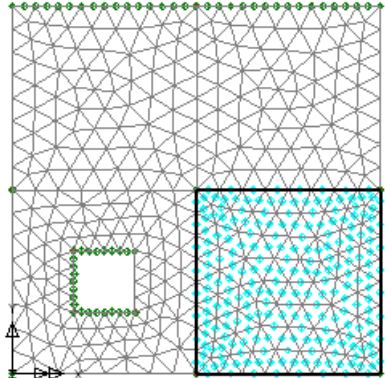
Running the Analysis

With the model loaded:

⬜ Select the **Solve Now** button from the toolbar and click **OK** to run the analysis.

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

If the analysis is successful...

Analysis loadcase results are added to the 🕒 Treeview.

Addition files will be created in the directory where the model file resides, including:

- ❑ **slab_design.out**   this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- ❑ **slab_design.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.

❑ **slab_design_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 **Slab_Design**

File
   Script   >
      Run Script...
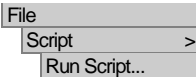
• To recreate the model, select the file **slab_design_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

▤ Rerun the analysis to generate the results.
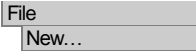
Viewing the Results

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

• Turn off the **Attributes** layer in the 🔲 Treeview.

• In the 🕒 Treeview right-click on results loadcase **Panel A Permanent** and choose **Set active**

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

• Select entity **Force/Moment** and results component **MX** to see the results for Permanent effects in Panel A.
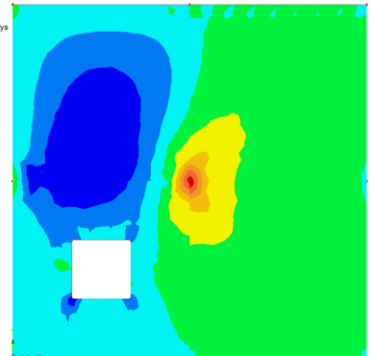
Wood-Armer Results

Slab design will be carried out based on the Wood-Armer moments in the slab. These results are computed from the orientation of the reinforcement, so the reinforcement

capacities/details must be defined and assigned to the model before results can be made available. Two attributes are available from which Wood-Armer moments can be calculated, and these are accessed from the Attributes > Wood Armer and Attributes > Design > RC Slab Design menu items. Since we are going to carry out slab design calculations we will create an RC Slab Design attribute.

Defining RC Slab design attribute

Attributes
　Design　　　　　>
　　RC Slab Design...

- From the **Reinforcement Library** select bar sizes as **Metric**.

- Set all **bar sizes** as **12mm** and leave the **spacing** as **150mm**.

- Set the **top and bottom cover** to be **20mm**.

- Ensure the **angle of bars in y' from bars in x** is set to be **90º**

- Enter the attribute name as **Slab** and click **OK.**
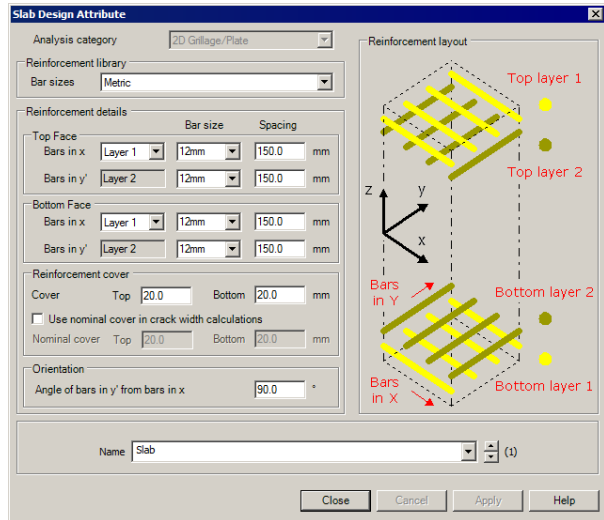
Assigning RC slab design attribute

- Select the whole model and drag and drop the RC Slab attribute **Slab (T[C=20mm, DiaX=12mm, DiaY=12mm], B[C=20mm, DiaX=12mm, DiaY=12mm], Skew=90)** from the 🔵 Treeview onto the selected Surfaces.

📝 **Note.** Assigning the RC Slab attribute adds a new results entity **Wood Armer** to the available results.

- In the Treeview right-click on **Contours** and choose **Properties**

- Set the **Entity** as **Wood Armer** and **Component** as **MX(B)**, click **OK** to close the dialog and plot the contours
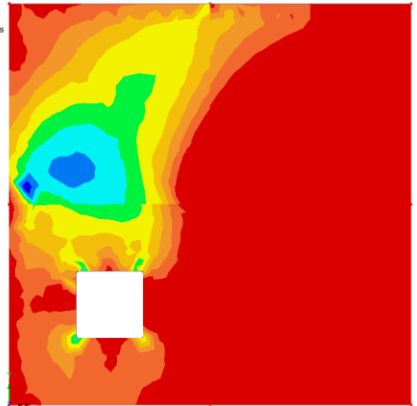
Defining Load Combinations

The loadcases will be combined to provide the most adverse loading effects. This is achieved using the smart load combination facility within LUSAS Modeller.

Ultimate Limit State (ULS) design to EN1990:2002 requires permanent actions to be factored by 1.0 where they are beneficial, and 1.35 where they are adverse. Variable actions are omitted where beneficial, and are factored by 1.5 where adverse. (These factors may be subject to national variation).

ULS design

Analyses
Smart
Combination...

- Include all loadcases in the smart combination. To do this select the first load case in the loadcase selector at the bottom-left of the then hold the **Shift** key down and scroll down the list and select the last loadcase.

- Click the ⏩ button to add these loadcases to the included list.

- In the grid set the beneficial and adverse factors as shown in bold text in the table that follows:

Name	Beneficial Factor	Adverse Factor
Panel A Permanent	1	1.35
Panel A Variable	0	1.5
Panel B Permanent	1	1.35
Panel B Variable	0	1.5
Panel C Permanent	1	1.35
Panel C Variable	0	1.5
Panel D Permanent	1	1.35
Panel D Variable	0	1.5

- Change the smart combination name to **ULS**

- Click the **OK** button to complete the definition of the smart combination.

A maximum and a minimum smart combination entry will be created in the 🕐 Treeview. The maximum combination will produce the most adverse hogging moments whilst the minimum combination will produce the most adverse sagging moments.

Note. When the properties of the max combination are modified the corresponding min combination is updated automatically.

SLS design

For SLS Design, calculation of crack widths will be based on service loads and hence a second combination is required.

Analyses
  Smart
  Combination...

- Include all the results loadcases (but not the two ULS combinations) in the Smart combination. To do this select the first load case in the loadcase selector at the bottom-left of the then hold the **Shift** key down and scroll down the list and select the last loadcase.

- Click the ⟩⟩ button to add these loadcases to the included list.

- Enter the beneficial and adverse load factors in the grid as follows

Name	Beneficial Factor	Adverse Factor
Panel A Permanent	1	1
Panel A Variable	0	1
Panel B Permanent	1	1
Panel B Variable	0	1
Panel C Permanent	1	1
Panel C Variable	0	1
Panel D Permanent	1	1
Panel D Variable	0	1

- Change the smart combination name to **SLS**

- Click the **OK** button to complete the definition of the smart combination

- Select **ULS (Min)** from the 🔍 Treeview with the right-hand mouse button and pick the **Set Active** option.

- In the Set Active dialog ensure **Max/ min primary component with coincident effects** is selected.


```
Combining on: MX
ULS (Min)
Entity: Wood Armer - Thick Plate
Component: MX(B) (Units: N.m/m)
     -74.1181E3
     -65.8828E3
     -57.6474E3
     -49.4121E3
     -41.1767E3
     -32.9414E3
     -24.706E3
     -16.4707E3
     -8.23535E3
Maximum 0.0 at node 16
Minimum -74.1181E3 at node 45
```
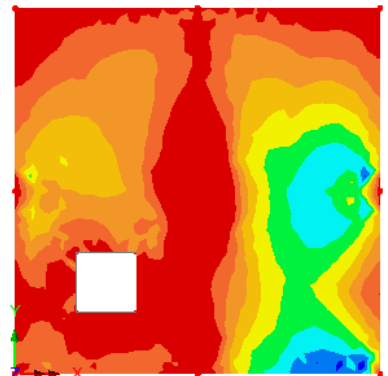
- Select entity **Force/Moment** and results component **MX** to combine, applying the variable factors based on the minimum moments in the X direction and click the **OK** button.

⚠ **Caution.** Whilst not a problem with this example, note that if you use smart combinations with 'derived components' you may not obtain the expected results. The online help provides details of such circumstances which you should fully understand before using this facility. In this worked example, a linear primary component is being used to create the smart combination from which the Wood-Armer values will be

computed. It will be necessary to carry out design checks using primary components in both directions (X and Y) to ensure the most onerous combination is found.

Using the RC Slab Design facility

The RC Slab design facility enables calculation of required steel reinforcement areas for ULS loadcases and calculation of crack widths for SLS loadcases. ULS reinforcement design is based on calculated Wood-Armer moments whereas crack width calculations are based on principal moments. The effective depth is computed from the top and bottom reinforcement bar sizes and covers provided in the RC Slab attribute.

Caution. This example uses an irregular mesh of thick plate elements and for these elements the results are output in global directions that, in this case, match the intended reinforcement directions. In other modelling situations, if the elements in a model are orientated such that their local axes vary from one another, or if an alternative coordinate system is required, the results will need to be transformed to a consistent direction. This can be achieved by either setting a local coordinate set or choosing an appropriate results transformation option on a results layer property dialog prior to opening the RC Slab Designer.

Design
  RC Slab Design...

- Ensure that **United Kingdom** and **BS EN 1992-1-1:2004/NA:2005** are selected from the dropdown list of countries and design codes

- Change the Characteristic yield strength of the reinforcement to **460** and ensure the Characteristic cylinder strength of concrete is **40**

- Click **Next** to proceed to the Crack Width Settings page.

Note. For 2D plate models the design calculations options are set to bending only. For a 3D model comprising shell elements, slab design can be carried out considering in-plane effects as well as bending.

- Ensure the crack width settings are as shown on the adjacent dialog.

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

Two new results entities **Slab Design (ULS)** and **Slab Design (SLS)** will now be available for selection in all results processing facilities.
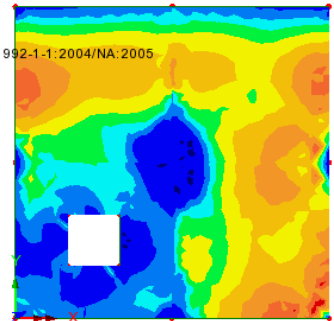
Viewing results for bottom face of slab

With loadcase **ULS (Min)** still set active in the 🔍 Treeview:

- In the 🔲 Treeview double-click on the **Contours** layer and select entity **RC Slab (ULS) – BS EN 1992-1-1:2004/NA2005** and component **UtilMax(B)**.

It can be seen that the utilisation is exceeded by approximately 51%.

Note. UtilMax(B) provides the maximum of two utilisations Util(Mx(B)) and Util(My(B)). The first is based on the Wood Armer component MX(B) and the slab bending capacity in the that direction. The second is based on MY(B) and the corresponding capacity in that direction. Individual utilisations may be viewed with respective components Util(Mx(B)) and Util(My(B)).

Plotting contours of bottom reinforcement area

- In the 🔲 Treeview double-click on the **Contours** layer and change the contour results component to **Asx(B)** to determine the area of steel required in the bottom layer x direction. A steel area of 1135mm²/m is required, which can be accommodated by using 16mm bars at 150mm centres.

Changing bottom reinforcement bar sizes

- In the Treeview double click the RC Slab attribute **Slab** and change the bottom bar diameters used in the bottom face to be **16mm** in both the **x** and **y'** directions. The slab design will be recomputed.

- In the Treeview double-click on the **Contours** layer and select results component **UtilMax(B)**. A maximum utilisation of around 0.92 can be seen, so the bottom reinforcement is now adequate.
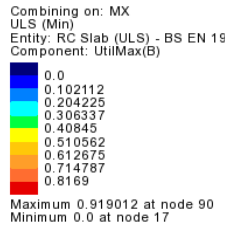
A check is also required that the same face is adequate when the smart combination is optimised for MY:

- Select **ULS (Min)** from the Treeview with the right-hand mouse button and pick the **Set Active** option.

- In the Set Active dialog set the primary component to be **MY** and check that this combination is also satisfied. A maximum utilisation of just under 0.95 is seen.

Viewing results for top face of slab

The viewing results process described previously needs to be repeated for the top face of the slab, using the combination that will generate the maximum effects in the top face:

- Select **ULS (Max)** from the Treeview with the right-hand mouse button and pick the **Set Active** option.

- In the Set Active dialog select **Max / min primary component with coincident effects** entity **Force/Moment** and results component **MX** to combine, applying the variable factors based on the moments in the X direction and click the **OK** button.

- In the Treeview double-click on the **Contours** layer and select entity **RC Slab (ULS) – BS EN 1992-1-1:2004/NA:2005** and component **UtilMax(T)**.

Here the utilisation is exceeded around the central support. To determine how widespread the problem is the contour range will be modified:

- Double-click the contours
  layer in the 🔲 Treeview,
  select the **Contour Range**
  tab, check the **maximum**
  option and enter **1.0**, and
  then click **OK.** The adjacent
  image shows the extent of
  the problem.

Combining on: MX
ULS (Max)
Entity: RC Slab (ULS) - BS EN 1992-1-1 2004/NA 2005
Component: UtilMax(T)

0.0
0.125
0.25
0.375
0.5
0.625
0.75
0.875
1.0

Maximum 4.01757 at node 708
Minimum 0.0 at node 22
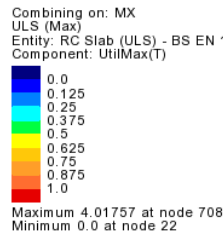
Changing top reinforcement bar sizes

- Repeat the procedure of
  changing the reinforcement
  bar sizes for the top layer
  from 12mm diameter to
 16mm diameter bars. The
  extent of the failure is
  reduced and now much
  more localised around the
  central support.

Combining on: MX
ULS (Max)
Entity: RC Slab (ULS) - BS EN 1992-1-1 2004/NA 2005
Component: UtilMax(T)

0.0
0.27056
0.54112
0.811679
1.08224
1.3528
1.62336
1.89392
2.16448

Maximum 2.43504 at node 708
Minimum 0.0 at node 22

- Revisit the contours layer
  properties and remove the
  maximum value for the contour range. Whilst the utilisation is greater than 1, the
  area of which this value occurs is very localised.
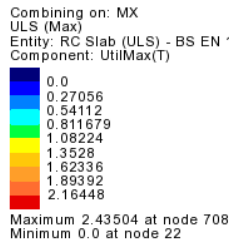
Note. Whilst it is tempting to increase the size of the reinforcement further it is important to understand the limitations of the idealised analysis model. Very high moments are generated at the point support. In reality this support will have some width which will reduce the support moments. It is possible that the model could be refined to account for these support conditions, alternatively the designer should satisfy him/herself, through hand calculation or otherwise, that the forces and moments in the actual configuration (accounting for column sizes and drops etc) can be satisfied.

Caution. The ULS design of reinforcement assumes that the slab is "under-reinforced" and therefore the calculations are only valid when this holds true. The contour plots Util(tcx(T)), Util(tcy(T)), Util(tcx(B)) and Util(tcy(B)) provide utilisations on Tension Control and indicate whether the slab will undergo ductile failure and is hence "under-reinforced." Plots of Tension Control must therefore be checked to ensure a valid design. For more information see the online help.

Concrete crack checking

The calculation of crack widths in the RC Slab Design facility is based on principal moments rather than Wood-Armer moments as used in reinforcement design. Effective steel areas are interpolated based on the orientation of the principal moments with respect to the steel bar directions. Extensive coverage of the crack calculation procedure is provided in the Application Manual (Bridge, Civil & Structural).

Crack width checking (bottom face)

- Select **SLS (Min)** from the Treeview with the right-hand mouse button and pick the **Set Active** option.

- In the Set Active dialog set the primary component to be **MX**

- In the Treeview double-click on the **Contours** layer and select entity **RC Slab (SLS) – BS EN 1992-1-1:2004/NA2005** and component **CWMax(B)**. This displays the maximum crack widths calculated in the bottom face.
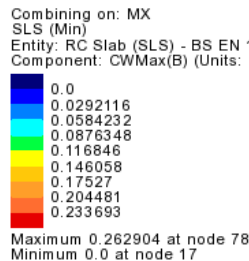
📝 **Note.** A warning message is displayed when the effective steel stress calculated exceeds the nominal yield value. Only the maximum value is reported along with the face at which that occurs but it is possible that the maximum value may be exceeded in both faces. In this example the maximum steel stress reported arises from hogging over the central support.

The crack widths in the bottom face are found to be within acceptable limits. The top face now needs to be checked.

Crack width checking (top face)

- Select **SLS (Max)** from the ⏰ Treeview with the right-hand mouse button and pick the **Set Active** option.

- In the Set Active dialog set the primary component to be **MX**

Combining on: MX
SLS (Max)
Entity: RC Slab (SLS) - BS EN 1992-1-1:2004/NA:2005
Component: CWMax(T) (Units: mm)

0.0
0.0826786
0.165357
0.248036
0.330714
0.413393
0.496072
0.57875
0.661429

Maximum 0.744107 at node 708
Minimum 0.0 at node 22

- In the 🔲 Treeview double-click on the

  Contours layer and select entity **RC Slab (SLS) – BS EN 1992-1-1:2004/NA2005** and component **CWMax(T)**. This displays the maximum crack widths calculated in the bottom face.
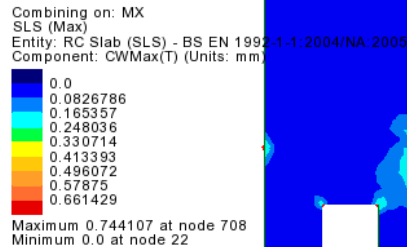
It can be seen that the over-utilised area is concentrated around the idealised support condition and the designer must determine the validity of this assessment.

Note. The crack checking calculations are based on strains calculated from an interpolated effective area of steel. However the actual crack width is calculated taking the bar spacing and bar diameter of the layer orientated closest to the principal moment in question. Contour plots for crack widths "controlled" by each reinforcement layer are provided to identify exactly which bars are controlling the cracks. By default the maximum crack width is displayed.

Note. Slab design results will be available as a results entity until they are turned off by selecting "None" on the slab design dialog.

Save the model

File
 Save

💾 Save the model file.

Note. All the Slab Designer inputs are saved to the model and will be saved when the model is saved. All load combinations, envelopes, and graph datasets, if defined when processing results, 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.

Discussion

## Element types and orientation of results

The orientation of the slab reinforcement is aligned with the results axes. In this example thick plate elements are used whose results axes are always transformed to the global directions.

If shell elements are used (when appropriate) the results must be transformed into a consistent direction coinciding with the intended direction of the reinforcement. By default shell results are orientated along the local element axes.

The following result contours shows the indicative difference between results from global (left) and local (right) axes for a ULS(Max) MX(B) plot.
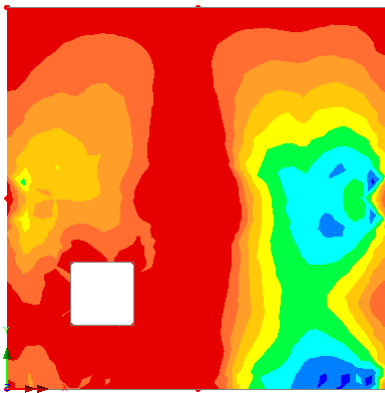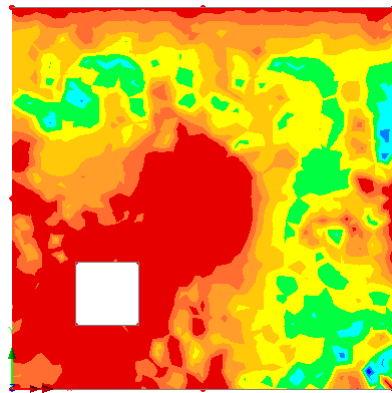

Global Axes   **Local Axes**

- The mesh orientation can be viewed from the Treeview by selecting **Mesh** > **Properties** and selecting the **Show Element Axis** option. Press **OK** to apply changes.

- Results transformations can be applied using the Transform **Set...** button on a results layer's Properties dialog. Selecting **Global axes** will transform local element results into the global axes system. Other options are available..

- Global results are indicated by the uppercase X, Y and Z in the component names, whereas lowercase x, y and z (when seen) are used to denote local results or transformed results.

Element density and results obtained

If the Evaluation Version of LUSAS was used to carry out this example with a reduced default mesh density of 4 divisions per line a slightly reduced accuracy of results will have been obtained.

The following graph for ULS(Min) shows how Bending Moment (MX) along a horizontal 2D slice section through the central three columns typically varies with the number of line mesh divisions used.
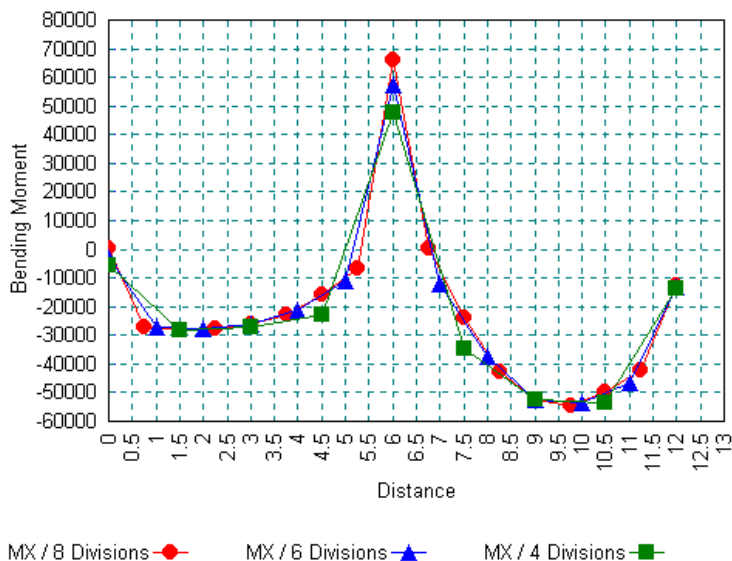
In general, more accurate results are obtained when using more line mesh divisions and hence more elements when modelling slabs of this type. Care should always be taken to use an appropriate number of elements together with a possible refinement of the mesh in areas of interest in order to obtain the best results.