Simple Building Slab Design

For LUSAS version:	20.0
For software product(s):	LUSAS *Civil & Structural* or LUSAS *Bridge.*
With product option(s):	RC Slab Design
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 utilisation plots under ultimate loads and check the locations of cracking under service loads

Keywords

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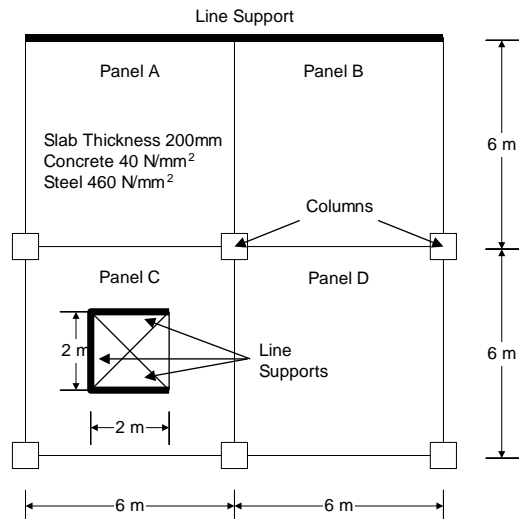
Associated Files

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

❑ **slab_design_modelling.lvb** 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.

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.
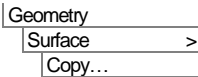
Feature Geometry
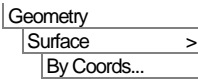
De-select the 3 Columns grid style check box and enter coordinates of (**0, 0**), to define the lower left-hand area of slab, and using the Tab key to move to the next entry field enter coordinates of (**6, 0**), (**6, 6**) and (**0, 6**).

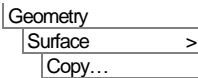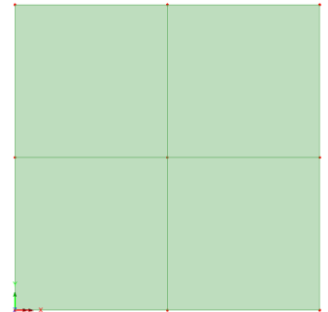- With all the coordinates entered click the **OK** button to create a surface.
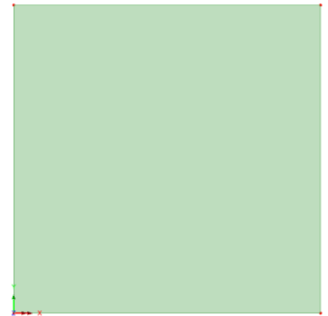
- Select the newly created Surface.

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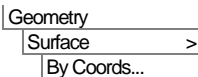
Copy the selected surfaces with a Y translation of **6**.

- 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.
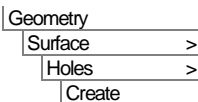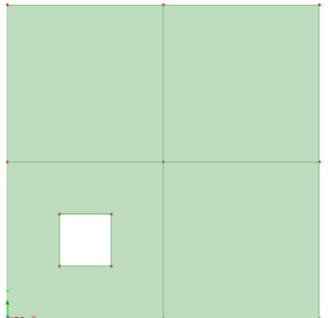
This new surface containing the hole can be seen /

checked by clicking in a blank part of the View 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. For this example, and to give greater accuracy, 8 divisions per line will be used.

Note. 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

File
  Model Properties…

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

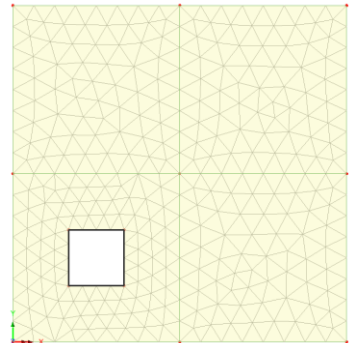Attributes
  Mesh   >
    Surface...

- 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.

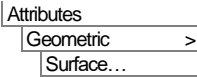Attributes
  Mesh   >
    Line...

- 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

<table>
<tr><td>Attributes</td><td></td></tr>
<tr><td>Geometric</td><td>></td></tr>
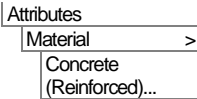<tr><td>Surface…</td><td></td></tr>
</table>

- Enter a thickness of **0.25**. Enter the attribute name as **Thickness 250mm** and click the **OK** button.

- With the whole model selected drag and drop the geometric attribute **Thickness 250mm** from the ⚫ Treeview onto the selected Surfaces. Click **OK** to assign to the selected features for Analysis 1.

Geometric properties are visualised by default.

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

Material Properties

<table>
<tr><td>Attributes</td><td></td></tr>
<tr><td>Material</td><td>></td></tr>
<tr><td>Concrete</td><td></td></tr>
<tr><td>(Reinforced)...</td><td></td></tr>
</table>

- On the RC Material dialog ensure that Country is set to **Europe** and Design code is set to **EN1992**.

- On the drop list for Concrete section select **New**.

- On the dialog that appears select grade **C40/50** and click **OK**.

- On the RC Material dialog for the drop list for Reinforcement select **New**.

- On the dialog that appears select grade **500B** and click **OK**. Click **OK** on the RC Material dialog.

- Select the whole model and drag and drop the material attribute **Mat1 (EN1992)** from the ⚫ Treeview onto the selected Surfaces.

Supports

<table>
<tr><td>Attributes</td><td></td></tr>
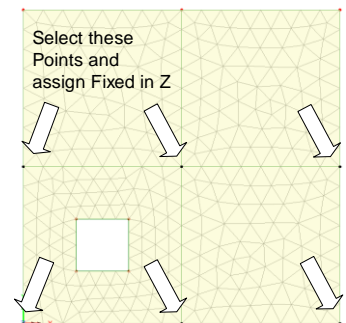<tr><td>Support…</td><td></td></tr>
</table>
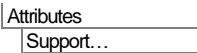
- Select **Fixed** for **Translation in Z** and enter the attribute name **Fixed in Z** and click **OK**.

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 **Points** for **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 **Lines** for **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 **Lines** for **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
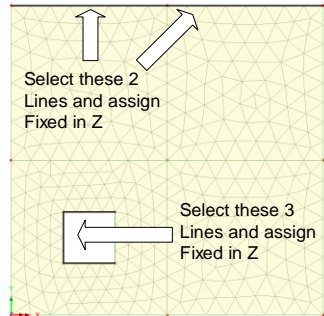
⊕ Select the isometric view button to check that all supports have been assigned correctly.

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

Loading
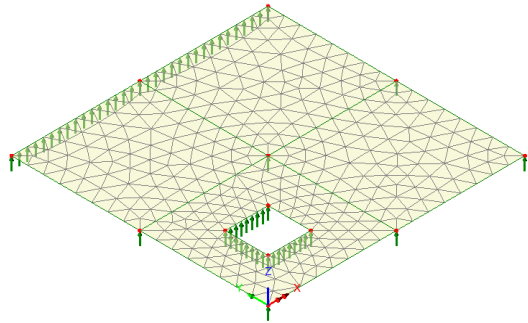
Attributes
Loading…

- 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 **-3000** in the **Z** Direction.

- Enter the attribute name as **Imposed Load 3kN/m2** and click the **Finish** button.

Now the dead and live loading needs to be assigned.

- In the Analyses 🕐 Treeview, right click on **Loadcase 1**, select the **Rename** option and change the loadcase name to **Permanent**

- With the whole model selected, 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 **Permanent**, then click the **OK** button.

- With the whole model still selected, drag and drop the dataset **Imposed Load 3kN/m2** from the Treeview onto the selection, accepting all defaults, and change the loadcase name to **Variable** and click the **OK** button.
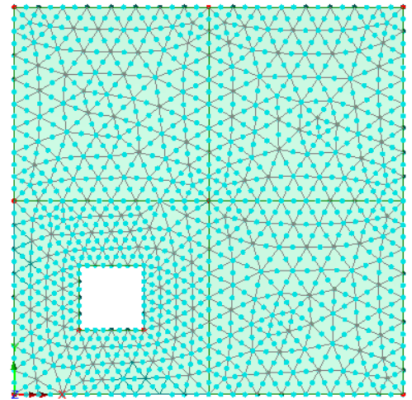
Saving the model

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

7

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.

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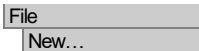
File
New…

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 **Slab Design**

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

- Ensure an Analysis type 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 **slab_design_modelling.lvb** that was downloaded and placed in a folder of your choosing.
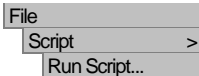
Solve the model to generate the results.

Viewing the Results

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

- In the ⬜ Treeview, turn off the display of the **Attributes** layer.

- In the ⏰ Treeview right-click on results loadcase **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** and click **OK** to see the results for Permanent effects in the slab.

Analysis: Analysis 1
Loadcase: 1:Permanent (G | Permanent Actions)
Results file: Slab_Design_Check~Analysis 1.mys
Entity: Force/Moment - Thick Plate
Component: MX (Units: N.m/m)

-11.5393E3
0.0
11.5393E3
23.0786E3
34.6179E3
46.1573E3
57.6966E3
69.2359E3
80.7752E3

Maximum 80.8931E3 at node 708 of element 332
Minimum -22.9607E3 at node 27 of element 15
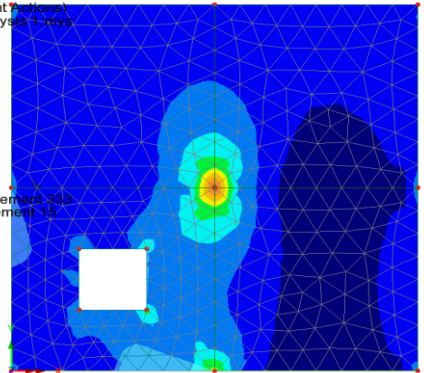
Wood-Armer Results

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 need to create and assign to the model an RC Slab Design attribute.

Defining a Results Transformation attribute

A results transformation attribute must be defined and assigned to all surfaces that are to be designed in order to transform the results according to reinforcement directions.

Attributes
Results
Transformation...

- Choose the **Global axes** option.

- Enter the attribute name as **Global Transformation** and click **OK**.

- Click **OK** to the message that is displayed.

Assigning Results Transformation attribute

- Select the whole model and drag and drop the Results Transformation attribute **Global Transformation** from the 🔵 Treeview onto the selected Surfaces.

- Click **OK** to assign the attribute to surfaces.

Defining the RC Slab/Wall Reinforcement attribute

Attributes
RC Slab/Wall
Reinforcement…

- From the **Reinforcement Library** select bar sizes to be **Metric**.

- For the **Top Face** set the **bar sizes** as **12mm** and set the **spacing** as **100mm** in both x and y'.

- For the **Bottom Face** set the **bar sizes** as **12mm** and set the **spacing** as **200mm** in both x and y'.

- Set the **Top** and **Bottom** cover values to be **20**.

- Ensure the **Angle of bars in y' from bars in x** is set to be **90** degrees.
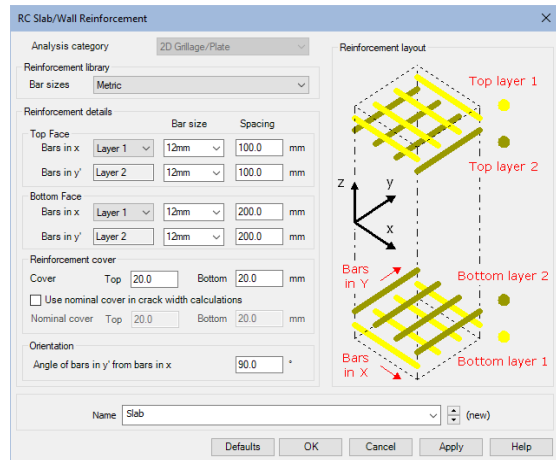
- Enter the attribute name to be **Slab** and click **OK.**

Assigning the 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
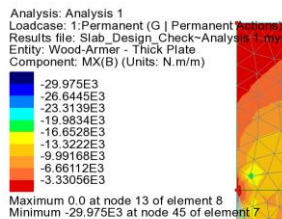
Defining Load Combinations

Loadcases will be combined to provide the most adverse loading effects. Different combination types are required in preparation for the RC Slab Design. This is achieved using the Design Combination facility within LUSAS Modeller.

Combinations will be provided to EN1990:2002 for ULS Fundamental combination (using equation 6.10), SLS Characteristic combination and SLS Quasi-permanent combination.

Design Combinations

- From the county list select **Europe**.

- From the Design code list select **EN 1990 – 2002 Buildings Recommended Values**.

- For the load Permanent set the type as **G | Permanent Actions**

- For the load Variable set the type as **Qi B | Imposed: Category B – office areas**

- Click **Next**.

On the combinations options page:

- Ensure that for ULS the option for **Fundamental combination Eq 6.10** is selected.

- Ensure that for SLS the options for **Characteristic combination** and **Quasi-permanent combination** are selected.

- Ensure that **Create an envelope for each combination type** is selected.

- For the Add / Replace / Delete Combinations option, ensure **Add** is enabled.

- For the Combination approach, ensure **Basic Combination** is selected.

- Click **Finish** to generate the design load combinations.

- From the 🔍 Treeview, select **ULS_F_B (Min)** 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.

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

- Select the 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.

Enveloping on: MX
ULS_F_B (Min)
Entity: Force/Moment - Thick Plate
Component: MX (Units: N.m/m)

-42.8073E3
-28.5382E3
-14.2691E3
0.0
14.2691E3
28.5382E3
42.8073E3
57.0764E3
71.3455E3

Maximum 80.8931E3 at node 708 of element 333 (3:ULS_F_B 1 (1.0*G))
Minimum -47.5287E3 at node 27 of element 15 (6:ULS_F_B 4 (1.35*G+1.5*Qi B))
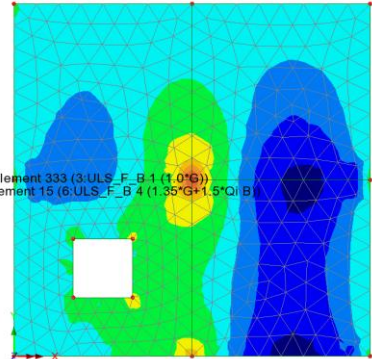
- Click **OK** to accept the warning that appears.

⚠️ **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.

- Ensure that **Europe (Eurocode recommended values)** and **EN 1992-1-1:2004** are selected from the dropdown list of countries and design codes

- Accept all other settings and click **OK**.

RC Slab/Wall Design Attribute

- On the **General** tab select the design calculation method as **Bending and in-plane (Sandwich model / Principal stresses)** and click **OK** to add the design attribute to the ⬡ Treeview.

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

- Select the whole model and drag and drop the design attribute **Des1** from the ⬡ Treeview onto the selected Surfaces.

Note. The Permanent loadcase was set active because a design attribute must be added to a loadcase and not a combination.

Note. Whilst only the General tab was visited for this example, it is possible to modify Shear, Min/Max reinforcement, SLS & Creep properties on the other tabs of this dialog.

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.

Adding RC Slab/Wall Design Results

On this dialog, selections can be made to specify which parts of a model should be checked for which load groups. From the resulting entry in the 🕒 Treeview the design check results can be viewed, and tables of results can be obtained.

- In the Extent panel, select **All members**.

- In the Design check locations panel ensure that **Averaged nodal** is selected.

- In the Load Groups panel press the **Select** button for **ULS**.

- On the Select Loadcases dialog presented, select **ULS_F_B (Max)** and **ULS_F_B (Min)** and add to the Included list. Click **OK**.

- In the Load Groups panel press the **Select** button for **SLS – characteristic**.

- On the Select Loadcases dialog presented select **SLS_C_B (Max)** and **SLS_C_B (Min)** and add to the Included list. Click **OK**.

- In the Load Groups panel press the **Select** button for **SLS – quasi-permanent**.

- On the Select Loadcases dialog presented select add **SLS_QP_B (Max)** and **SLS_QP_B (Min)** and add to the Included list. Click **OK**.

- On the RC Slab/Wall Design Results dialog accept the default name of **RC Slab/Wall Design 1** and click **OK** to finish.
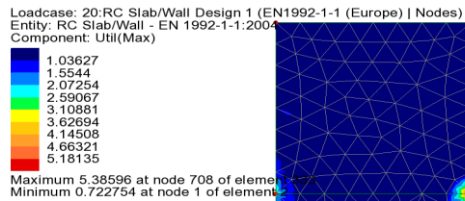
An RC Slab/Wall Design loadcase entry 🛠 named RC Slab/Wall Design 1 will be added inside a Post processing folder in the 🕐 Treeview,

Viewing the Results

- In the 🕐 treeview right-click on the loadcase **RC Slab/Wall Design 1** and select **Set Active**.

- In the 🔲 Treeview double-click on the **Contours** entry to edit the properties.

- Select **OK** on the warning dialog that appears.

- Ensure that Entity is set to **RC Slab/Wall – EN 1992-1-1:2004** and component set to **Util(Max)** and click **OK**.

Loadcase: 20:RC Slab/Wall Design 1 (EN1992-1-1 (Europe) | Nodes)
Entity: RC Slab/Wall - EN 1992-1-1:2004
Component: Util(Max)

1.03627
1.5544
2.07254
2.59067
3.10881
3.62694
4.14508
4.66321
5.18135

Maximum 5.38596 at node 708 of eleme[...]
Minimum 0.722754 at node 1 of element[...]

It can be seen that the maximum utilisation is 5.386. This will be investigated further.

- In the ⏱ treeview right-click on the loadcase **RC Slab/Wall Design 1** and select **Show Results**

- On the **Summary** tab click the **Util(Max)** label at the top of the last column twice to sort by utilisation value in descending value.

RC Slab/Wall Design Results (EN 1992-1-1:2004)

SectionID: All   SurfaceID: All   Loadcase: n/a   Face/Direction: n/a   Help

Detailed results for selected row...   Display only results for selected surfaces   Select surfaces in model   Select elements in model

Summary | Enveloped results | Section definitions | ULS (6.1 & 6.2) | Min-max reinforcement | SLS (crack control) check (7.3.4) | SLS stress limits (7.2) | Min reinforcement (7.3.2)

Section	Surface	Element	Node	Util(ULS)	Util(Asmin,Asmax)	Util(CC)	Util(SLS,S)	Util(AsCC)	Util(Max)
1	3	333	708	5.386	0.723	1.952	2.570	0.306	5.386
1	3	351	708	5.386	0.723	1.952	2.570	0.306	5.386
1	2	304	708	5.377	0.723	2.056	2.651	0.306	5.377
1	2	323	708	5.377	0.723	2.056	2.651	0.306	5.377
1	4	334	708	5.345	0.723	1.945	2.588	0.306	5.345
1	4	354	708	5.345	0.723	1.945	2.588	0.306	5.345
1	1	306	708	4.912	0.723	1.932	2.465	0.306	4.912
1	1	328	708	4.912	0.723	1.932	2.465	0.306	4.912
1	4	352	724	4.277	0.723	1.019	1.223	0.303	4.277

From the summary table it can be seen that:

- The maximum utilisations are due to the Util(ULS) check.

- The Util(CC) and Util(SLS,S) checks also have utilisations greater than 1.0.

These results are to be investigated further by selecting the appropriate tabs.

Viewing detailed results for a slab location

Util(ULS) Check results

- On the main RC Slab/Wall Design Results summary table select the **ULS (6.1 & 6.2)** tab.

- Expand the width of the table and sort the **Util(Max)** column in descending order.

It can be seen that the peak values are due to the shear check Util(VRdc).

Util(CC) check results

- On the main RC Slab/Wall Design Results summary table select the **SLS (crack control) check (7.3.4)** tab.

- Sort the **Util(CW)** column in descending order.

Investigating Util(CC) results further

- Select the first row in the SLS crack control check table and then click the **Detailed results for selected row** button at the top of the dialog. Double clicking the row will also give the detailed results.

- In the Design Calculation Viewer select the **SLS (crack control) check (7.3.4)** and scroll through the results to view data relating to the Utilisation being exceeded. This will be highlighted by red text – there are two instances of this in this check.

From the calculations it can be seen that the utilisation exceeds 1.0 in both the x and y directions.

- Close the Design Calculation Viewer.

Util(SLS,S) check results

- On the main RC Slab/Wall Design Results summary table select the **SLS stress limits (7.2)** tab.

- Sort the **Util(Max)** column in descending order.

The maximum utilisation is due to the check on Util(sigmaS).

- Close the RC Slab/Wall Design Results.

Viewing the contour plots

The contour plots for the results of interest will now be viewed. This is best done using the Design contours option that shows values of utilisation from 0.7 to 1.0 and above.
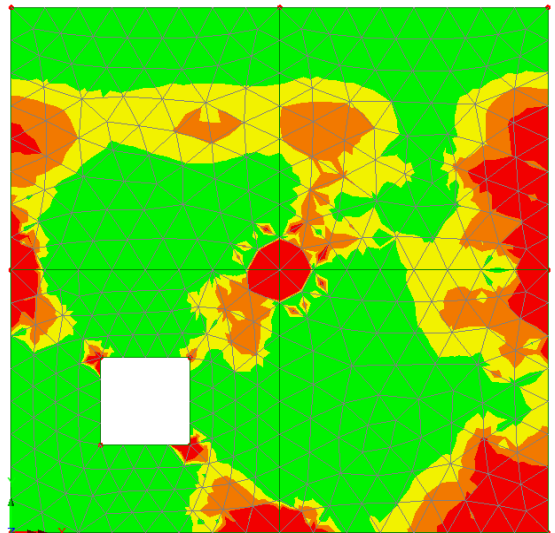
- In the Treeview double-click on the contours layer.

- Ensure the Entity is set **to RC Slab/Wall – EN 1992-1-1:2004** and the component is set to **Util(Max).**

- Select the **Appearance** tab and select the **Design** option. Press **OK**.

The contour plot is as shown below.

Similarly the plots of interest for **Util(ULS), Util(CC)** and **Util(SLS,S)** follow:

Simple Building Slab Design

Util(ULS)

Loadcase: 20:RC Slab/Wall Design 1 (EN1992-1-1 (Europe) | Nodes)
Entity: RC Slab/Wall - EN 1992-1-1:2004
Component: Util(ULS)

0.7
0.8
0.9
1.0

Maximum 5.38596 at node 708 of element 333
Minimum 0.171288 at node 6 of element 1

Util(CC)

Loadcase: 20:RC Slab/Wall Design 1 (EN1992-1-1 (Europe) | Nodes)
Entity: RC Slab/Wall - EN 1992-1-1:2004
Component: Util(CC)

0.7
0.8
0.9
1.0

Maximum 2.05619 at node 708 of element 304
Minimum 7.00426E-3 at node 1260 of element 586
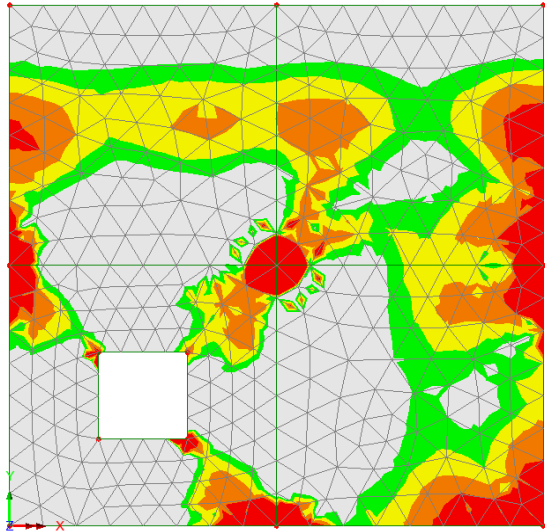
Util(SLS,S)

Loadcase: 20:RC Slab/Wall Design 1 (EN1992-1-1 (Europe) | Nodes)
Entity: RC Slab/Wall - EN 1992-1-1:2004
Component: Util(SLS.S)

0.7
0.8
0.9
1.0

Maximum 2.65072 at node 708 of element 304
Minimum 0.0122267 at node 1260 of element 586

Further contour plots are available which can provide additional information. For example the plot of Util(CC) can be separated into top and bottom checks, **Util(CC(T)** and **Util(CC(B))** receptively as shown below.
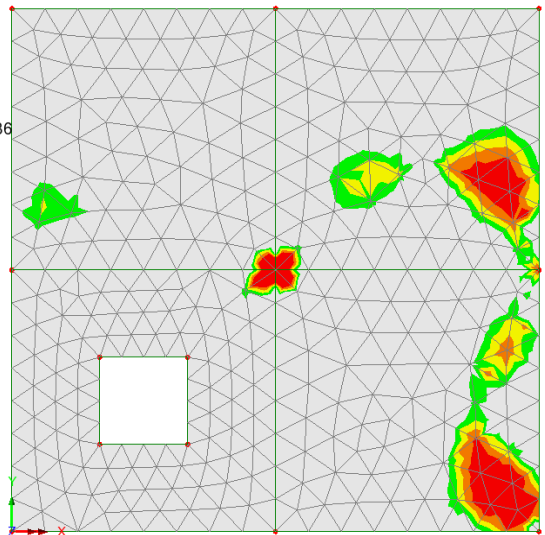
Util(CC(T))

Loadcase: 20:RC Slab/Wall Design 1 (EN1992-1-1 (Europe) | Nodes)
Entity: RC Slab/Wall - EN 1992-1-1:2004
Component: Util(CC(T))

0.7
0.8
0.9
1.0

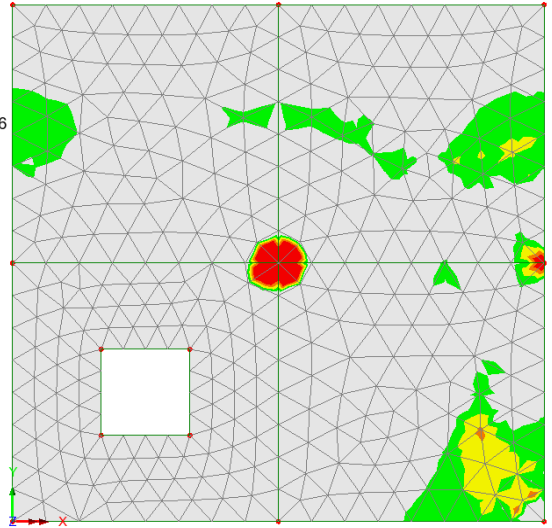Maximum 2.05619 at node 708 of element 304
Minimum 0.0 at node 5 of element 1
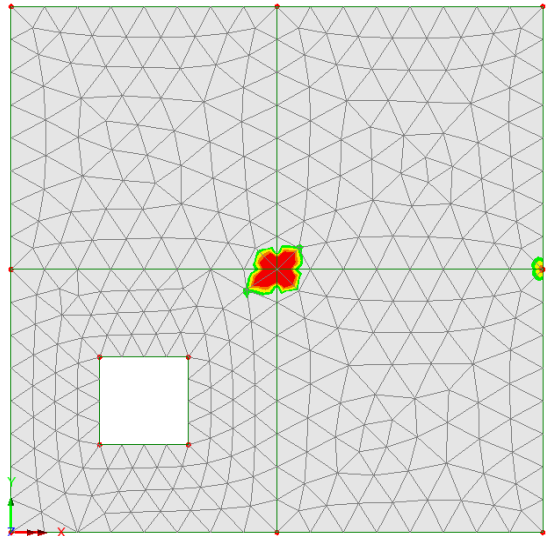
Util(CC(B))

Loadcase: 20:RC Slab/Wall Design 1 (EN1992-1-1 (Europe) | Nodes)
Entity: RC Slab/Wall - EN 1992-1-1:2004
Component: Util(CC(B))

0.7
0.8
0.9
1.0

Maximum 1.63847 at node 78 of element 7
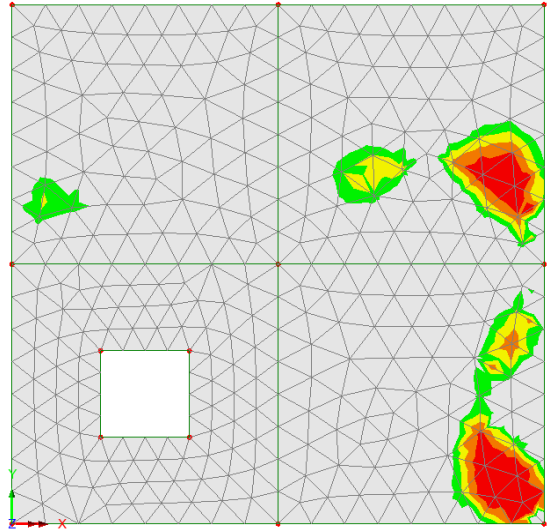Minimum 0.0 at node 17 of element 6

From this it can be seen that the Util(CC(T)) is critical over the central column support while Util(CC(B)) is critical in the span near the right hand edge of the slab.

Notes regarding utilisations obtained

The objective of this example is to demonstrate the use of the RC slab/wall design facility. To this end the modelling of the structure has been somewhat simplified.

A more accurate model could be considered by the inclusion of details such as column heads or drop panels as necessary. Also, the modelling approach could be developed to include the columns themselves or to provide constraint equations to reflect behaviour more accurately at the head of the column.

In addition, the plate/shell elements in LUSAS are high performance. Therefore, the effects of stress concentration may be considerable, as is the case in this example. As with any finite element software, engineering judgement should be applied when analysing the results.

As a result of this there are very high utilisations at and around the support locations. Whilst some increase to the area of reinforcement is required to particular regions the previous comments should be noted when considering the reinforcement requirements near the supports.

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.

Caution. If shell elements are used (when they may be appropriate) the results must be transformed into a consistent direction coinciding with the intended direction of the reinforcement. Note that by default shell results are orientated along the local element 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 Teaching and Training version of LUSAS was used to carry out this example with a reduced default mesh density of 4 divisions per line a reduced accuracy of results will be obtained.

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.