Simple Building Slab Design

For LUSAS version:	23.0
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

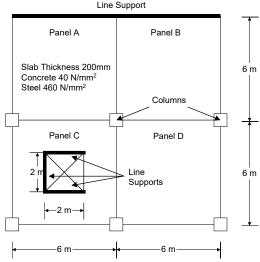
Four panels of a concrete slab supported by a wall, columns and a lift shaft are to be analysed, with reinforcement areas computed for bending moment only. Shear and displacement checks need to be carried out separately. The geometry of the slab is as shown.

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.

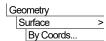


Note. The Analysis category determines which elements will be available for modelling. Here we are only interested in the out-of-plane bending effects in a flat slab and therefore we can approximate this behaviour with 2D plate elements.



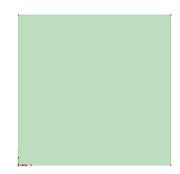
Note. Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.

Feature Geometry

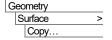


De-select the 3 Columns grid style check box and enter coordinates of (0, 0), to define the lower left-hand area of slab, and using the Tab key to move to the next entry field enter coordinates of (6, 0), (6, 6) and (0, 6).

 With all the coordinates entered click the OK button to create a surface.

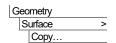


• Select the newly created Surface.



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.

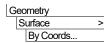


Copy the selected surfaces with a Y translation of **6**.

Click the OK button to define two new Surfaces.

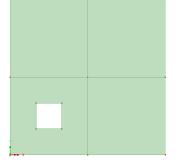
Defining a Hole in a Slab

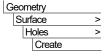
Next, a hole representing the lift shaft needs to be defined. This is done by first creating a surface representing the extent of the lift shaft and then selecting both the surrounding and inner surface to define the hole.



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 slightly less accurate results.

| File | Model Properties...

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

 • Select the element type as **Thick Plate**, the element shape as **Triangle** and the interpolation order as **Quadratic**. Select a **Quadratic 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.

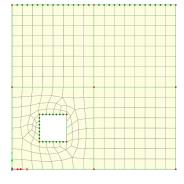
 With the whole model selected drag and drop the mesh attribute Thick Plate from the Treeview onto the selected Surfaces

In the vicinity of the lift core less elements are required.

Attributes

| Mesh > |

- 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 **0.25**. 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

Attributes

Material

Concrete (RC frame / slab design)...

- 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.

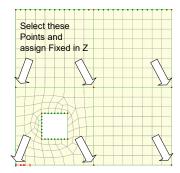
Supports

Attributes
Support.

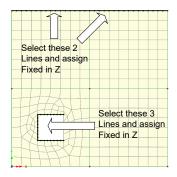
 Select Fixed for Translation in Z and enter the attribute name Fixed in Z and click OK.

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.

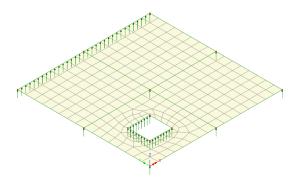


- Select the 2 Lines representing the line support and drag and drop the support attribute Fixed in Z from the Treeview and assign to Lines for All analysis loadcases by clicking the OK button.
- Select only the top, bottom and left-hand lines defining the lift shaft and drag and drop the support attribute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



Select the isometric view button to check that all supports have been assigned correctly.

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



Loading

- Select Body force, viscous support loading then 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.
- Select Distributed loads, then Global Distributed 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**

Attributes Loading...

Attributes Loading...

- 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.

		8.0	E3 ₁ N	m2				€.0E	3'N	m2			8.0	E3 N.	/m2
8.	E3 N	/m2						-8.0E	3 N	/m2			6.0	E3/N	m2
6.	3E3 N	/m2					48.0	E3 N/r	nž					6.0	E3 N/
8.	3E3 N	/m2						43.0E	3 N	/m2				6.0	E3 N/
	18.0	E3 N	/m2					43.0E	3 N	/m2				6.0	E3 N.
		-6.	DE3 N	l/m2	۰			43.0E	3 N	/m2			18.1	0E3 N	/m2
8.	9E3 N	/m2			٠			48 OF	EB N	/m2			18.	0E3 N	/m2
	۰	+8.0)E3 N	/m2				0	8.0	E3 N.	/m2			8.0	DE3 N
		-3.0	E3 N	/m2			8.	0E3 N/I	m2					6 .0	E3 N
8.0	DE3 N	-	1	Z	1.			E3 N/r	m2				6.0	E3 N	(m2
	•			m2 m2		N/mi		0E3 N/I	m2				18.1	0E3 N	/m2
	1	3 OB 3			1	+	-8.	0E3 N/	mi2				6.1	0E3 N	/m2
	X	e de		n2			-6.	0E3 N/	m2				18.	0E3 N	/m2
	N		3 N/		1		8.	0E3 N/	m2			٠	18.	0E3 N	/m2
		-	3 N/	1	X	1.	8.	0E3 N/	m2				18.	0E3 N	/m2
		-	E3 N	-			6.1	3E3 N/r	m2				8	0E3 N	/m2

Saving the model



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.

In addition, these files will be created in the LUSASFiles\<model_name> folder:



- □ **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. You may download this file from the user area of the LUSAS website.



□ slab_design_modelling.lvb carries out the modelling of the example.

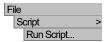


Start a new model file. If an existing model is open Modeller will prompt for unsaved data to be saved before opening the new file.

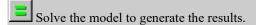
- Enter the file name as **Slab Design**
- Use the default **User-defined** working folder.
- Ensure an Analysis type of **2D Inplane** is set.
- Click the **OK** button.



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.



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



Viewing the Results

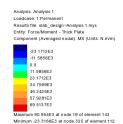
Analysis loadcase results are present in the Treeview and loadcase results for the last solved loadcase are set to be active by default.

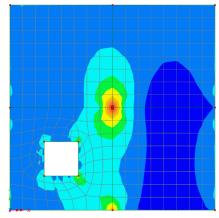
• In the Treeview, turn off the display of the **Attributes** layer.

The Geometry layer is to be retained to allow assignment of reinforcement layout to surfaces in the model.

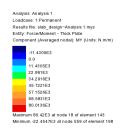
- In the Treeview right-click on results loadcase Permanent and choose Set active
- With no features selected, click the right-hand mouse button in a blank part of the
 active window and select the Contours option to add the contours layer to the
 Treeview.

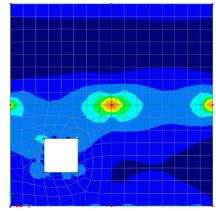
• Select entity
Force/Moment –
Thick Plate and
results component
MX and click OK
to see the results
for Permanent
effects in the slab
in the global X
direction.





Select entity
 Force/Moment –
 Thick Plate and results component
 MY and click OK to see the results for Permanent effects in the slab in the global Y direction.





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 assigned to the slab, 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 the RC Slab/Wall Reinforcement attribute

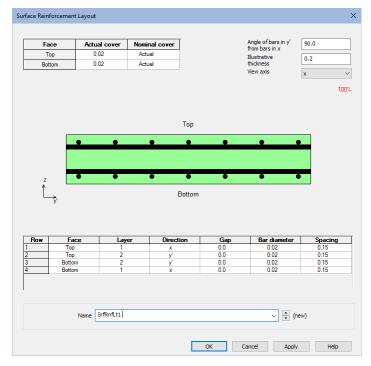
| Attributes | Design | Surface | reinforcement...

 On the Surface reinforcement dialog choose New... from within the cell shared by the 'Stretch to fit' row and column.



In its simplest form a single set of top and bottom slab reinforcement can be defined. This can then be assigned to selected surfaces in the model, where it will stretch to fit the extents of those surfaces that represent the concrete slab.

- On the Surface Reinforcement layout dialog, ensure the top and bottom actual cover values are set to be **0.02**
- Ensure the Angle of bars in y' from bars in x is set to be 90 degrees.
- Enter an Illustrative thickness of **0.2**
- Define the bar diameter for the top and bottom reinforcement layers to be **0.02** and ensure the spacing of bars in the top and bottom layers is **0.15**



- Click **OK** to return to the previous dialog. Then click **OK** on that dialog to complete the definition.
- Accept the default surface reinforcement layout name of SrfRnfLt1 and click OK.

Assigning the RC slab design attribute

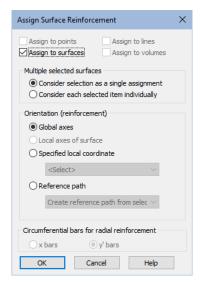
- Select the whole model and drag and drop the Surface reinforcement attribute Slab reinft from the Treeview onto the selected Surfaces.
- On the resulting dialog accept the default settings and click **OK**.



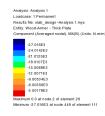
Note. Considering the surfaces selected as a single assignment assigns the reinforcement arrangement across the extent of all surfaces selected. Considering each selected surface individually inserts the defined reinforcement arrangement within the extent of each surface.

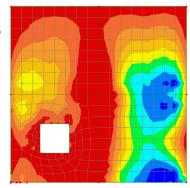


Note. Assigning the Surface reinforcement attribute adds a new results entity **Wood Armer** to the available results.



- In the Treeview rightclick on Contours and choose Properties
- Set the Entity as Wood-Armer – Thick Plate and Component as MX(B), click OK to close the dialog and plot the contours





• Click in an empty region of the view window to deselect the previous selection

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.

Defining a design combination

Design Design

Combinations...

- From the country 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.

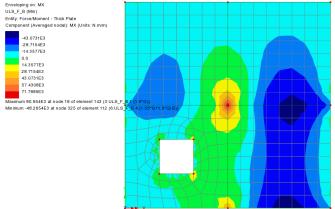
On the combinations options page:

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

- Ensure that for SLS the options for Characteristic combination and Quasipermanent 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.

Plotting results for a design combination

- From the Treeview, select ULS_F_B (Min) with the right-hand mouse button and pick the Set Active option.
- Select the entity Force/Moment -Thick Plate and results component MX combine, to applying the variable factors based on the minimum moments in the X direction.



Ensure Change existing drawing layers to above component is selected 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 the covers provided in the RC Slab attribute.



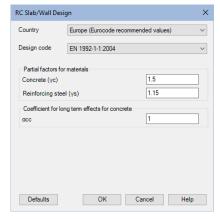
Caution. This example uses an irregular mesh of thick plate elements and for these elements the results are output in global directions that, in this case, match the intended reinforcement directions. In other modelling situations, if the elements in a model are orientated such that their local axes 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/Wall

Design...

- 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** (see note below) 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 is 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 inplane effects as well as bending.

Adding RC Slab/Wall Design Results

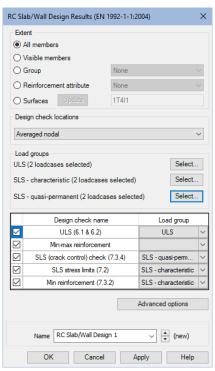
Design RC Slab/Wall Design

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 – quasipermanent.
- 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,



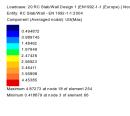
Viewing the Results

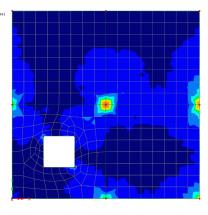
- In the treeview right-click on the loadcase RC Slab/Wall Design 1 and select Set Active.
- In the Treeview double-click on the Contours entry to edit the properties.
- If needed, select **OK** on the warning dialog that appears.
- Ensure that Entity is set to RC Slab/Wall EN 1992-1-1:2004 and component set to Util(Max)

and click **OK**.

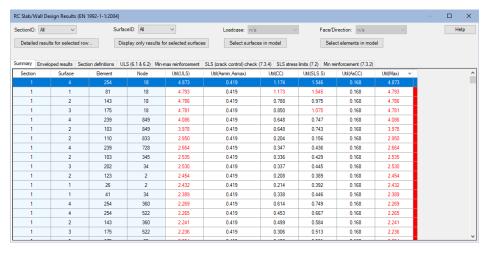
It can be seen that the maximum utilisation of 4.82 is above the single point support representing the central column. This

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.



From the summary table it can be seen that:

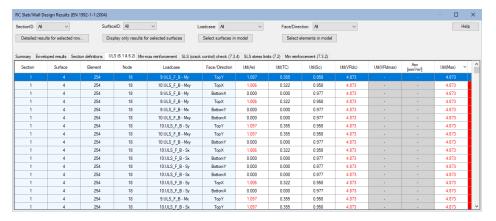
- The maximum utilisations are due to the Util(ULS) check.
- The Util(CC) and Util(SLS,S) checks also have utilisations greater than 1.0.

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

Viewing detailed results for a slab location Util(ULS) Check results

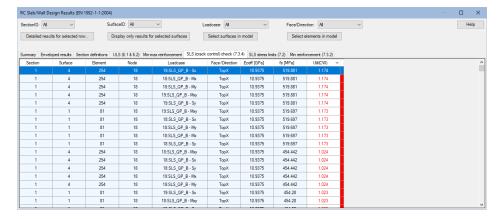
- On the main RC Slab/Wall Design Results summary table select the ULS (6.1 & 6.2) tab.
- Expand the width of the table and sort the Util(Max) column in descending order.

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



Util(CC) check results

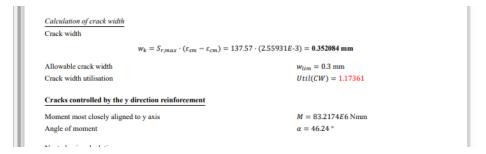
- On the main RC Slab/Wall Design Results summary table select the SLS (crack control) check (7.3.4) tab.
- Sort the **Util(CW)** column in descending order.



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 bar spacings and utilisation. Failures are highlighted in red text.

From the calculations it can be seen that for crack width calculations for the top face of the slab, the reinforcement spacing limited has been satisfied in both the x and y directions but the crack width utilisation exceeds 1.0 in the x-direction.

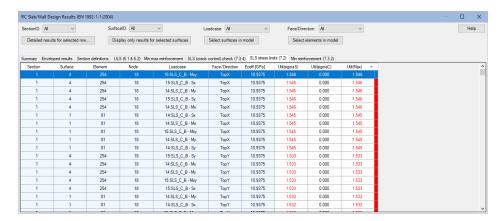


Close the Design Calculation Viewer.

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).



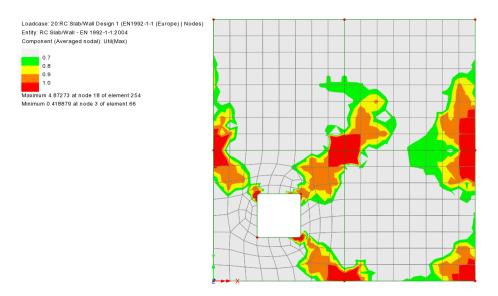
• Close the RC Slab/Wall Design Results.

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.

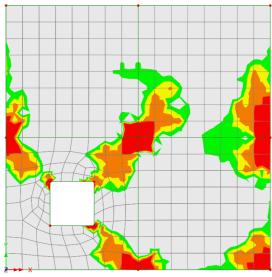
- In the Treeview double-click on the contours layer.
- Ensure the Entity is set to RC Slab/Wall EN 1992-1-1:2004 and the component is set to Util(Max).
- Select the Appearance tab and select the Design option. Press OK.

The contour plot is as shown below.



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





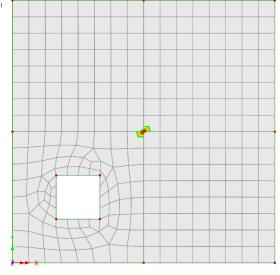
Util(CC)

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

0.7
0.8

0.8 0.9 1.0

Maximum 1.17361 at node 18 of element 254 Minimum 1.00452E-3 at node 1425 of element 231

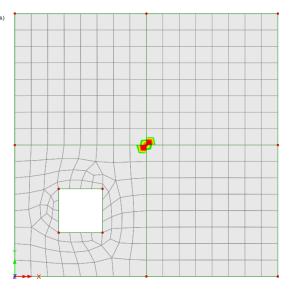


Util(SLS,S)

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

0.7 0.8 0.9 1.0

Maximum 1.5457 at node 18 of element 254 Minimum 0.167534 at node 1 of element 66



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 (Averaged nodal): Util(CC(T))

0.7 0.8 0.9 1.0

Maximum 1.17361 at node 18 of element 254 Minimum 0.0 at node 59 of element 27

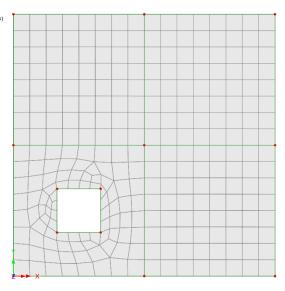
Util(CC(B))

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

0.7

0.8 0.9 1.0

Maximum 0.309451 at node 395 of element 109 Minimum 0.0 at node 18 of element 81



From this it can be seen that the Util(CC(T)) is critical over the central column support while the utilisation of Util(CC(B)) is less than 1.

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





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.

Simple Building Slab Design	