SCADA Pro™ 18
Structural Analysis & Design

Example 5
Flat slabs

ACE HELLAS
INTEGRATED SOLUTIONS

CONTENTS

I. FLAT SLABS - EXAMPLE 3

1. Simulation Process 4

2. Loads 14

3. Analysis 15

4. Results 16

5. Members Design 17

5.1 Parameters 18

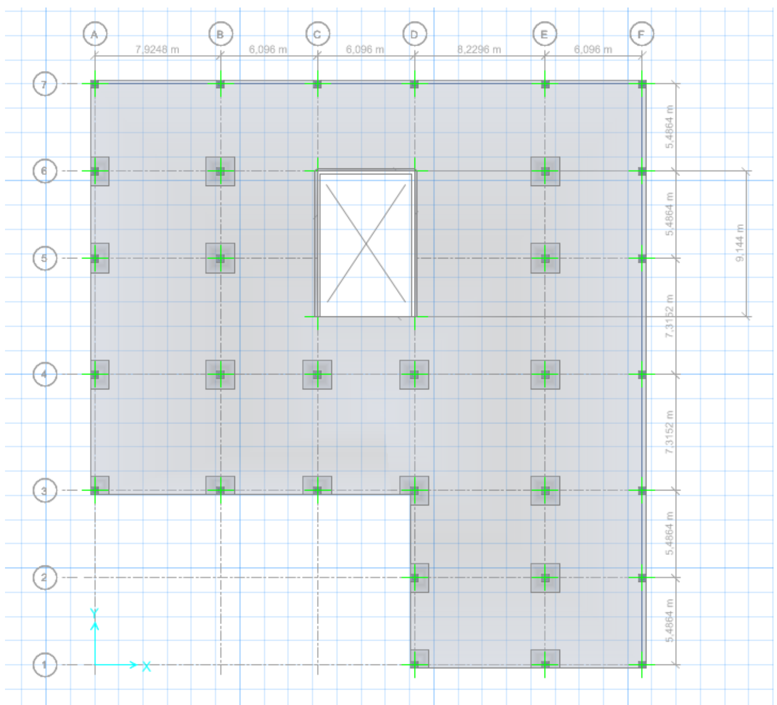
5.3 Display X, Z 20

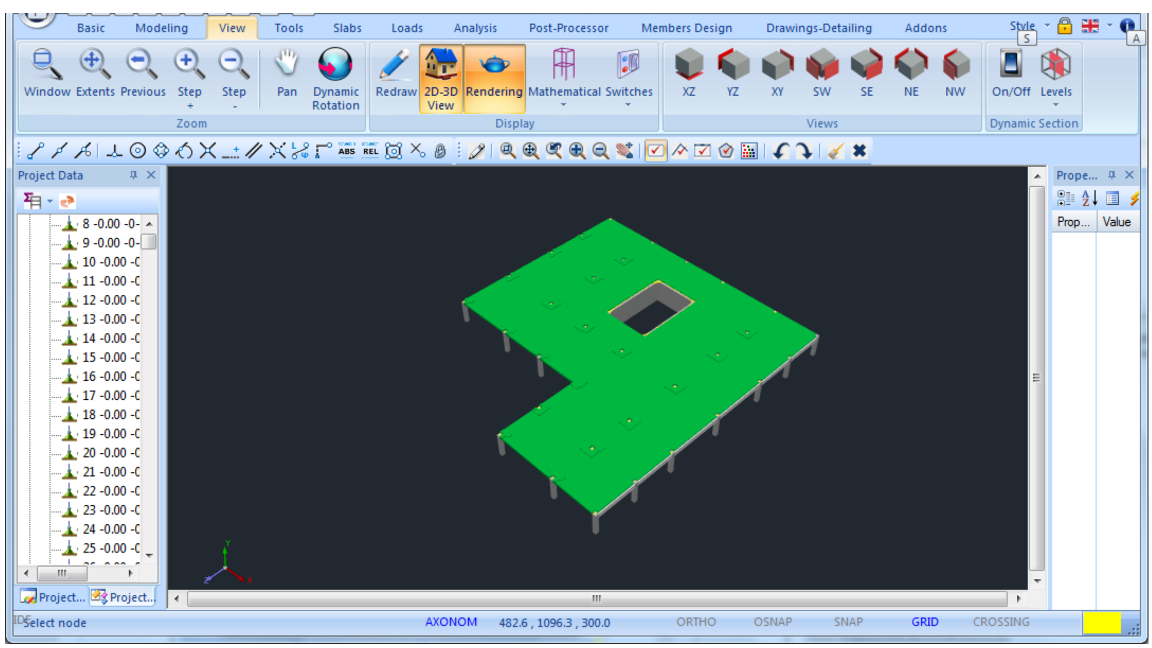
5.4 Diagrams X, Z 22

5.5 Results 23

I. FLAT SLABS - EXAMPLE

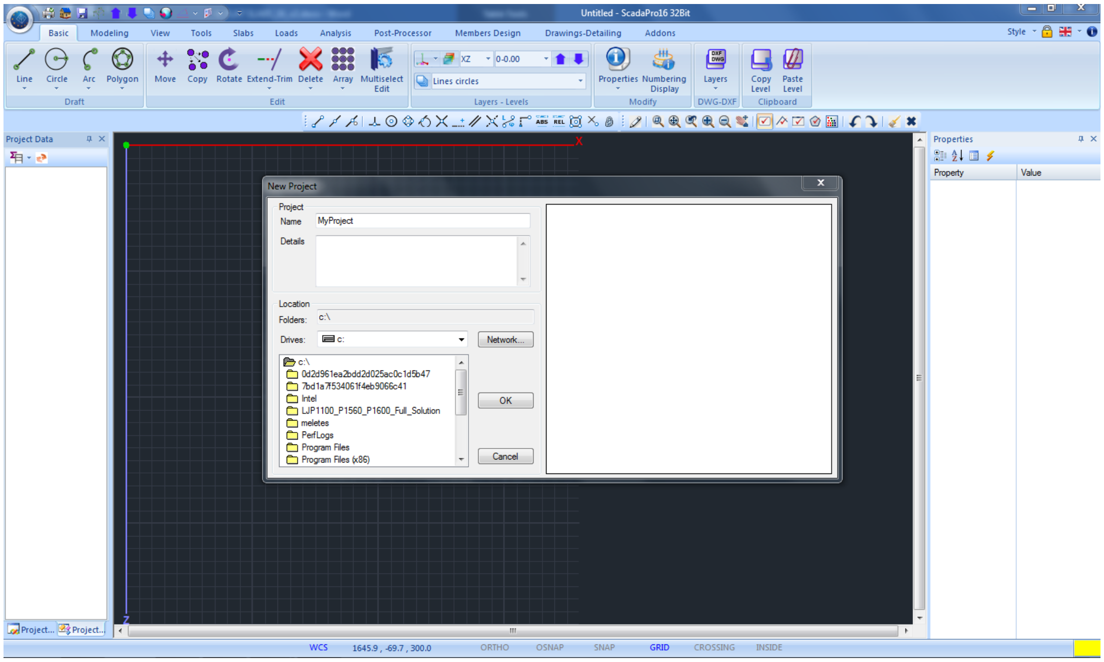
In the following example, the structure below is analyzed. It is the same structure analyzed in the CSI Safe 2014 tutorial. Useful conclusions may be drawn by the comparison of the results from the two software.



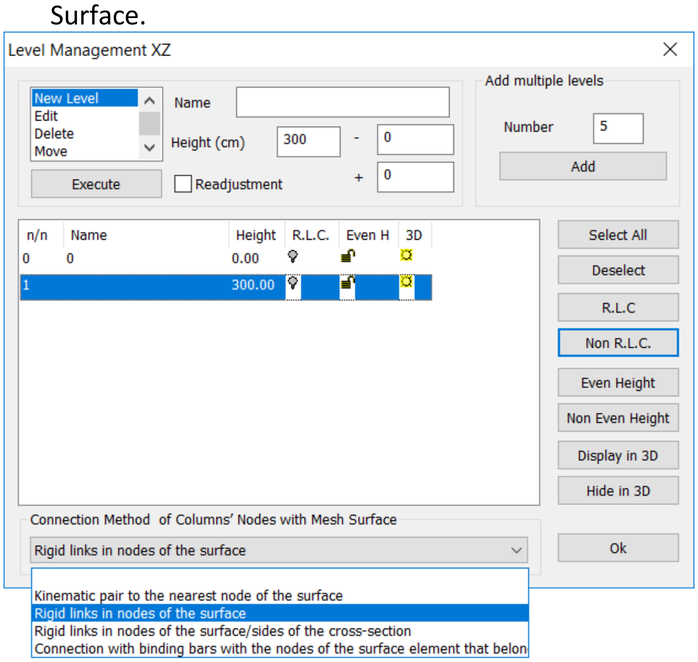


1. Simulation Process

1. The process begins by creating a new project and name the file.

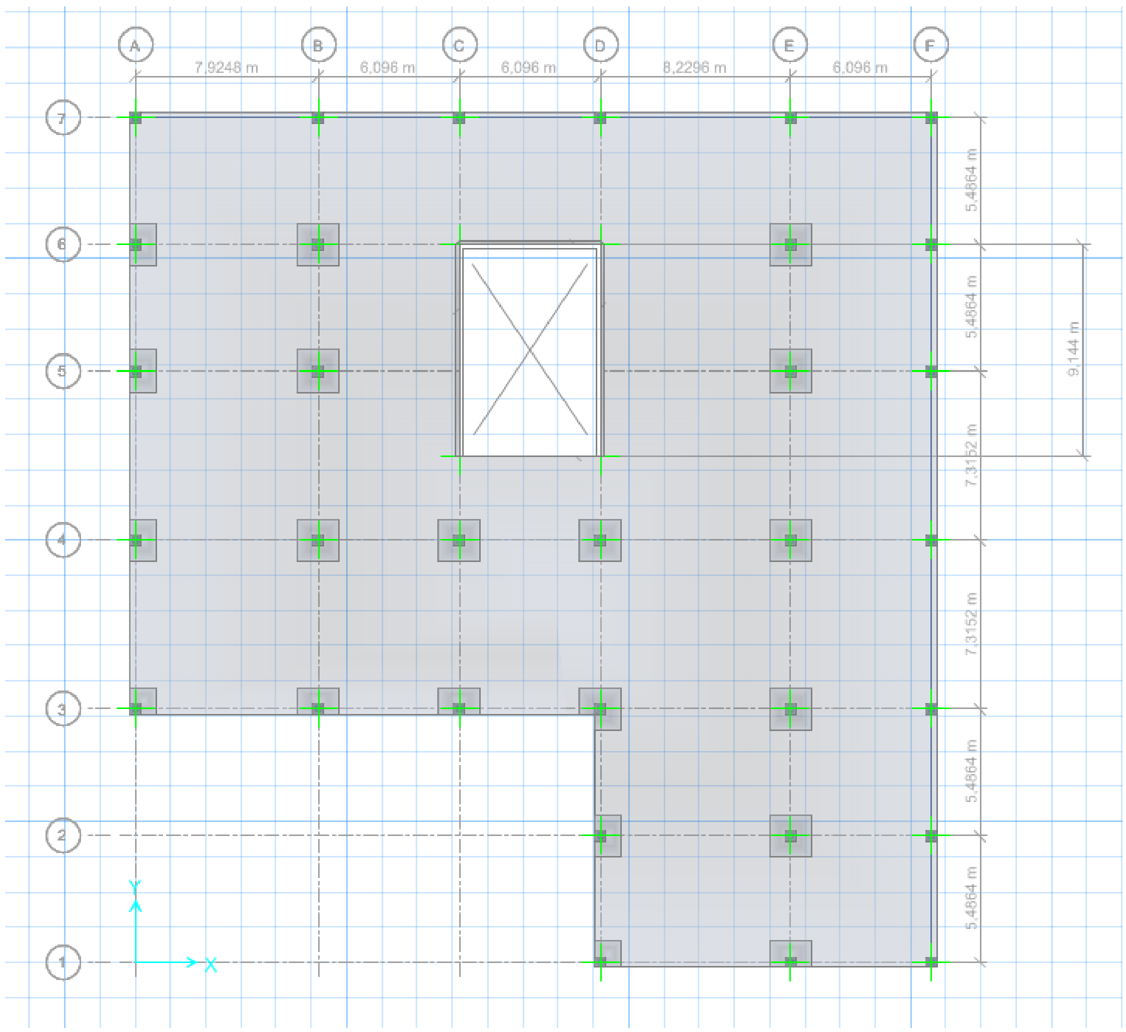


2. Then define the levels. At the level that will define the flat slab, turn off the Rigid Link Constrain function and choose how the Connection Method of Columns nodes with Mesh Surface.



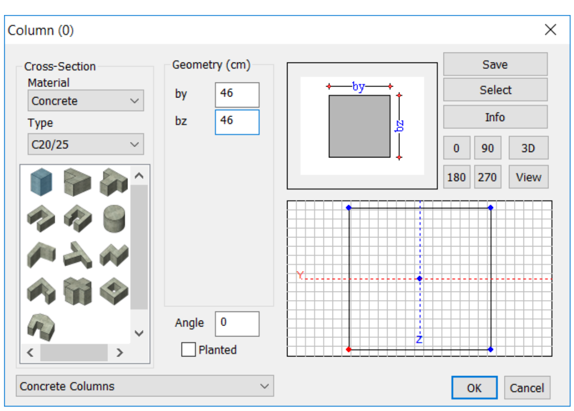
At the bottom of the window there is the option for choosing the Connection Method of the Columns nodes with the Mesh Surface, for the selected level. Select “Rigid Links in nodes of the surface” and press Update.

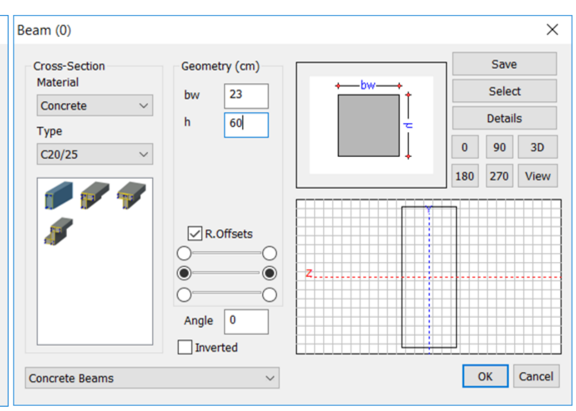
3. The top view of the model that will simulate described in the picture below.



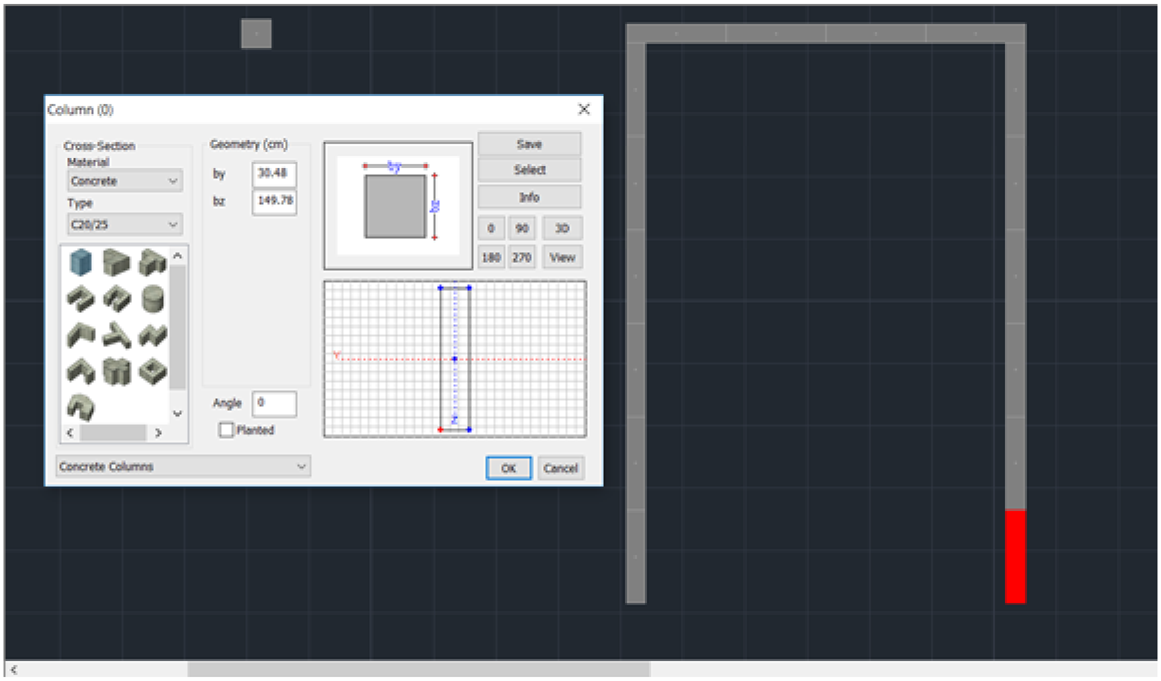
Level 1

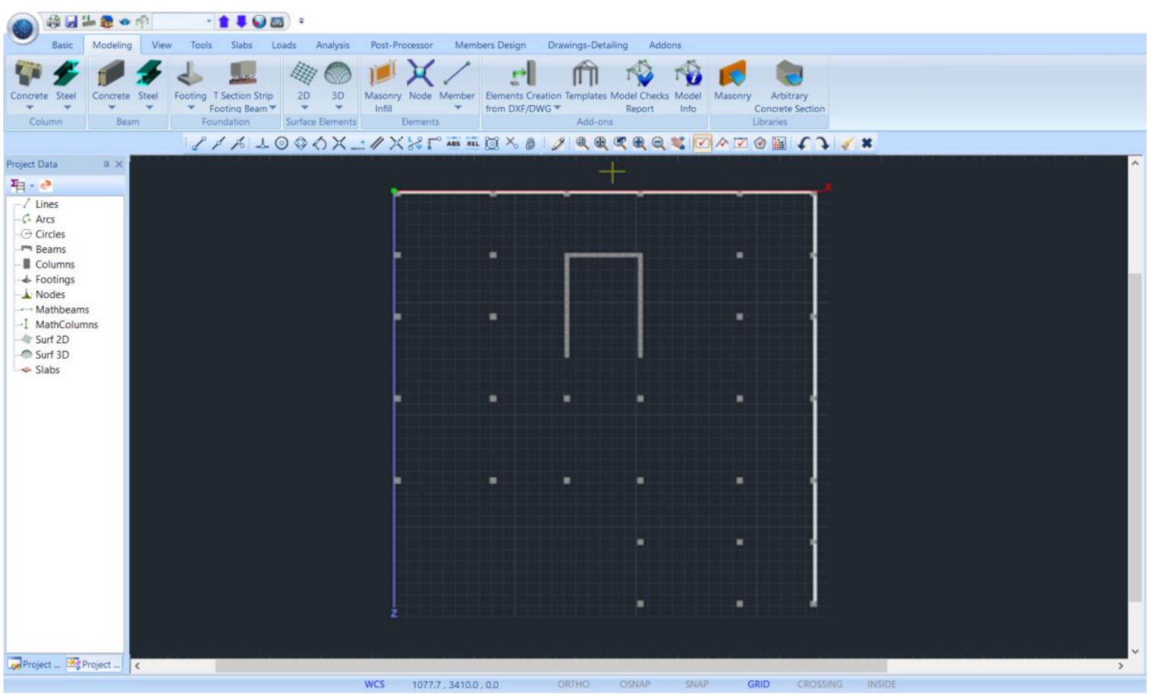
4. Activate level 1 and start inserting the columns and the beams using modeling commands according the image's dimensions:



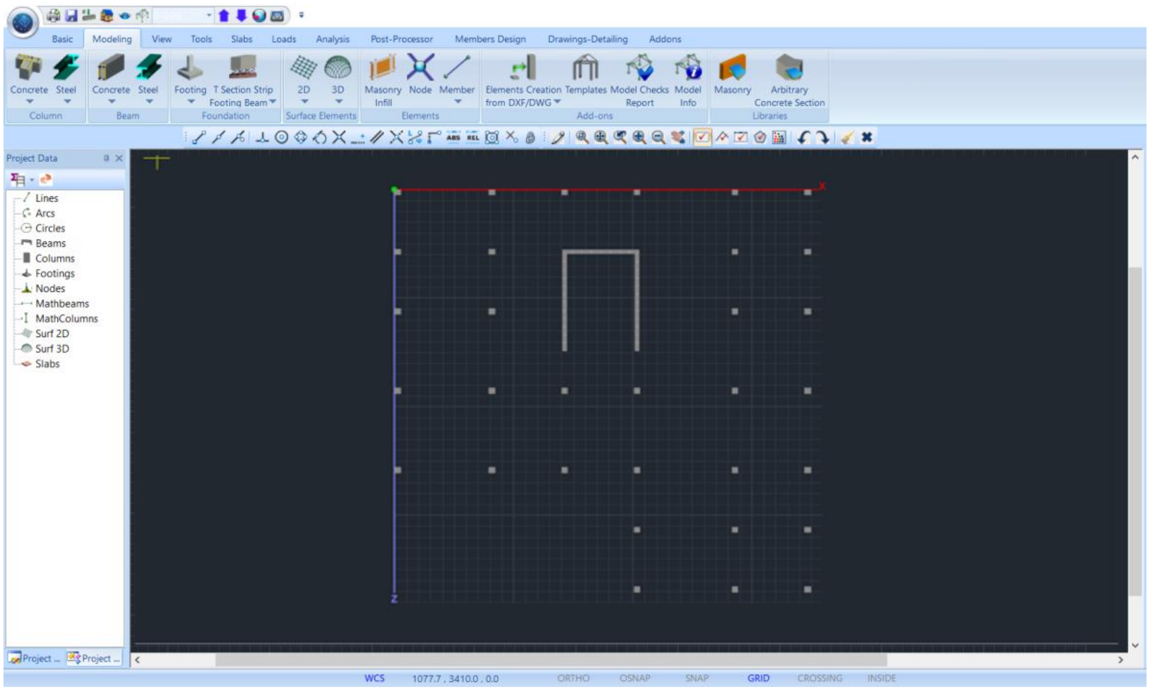


About the core, choose simulation with complex cross section, due to the large dimensions. Use rectangular columns profiles, and then join them by using height rigidity beam members to yield the stiffness of the component.

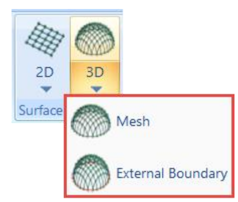


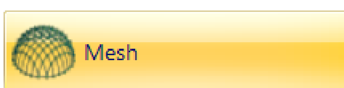


Level 0 5. Using Copy-Paste Level copy all in level 0 and delete the beams.

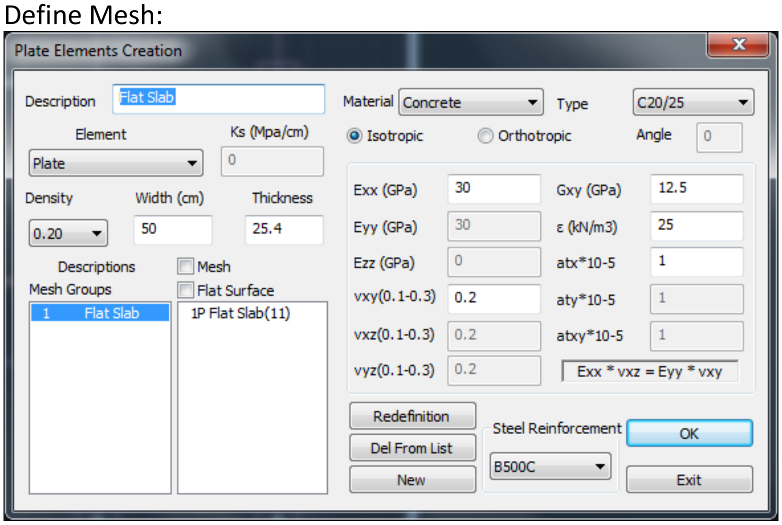


6. The next step is the definition and creation of the mesh surface which defines the flat slab.





Define Mesh:



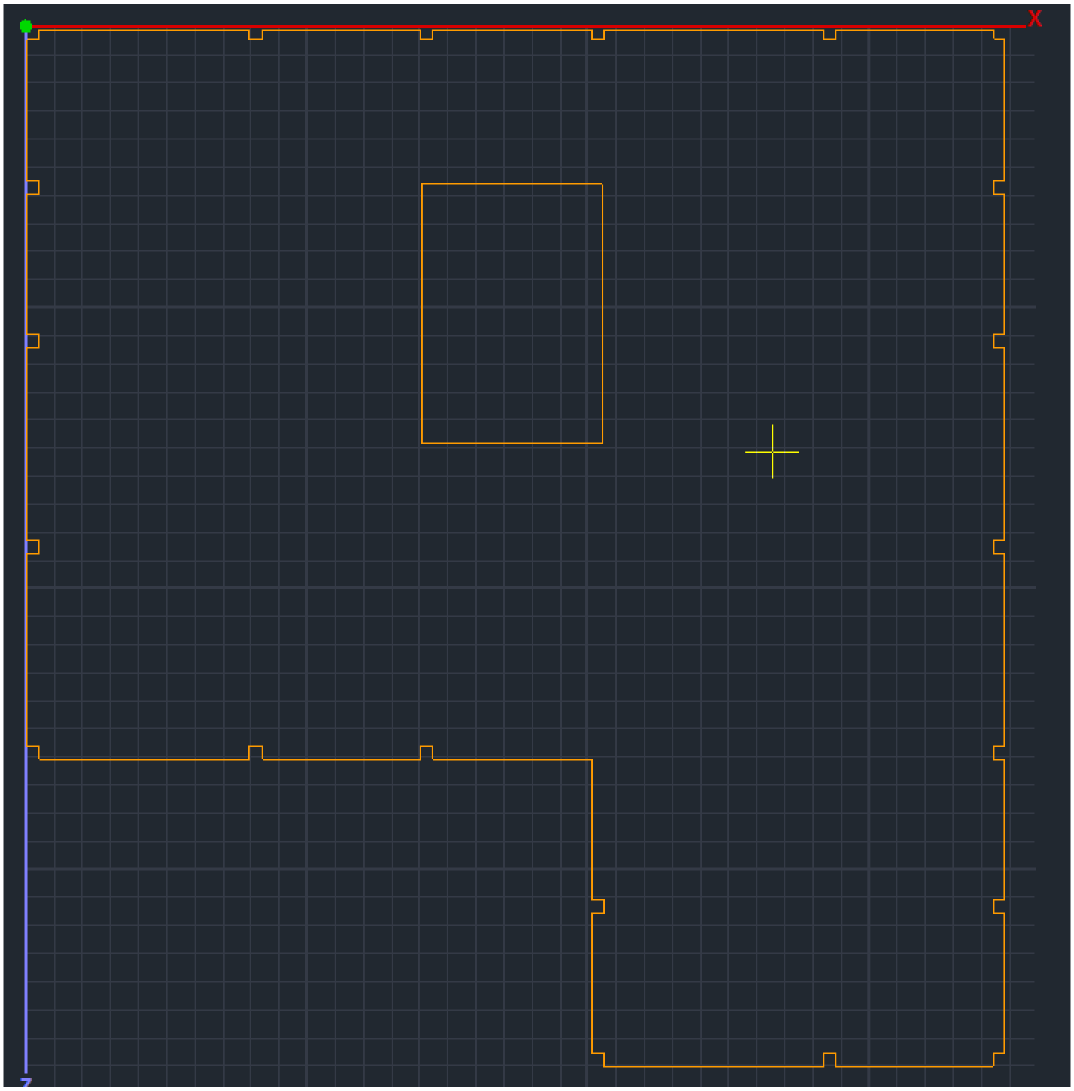
And using External Boundary set the slab outline. The boundary can be defined automatically, simply by selecting one of its lines and then right-click.

Preconditions: The boundary:

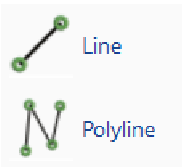
- should be closed and without ramifications and
- the external columns should be excluded from this.

So we must define a border like the orange solid line shown in Figure.

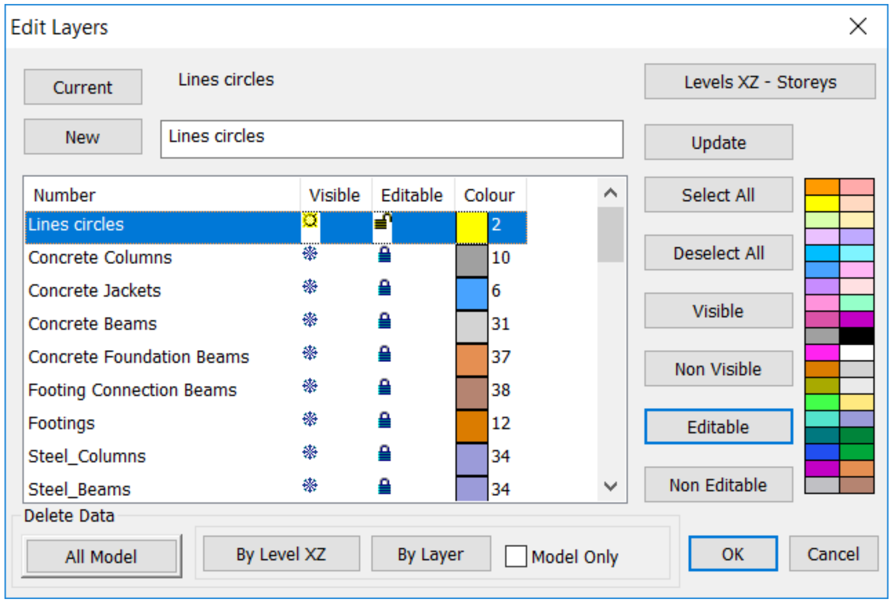
⚠ The outer limit of the hole in the center will be set later as a hole.

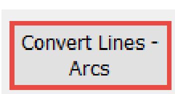


NOTES!!!

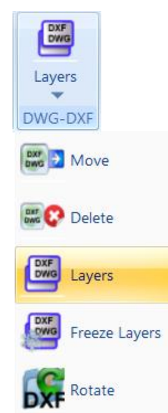


❖ To draw the External Boundary you can use the design commands and osnaps. Edit Layers command needs to make Layers Non Visible and Non Editable for easier design.





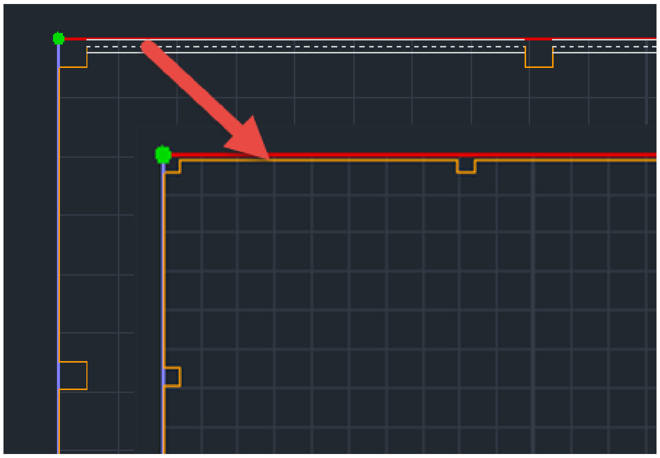
❖ Alternatively, insert an auxiliary design file, containing the outer limit and by using the Layers command, convert the design lines in SCADA lines.



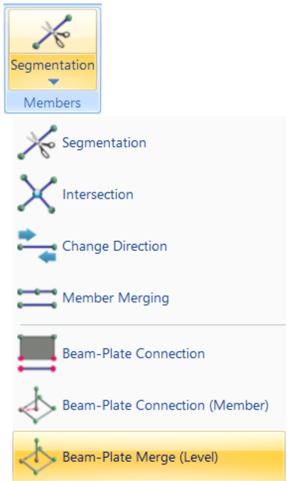
ATTENTION!!!

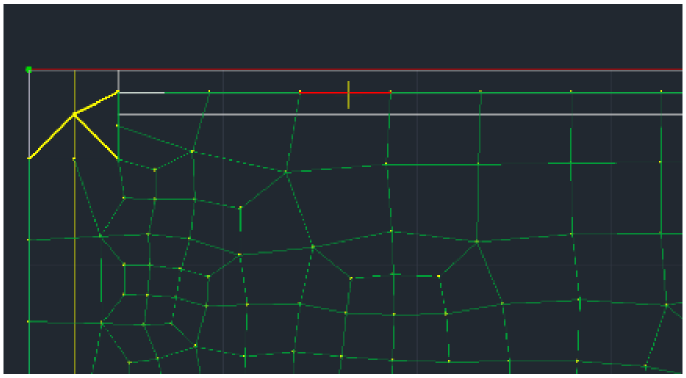
ATTENTION: *In cases where in the plate boundary there are beams, we must ensure the connection of the surface elements of the plate with the members of the perimeter beams. That's why:*

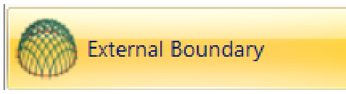
- *The external boundary of the slab should coincides with the axis of the beam*



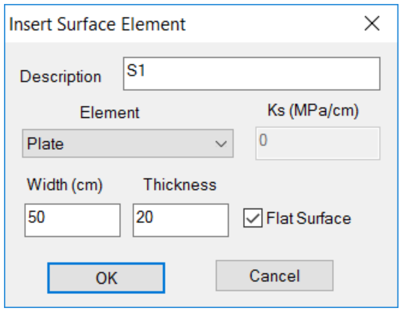
- *after create the mesh and the mathematical model, you should use the command that breaks the beam's members and connects them with the mesh elements.*



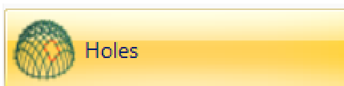




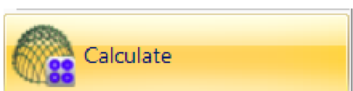
Select the command and left-click on one of the contour lines. Right click and displays a dialog boxes with the surface setting.



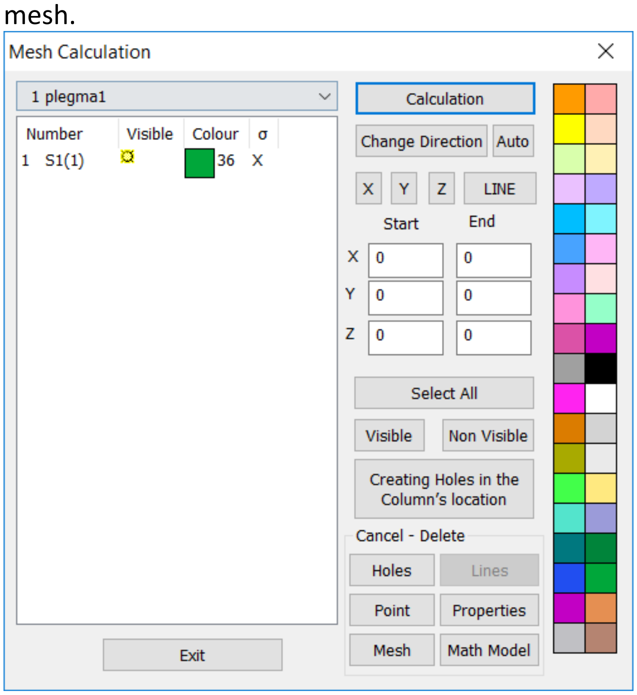
⚠ If there is one or more points where the perimeter is not a closed area, on the screen will appear one or more **X** indicates that point/s.



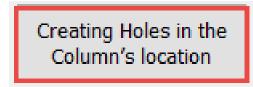
Select Holes command and show the contour of the hole like you did before for the external boundary, left click on a line and right click to complete the command.



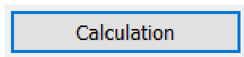
When the form and the properties of the mesh are completed, comes the calculation. Select the command and in the dialog box that opens displays the list of the mesh. In this example there is a mesh S1. The number in brackets (1) shows the number of holes that have been set for the mesh.



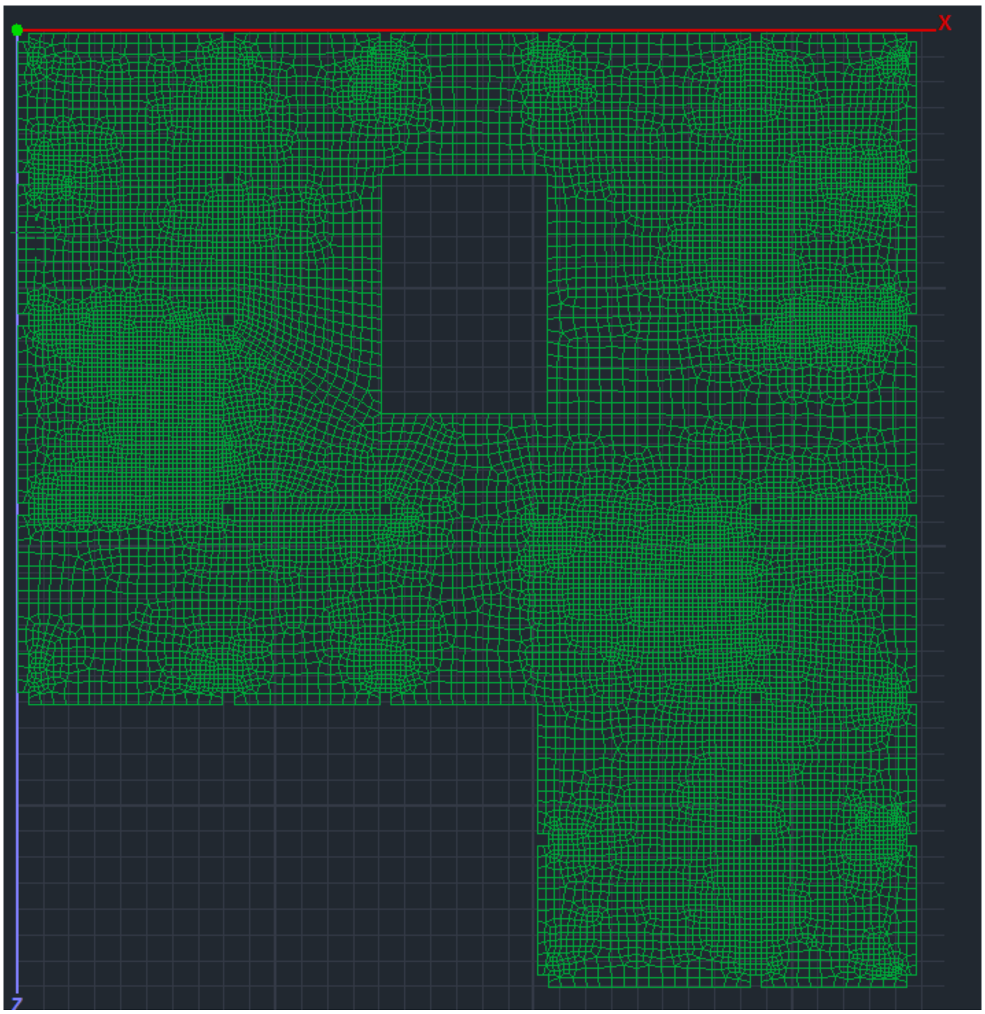
Apart from the central hole should be also present the holes in place of the internal columns.

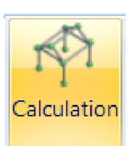
These holes are generated automatically by selecting the command 

and their number is added to the value in brackets.

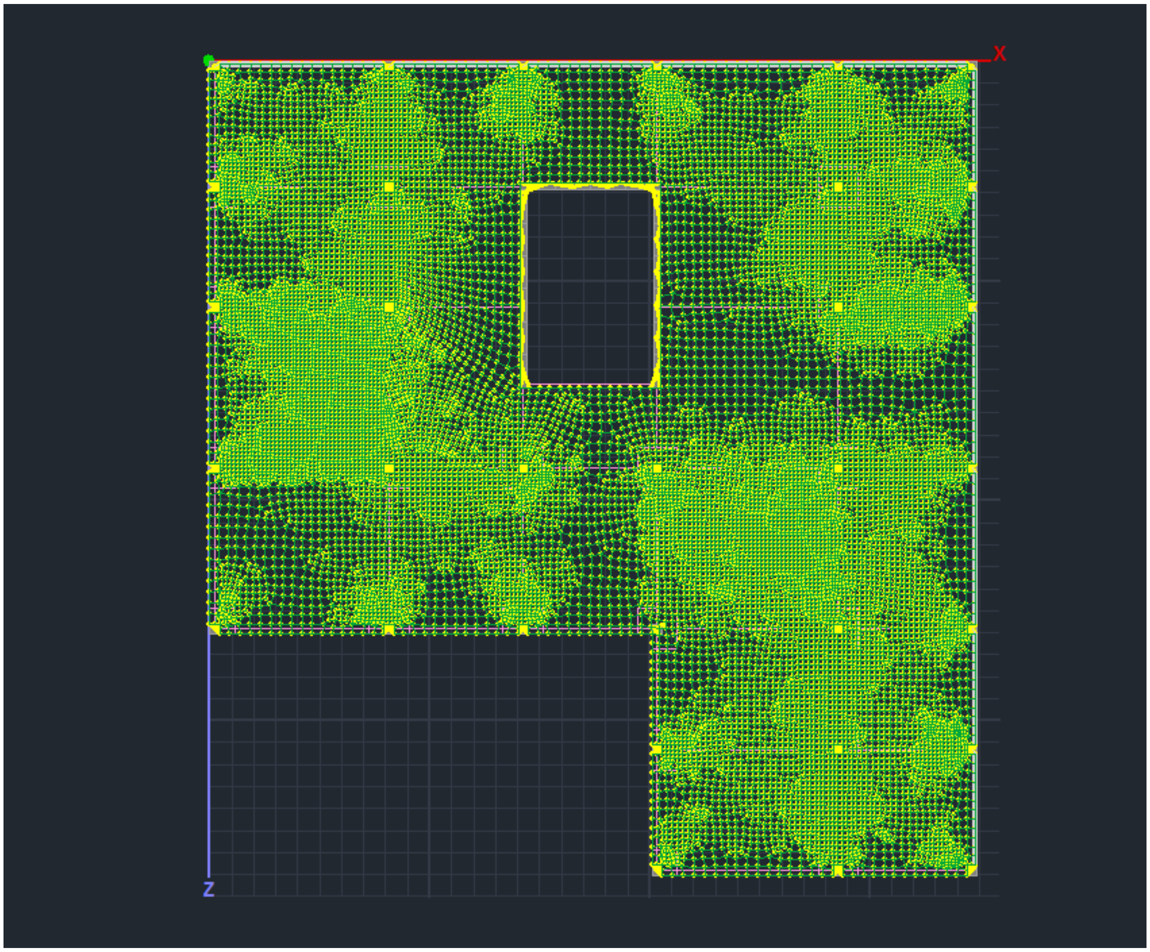
Use 

to calculate the mesh.

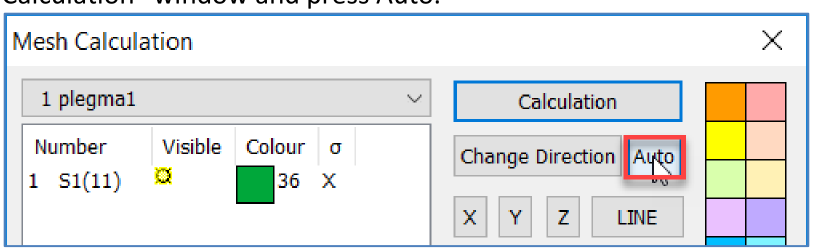




The modeling is completed by the creation of the Mathematical Model.



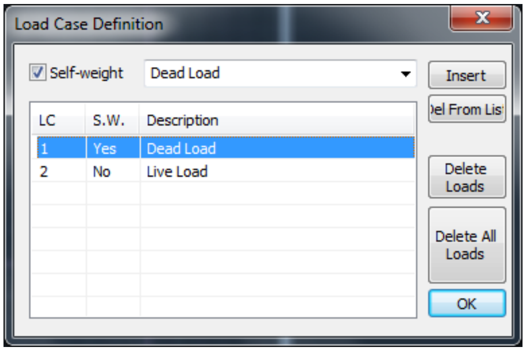
⚠ **NOTE**: After Mesh Mathematical Model creation, always remember to open "Mesh Calculation" window and press Auto.

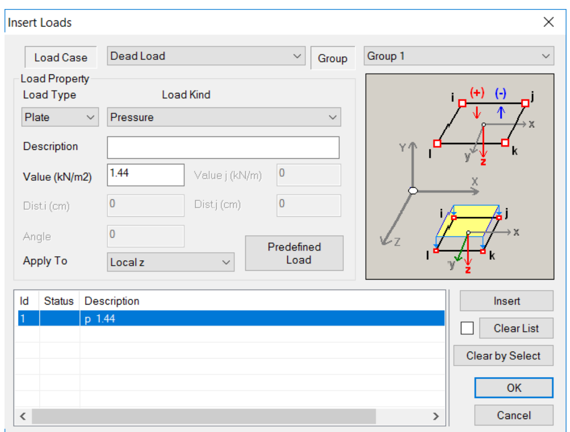


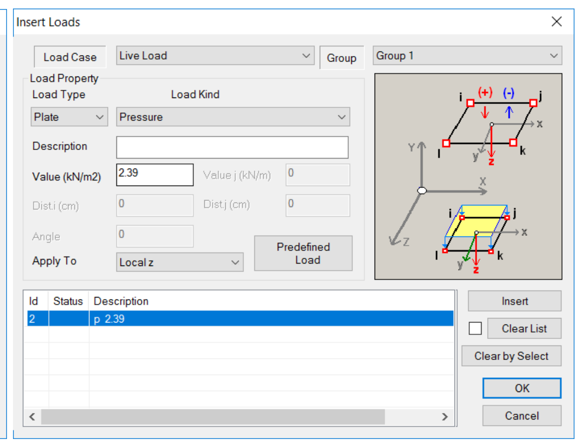
- After **Modeling** follow **Load's Import**, **Analysis** and **Combinations** creation.

2. Loads

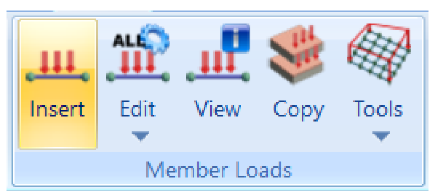
Apply Dead Loads (1,44KN/m2) and Live Loads (2,39KN/m2) to the entire slab. In the value of the Dead Loads The value of the self weight of the slab is not included in the value of Dead Loads (the S.W. is checked in the Dead Load definition.

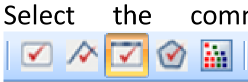


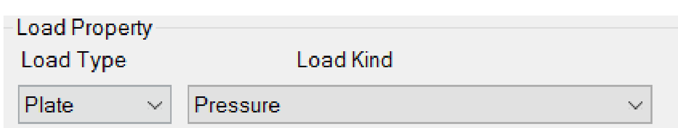




Select the command

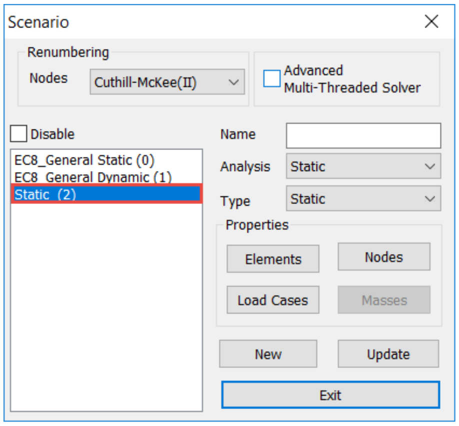


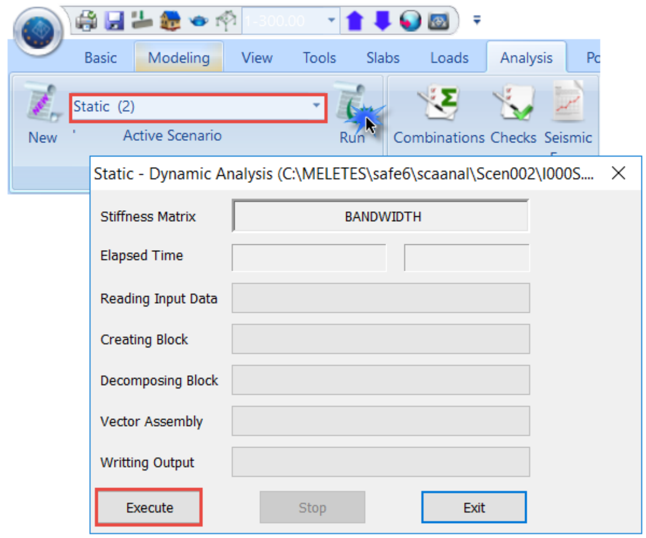
and use the Window  to include all the plan view. Apply Dead (1,44KN/m2) and Live (2,39KN/m2) loads as pressure on the Plate.



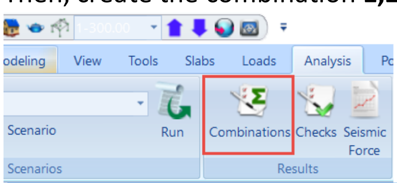
3. Analysis

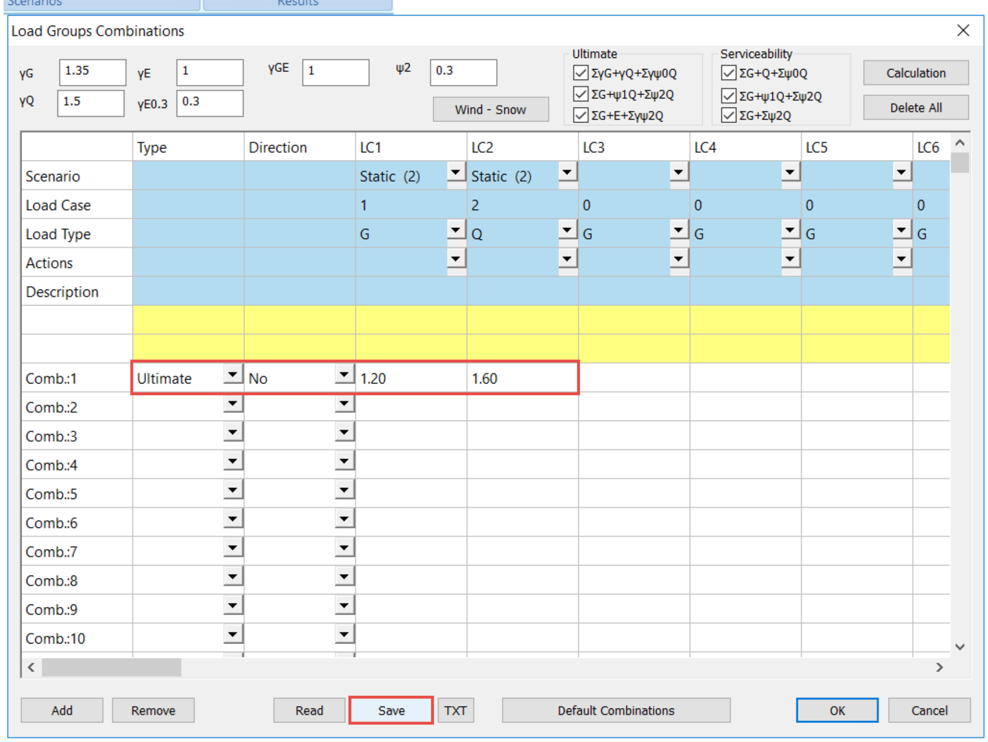
Create a **Static** Scenario and run the simple static analysis.





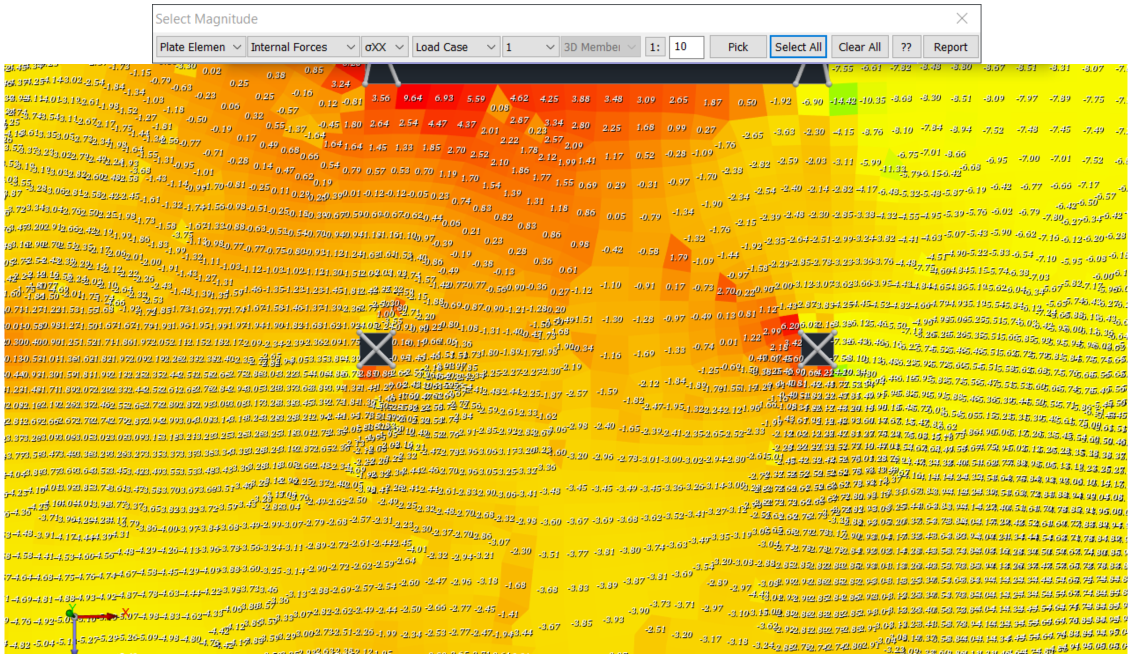
Then, create the combination **1,20G + 1,60Q** in the combinations field and save it.



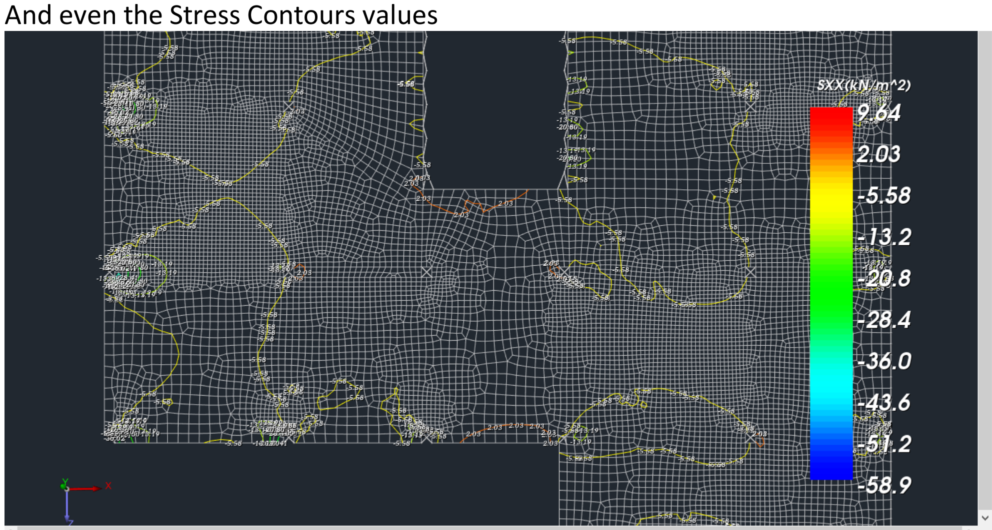


4. Post-Processor

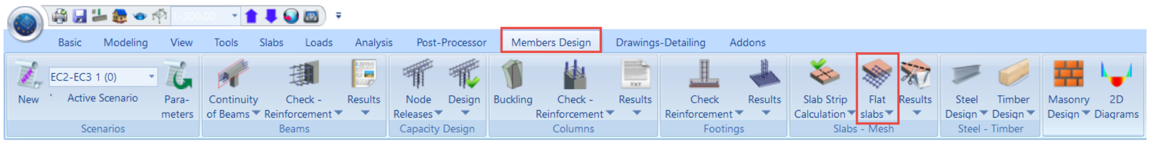
In **Post-Processor** field, you can read the values of the different sizes, either by the color imaging either by reading the values of the selected size in the surface of the mesh element, activating VALUES in the lower horizontal bar.



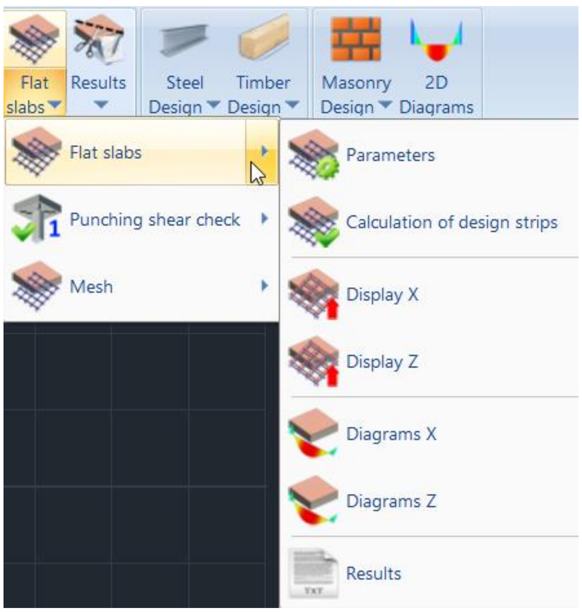
And even the Stress Contours values



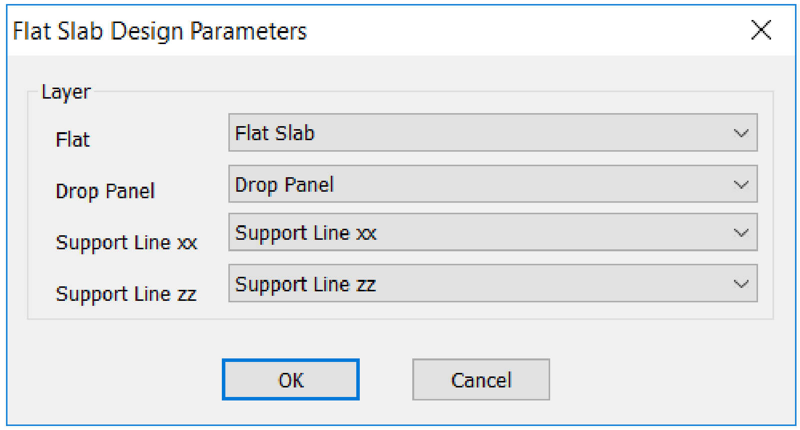
5. Members Design



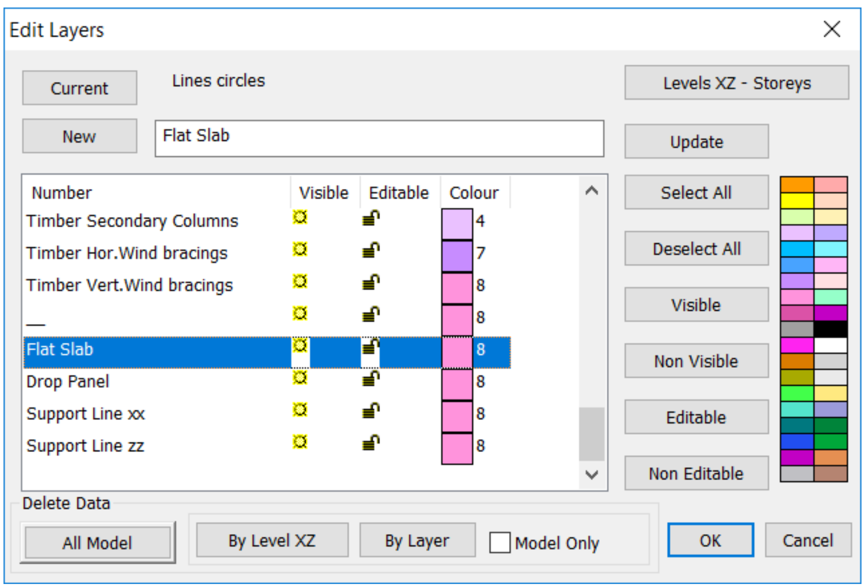
In Members Design field there is the Flat Slab command and the necessary sub commands for solving them.
First load and calculate the combinations and then select the command and start the following procedure :



5.1 Parameters



In the dialog box you define the correlation between Layers.

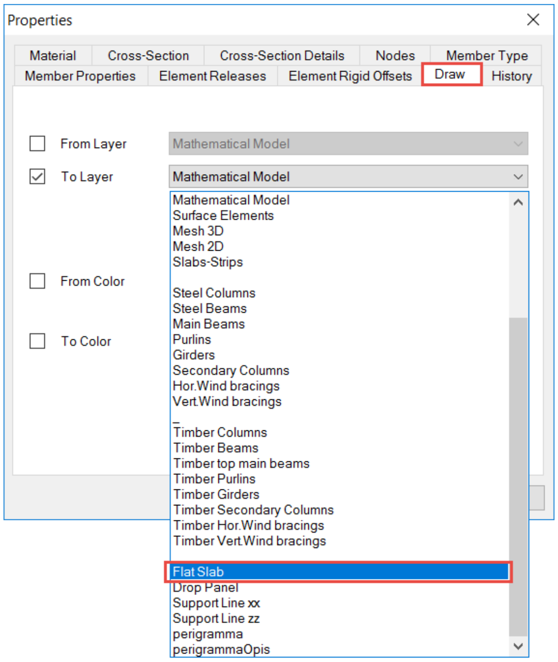


The default Layer's list of SCADA includes the layers related to the flat slabs.

- In "Flat Slab" layer transfer the outline of the slab (including external columns) and correspond it to the Layer "Flat".

Outline the slab using lines.
To transfer the contour lines of the layer "Lines circles" into the layer "Flat Slab",

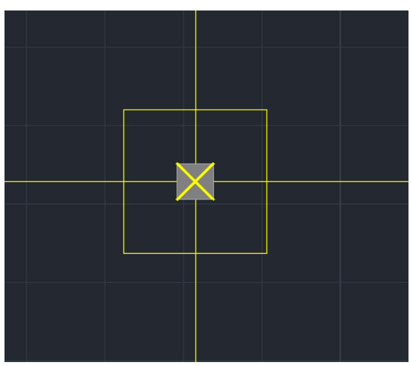
- Frizz all Layers (Not Visible-Not Editable), except " Lines circles "
- Select the command Multiselect Edit (Basic)
- By Left-click select all contour lines
- Right click to complete.
- In the dialog box, in Draw, change layer to Layer "Flat Slab"

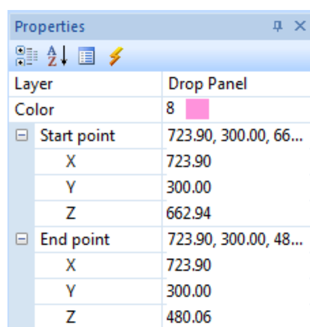


For convenience, you can initially choose the Layer "Flat Slab", immediately after the selection of the command line or Polyline, so the contour lines will belong to the correct layer without need to transfer.

- Respectively, in "Drop Panel" layer transfer the Lines that define the area around the columns, where will increase the thickness of the slab locally.

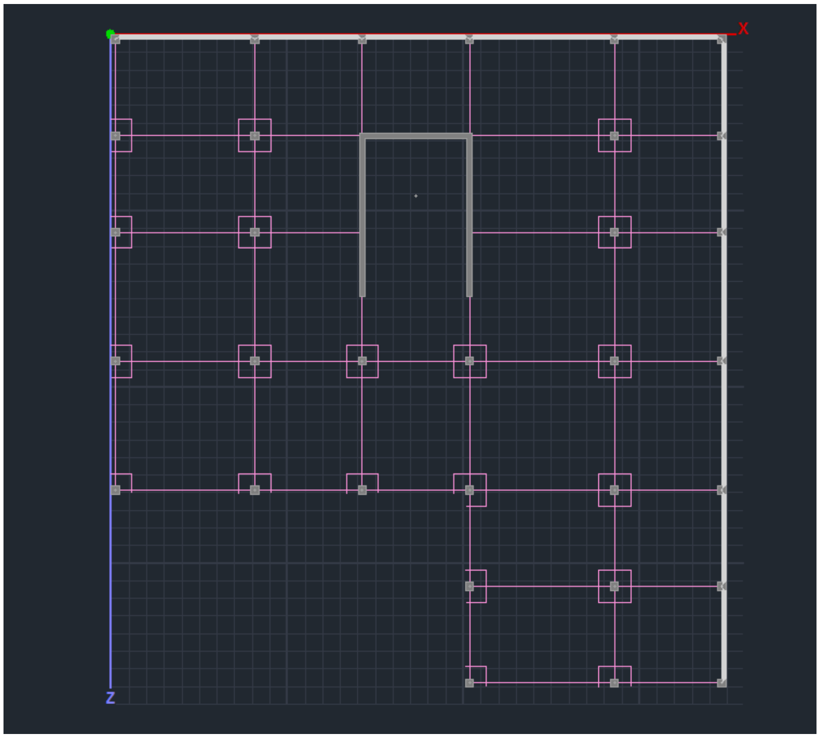
The "Drop Panels" are inserted optionally around the columns of the slab relieving the fatigue from drilling.





- Same procedure to transfer, in "Support Lines xx" and "Support Lines zz" the Lines that define the Support Lines.

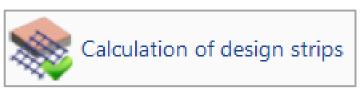
These are lines insert in both X and Z directions between successive points of the slab. Usually connect column's nodes and end on the outline of the slab.



Drop Panels and Support Lines

⚠ Based on the designed Support Lines will be generate the corresponding Design Strips.

5.2 Calculation of design strips

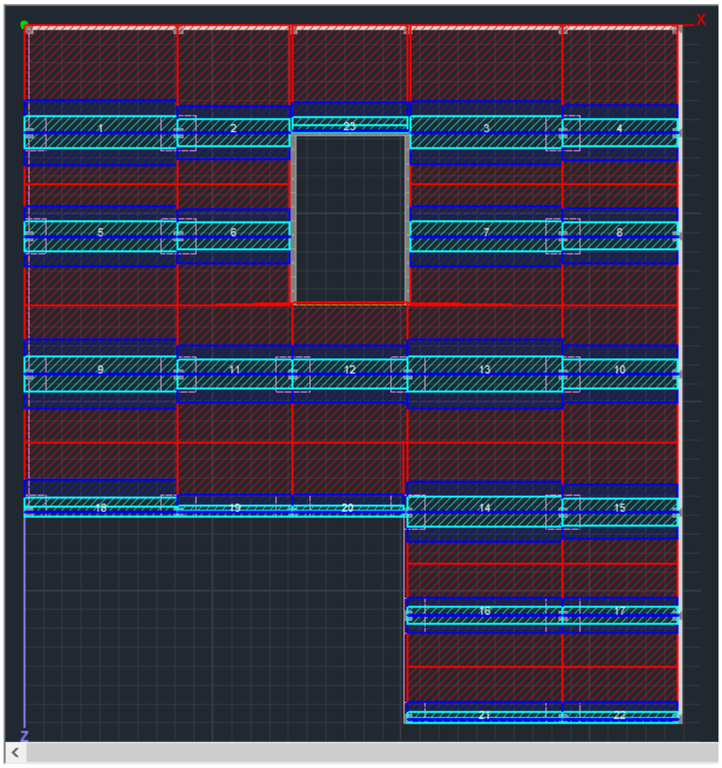


According to Annex I of EC2 flat slab is divided into design strips. These are the areas that are automatically created by the program on both sides of the Support Line, according to Figure I.1 of EC2.
Select the command Calculation of design strips and the program automatically creates them.

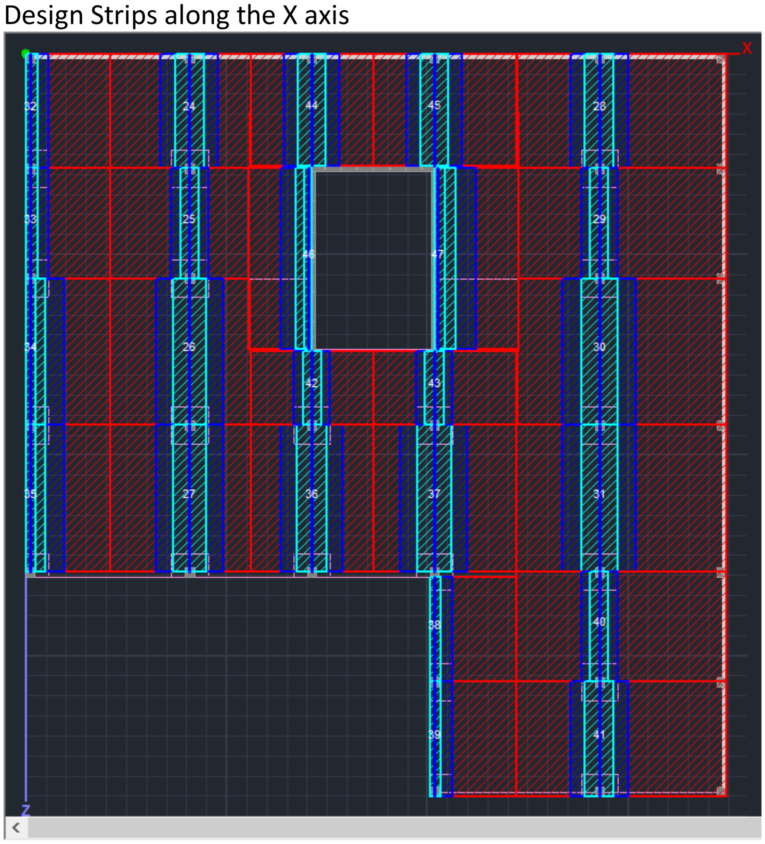
Each Design Strip is divided into sections along its length perpendicular to the Support Line. In each section SCADA integrates the internal forces of finite surface elements intersect. By completing this occur the bending moment around the axis of the section. This intensive value used to calculate the armature in each section.

5.3 Display X, Z

Select to display the Design Strips in both directions.



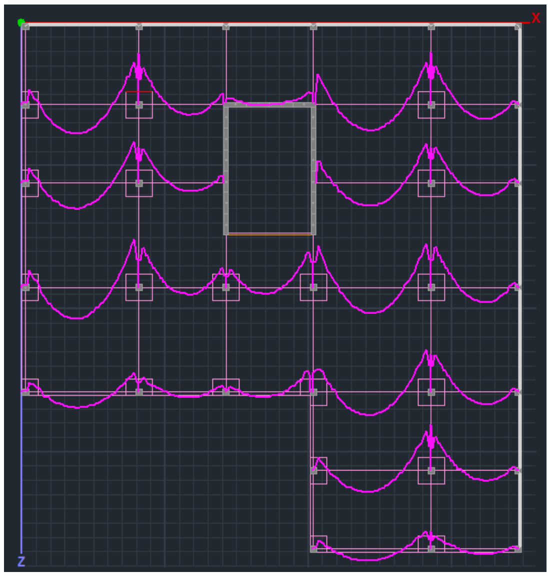
Design Strips along the X axis

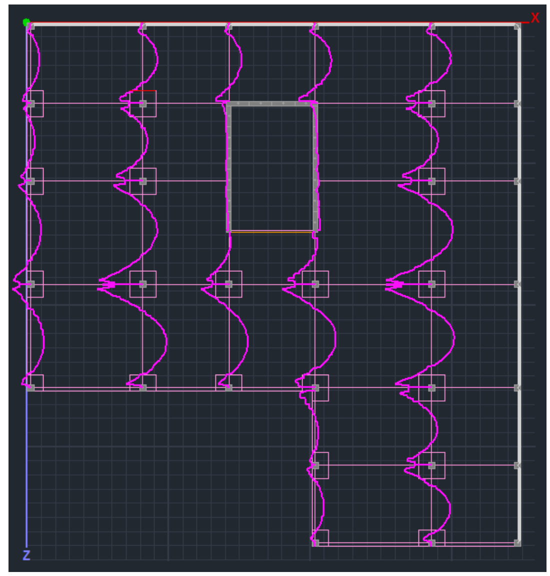


Design Strips along the Z axis

5.4 Diagrams X, Z

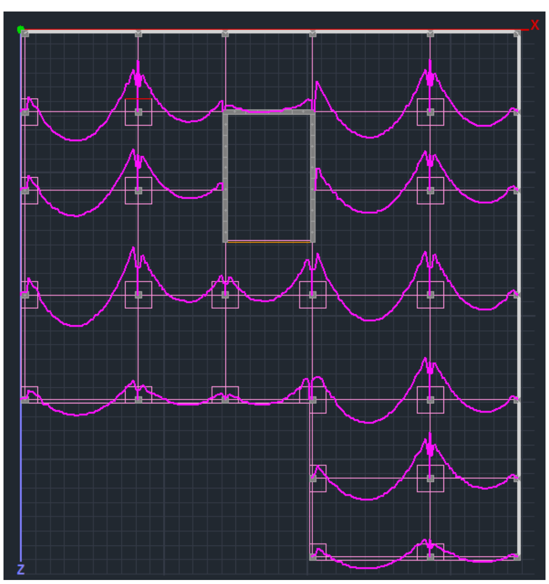
Select Diagrams in both directions to see the corresponding diagrams.

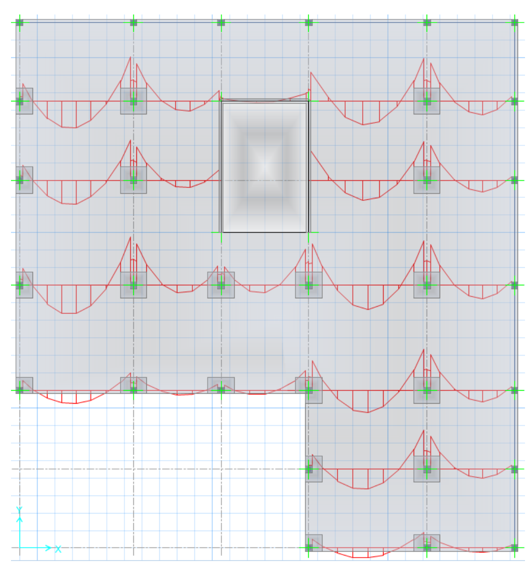




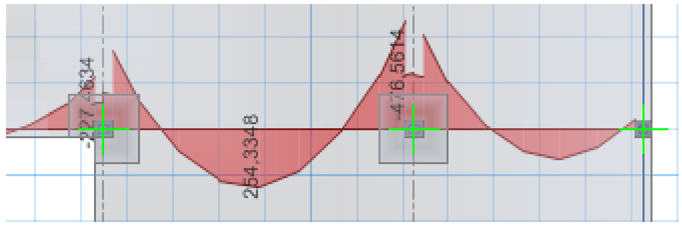
<u>Comparison to the results of CSI Safe 2014</u>

After drawing the same design strips in the Safe (only Column strips are defined (containing the width of the middle strips) to be compatible with the definition of SCADA Pro) we take the following Bending moment diagrams. The diagrams of the two software look identical.





Let’s pay closer attention to the strips 14-15 along X axis (numbering of the strips is shown in a previous figure). Safe 2014 reports a maximum value of bending moment equal to 254,3348 kNm at the middle of span 14.



The corresponding value computed in SCADA Pro is equal to 245.990 kNm, that is a difference of about 3% in the values of the two software (the above value can be found in the folder scades_FlatSlab of the project, file L_14.txt).

5.5 Results

This command opens the Results file through the Report.

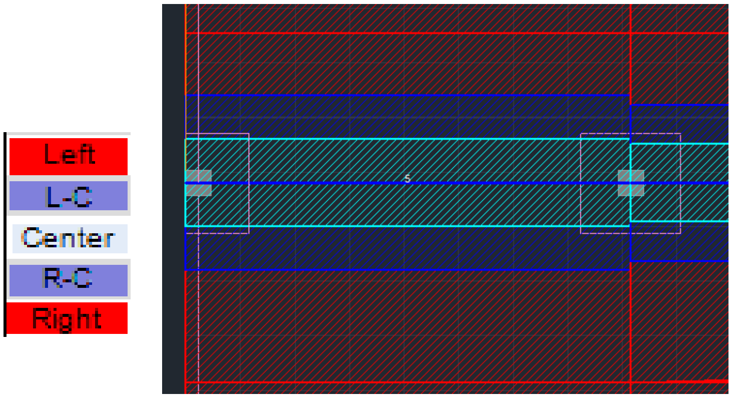
Each page concerns a Strip Line.

Initially described the characteristics of the Strip.

							Page : 1
Strip Calculations							
Description	**Value**	**Units**	**Code**	**Description**	**Value**	**Units**	**Code**
Floor	1			Starting point	corner column		9.4.1&2
# of strip	1			Drop panel	Yes		
Orientation	x-x			*Thickness*	182.88	(cm)	
Length	815.48	(cm)		*Width*		(cm)	
Concrete	C20/25			Finishing point	internal column		9.4.1&2
f_{ck}	20	(MPa)	Table 3.1	Drop panel	Yes		
f_{ctm}	2.20	(MPa)	Table 3.1	*Thickness*	182.88	(cm)	
Steel	S400s			*Width*		(cm)	
f_{yk}	400	(MPa)		Minimum reinforcement			
Cover	20	(mm)		*Tension reinf.*	0.00143	(cm²/m)	9.2.1.1(1)
Slab thickness	0.25	(cm)		*Compression reinf. (% of span reinf.)*	25	%	9.3.1.2

Then displayed the reinforcement results above and below in detail for each zone, dividing them into sub-zones.

- Left-Right -> red zone
- L-C R-C-> blu zone
- Center-> light blue zone



Analysis Results and Reinforcement — Top

	203.87 cm (L_{start})					407.74 cm (L_{centre})				
Zone	**M (kNm)**	**Width (cm)**	**$A_{s,rqd}$ (cm^2/m)**	**$A_{s,prvd}$ (cm^2/m)**	**Φ/s**	**M (kNm)**	**Width (cm)**	**$A_{s,rqd}$ (cm^2/m)**	**$A_{s,prvd}$ (cm^2/m)**	**Φ/s**
Left		400.0		3.246	8/15		401.1		0.812	8/20
L-C		400.0		3.246	8/15		85.3		1.763	8/20
Center	-80.283	46.0	27.271	27.271	14/5		170.5		6.818	8/7
R-C							85.3		2.043	8/20
Right							103.8		1.471	8/20

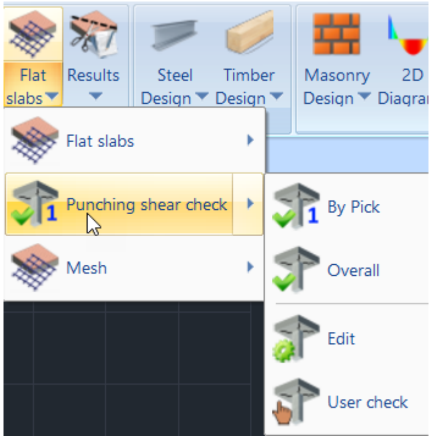
	203.87 cm (L_{end})				
Zone	**M (kNm)**	**Width (cm)**	**$A_{s,rqd}$ (cm^2/m)**	**$A_{s,prvd}$ (cm^2/m)**	**Φ/s**
Left	-88.070	401.1	2.873	3.246	8/15
L-C	-44.824	85.3	7.054	7.054	8/7
Center	-152.524	170.5	12.422	12.422	10/6
R-C	-51.588	85.3	8.172	8.172	8/6
Right	-45.848	103.8	5.886	5.886	8/8

Analysis Results and Reinforcement — Bottom

	203.87 cm (L_{start})					407.74 cm (L_{centre})				
Zone	**M (kNm)**	**Width (cm)**	**$A_{s,rqd}$ (cm^2/m)**	**$A_{s,prvd}$ (cm^2/m)**	**Φ/s**	**M (kNm)**	**Width (cm)**	**$A_{s,rqd}$ (cm^2/m)**	**$A_{s,prvd}$ (cm^2/m)**	**Φ/s**
Left	9.207	400.0	0.294	0.812	8/20	70.543	401.1	2.293	3.246	8/15
L-C	9.207	400.0	0.294	0.844	8/20	21.929	85.3	3.377	3.377	8/14
Center	80.591	46.0	27.408	27.408	14/5	43.857	170.5	3.377	3.377	8/14
R-C						21.929	85.3	3.377	3.377	8/14
Right						25.982	103.8	3.284	3.284	8/15

	203.87 cm (L_{end})				
Zone	**M (kNm)**	**Width (cm)**	**$A_{s,rqd}$ (cm^2/m)**	**$A_{s,prvd}$ (cm^2/m)**	**Φ/s**
Left	17.335	401.1	0.555	0.812	8/20
L-C	6.505	85.3	0.984	0.984	8/20
Center	23.135	170.5	1.762	1.762	8/20
R-C	3.583	85.3	0.539	0.844	8/20
Right	3.311	103.8	0.409	0.821	8/20

6. Punching shear checks



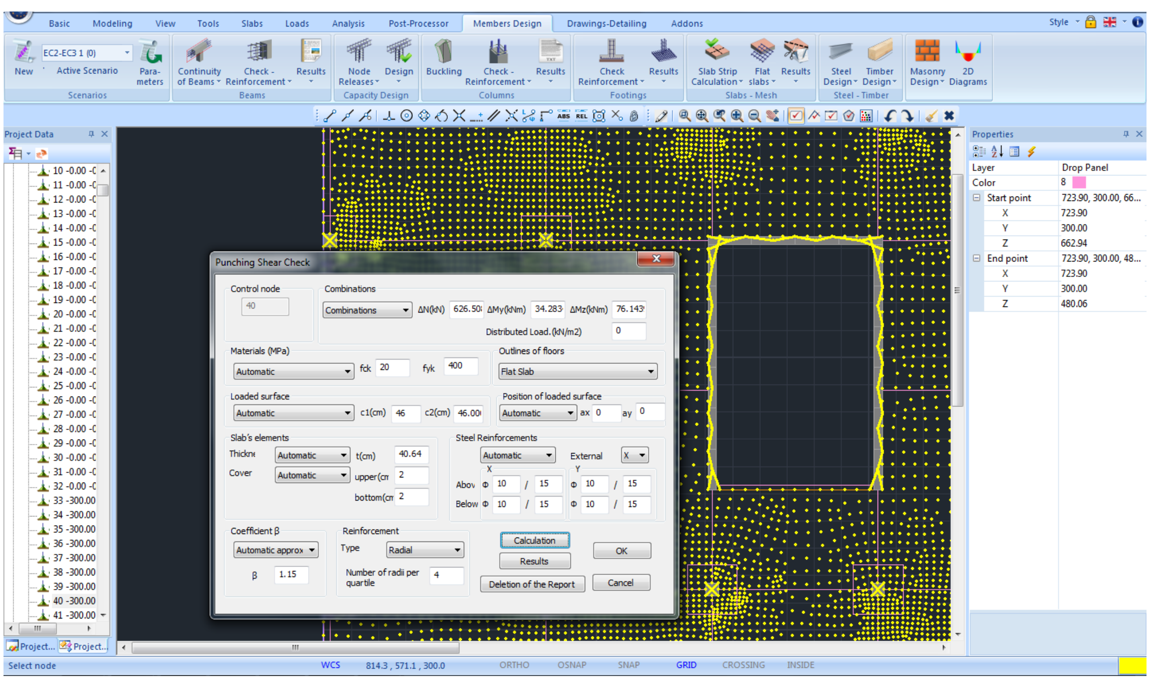
Added in the new version also the punching checks by EC2. The check is part of the check and design process of the flat slabs which runs automatically, but can also run as an individual check for any column. All data can be set automatically or even manually.

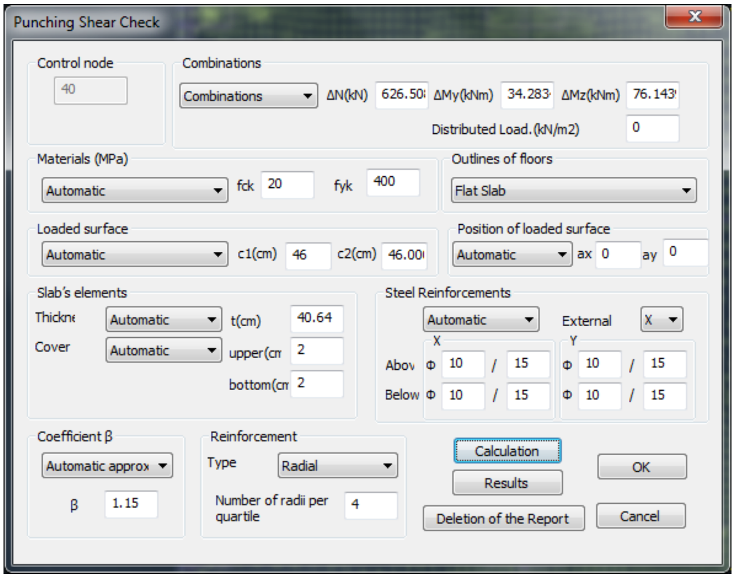
6.1 By Pick

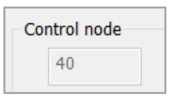
Select By Pick command with the left mouse button show the node of a column and right-click to open the dialog box where you specify all the necessary parameters.

⚠ In this example we select the node of an Internal column, node 40.

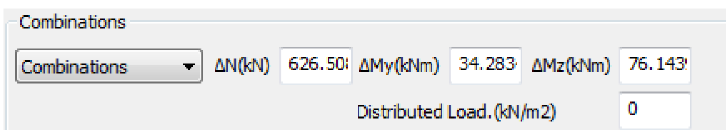
Here are explanations for the other positions of the columns on the perimeter of the slab:





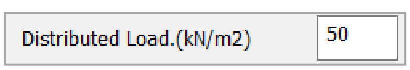


Automatically fill in with the number of the selected node and is not editable.



In Combinations field:

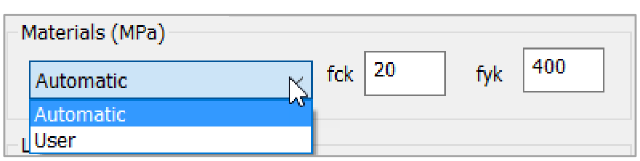
- Choosing Combinations, the program finds automatically the combination that gives the worst Axial resulting, displaying ΔN value the with the corresponding moments.
- Choosing User, enables the user values for axial and moments, in their respective fields, and the definition of a Distributed Load



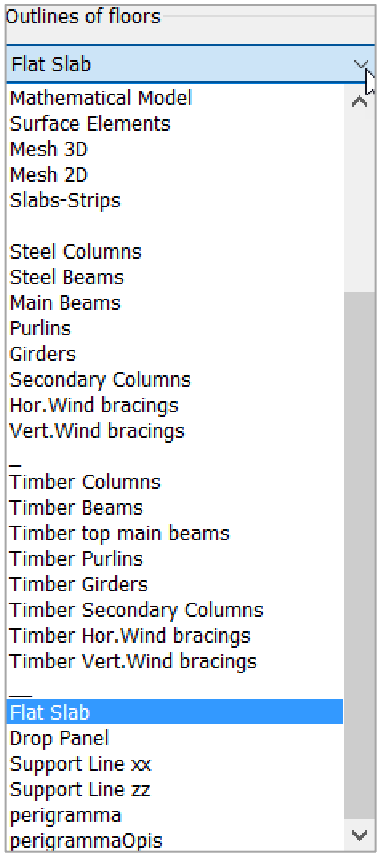
that works "relieving" the slab at that point, so the shear calculation be impaired relative to the original.

Shear force ($V_{Ed,init.}$)	626.5	(kN)
Distributed load (p)	50.0	(kN/m^2)
Reduced shear force ($V_{Ed,fin.}$)	478.8	(kN)

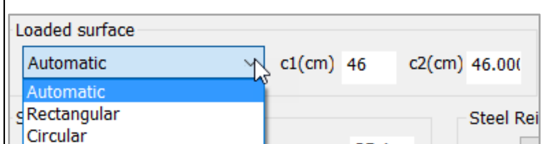
⚠ In this example we consider a single combination 1,20G + 1,60Q and therefore the values of the internal forces resulting from that place.



In the field Material coefficients fck and fyk filled automatically with the Automatic option or defined by the user with the User option.



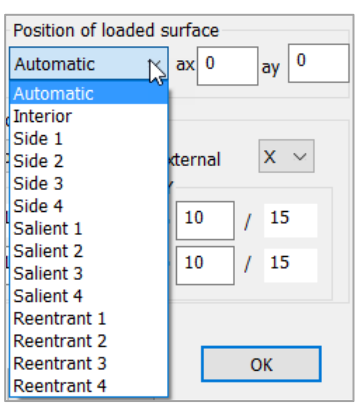
In Outlines of Floors, choose the Layer containing the contour of the slab including the columns located in the outline.
Then select the layer Flat Slab that includes exactly the lines that define the overall contour of the slab.



As loaded surface is defined the equivalent surface of the selected column.

Choosing:

- Automatic, the program calculates the surface of any column-shaped by reduction in the equivalent rectangular with the corresponding dimensions c1 and c2.
- Rectangular, the user defines his dimensions c1 and c2 for calculating the loaded rectangular surface
- Circular, the user defines his diameter c1 for calculating the loaded circular surface

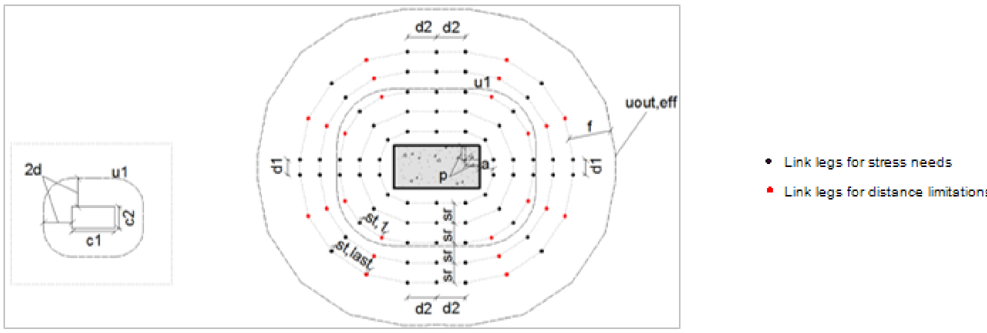


The Position of loaded surface may be determined either automatically or selectively. It depends on the contour of the slab and the position of the selected column in it.

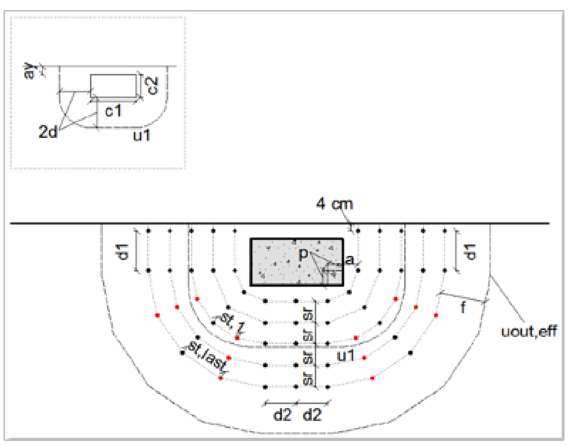
The proposed positions are:

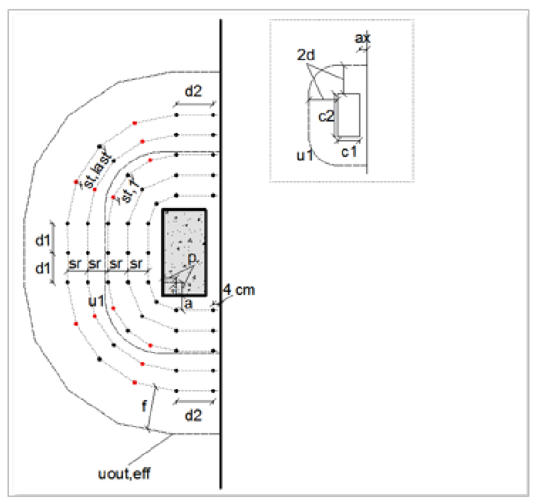
- Interior
- Side: 4 directions
- Salient: 4 directions
- Reentrant: 4 directions

Select the position of the selected column and set the distances from the perimeter, ax and ay, (except indoor) according to the following figures:



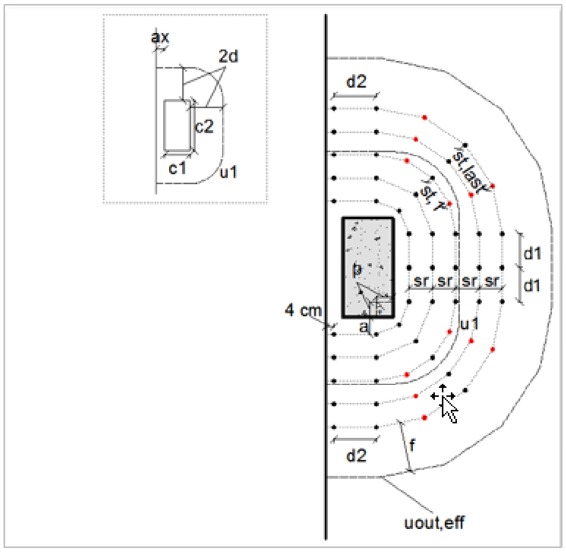
INTERIOR



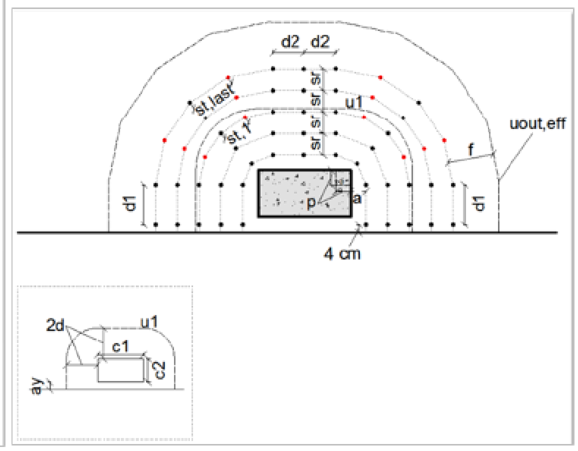


SIDE 1

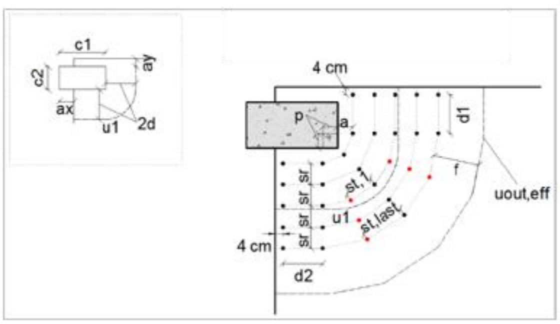
SIDE 2



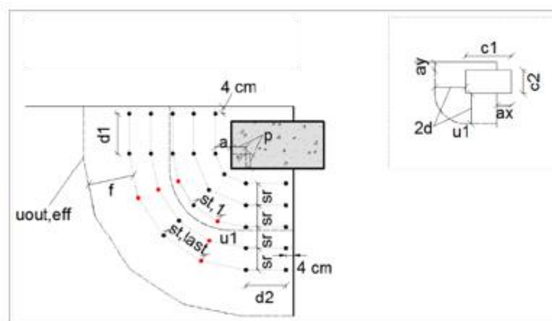
SIDE 4



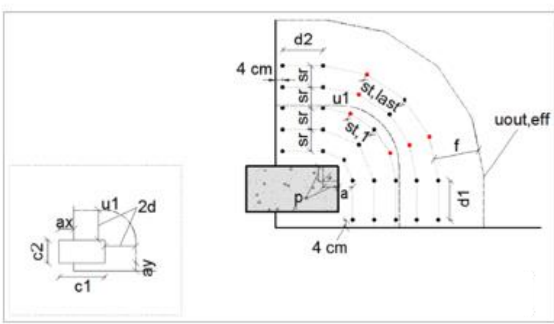
SIDE 3



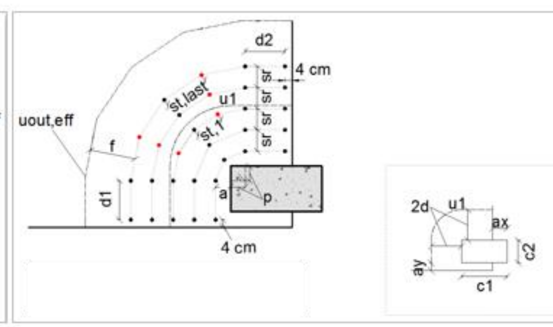
SALIENT 1



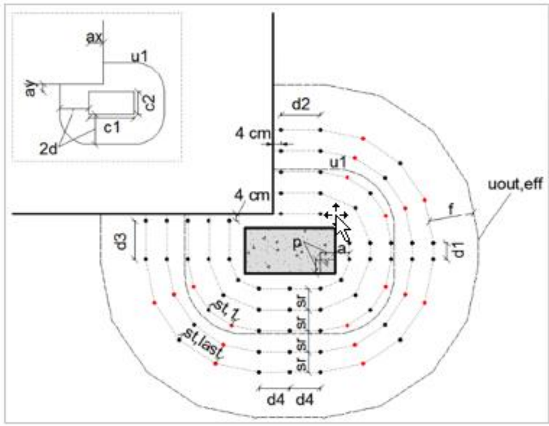
SALIENT 2



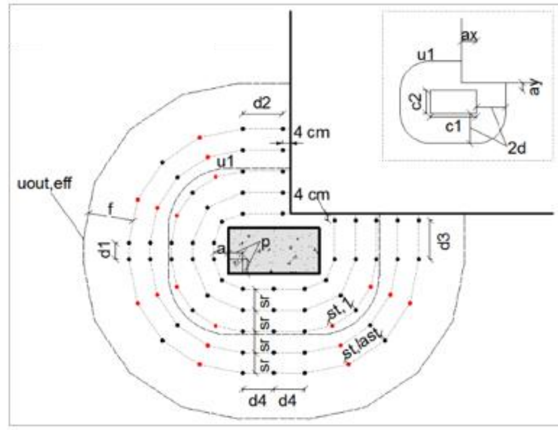
SALIENT 4



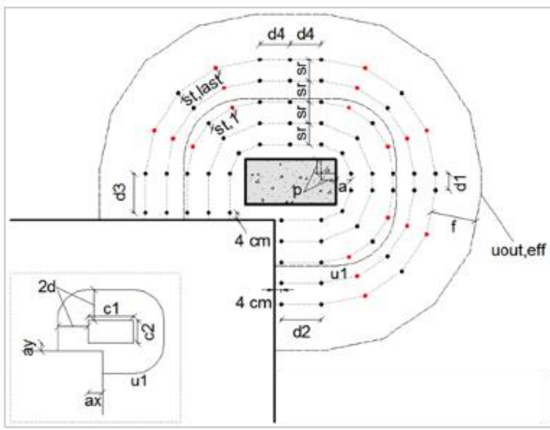
SALIENT 3



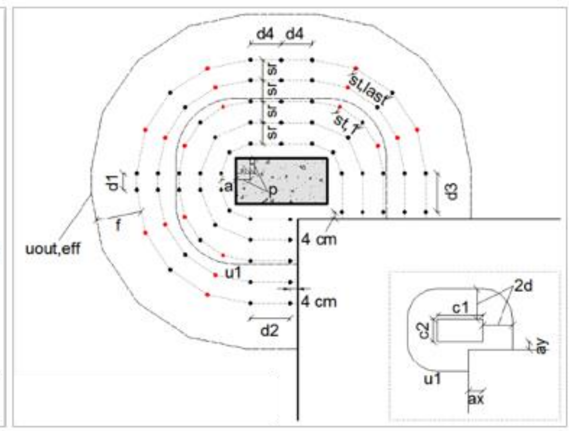
REENTRANT 1



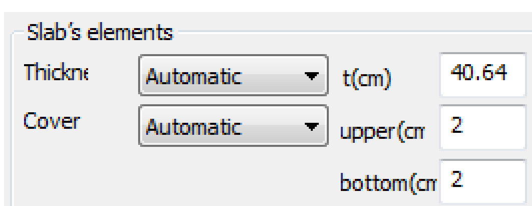
REENTRANT 2



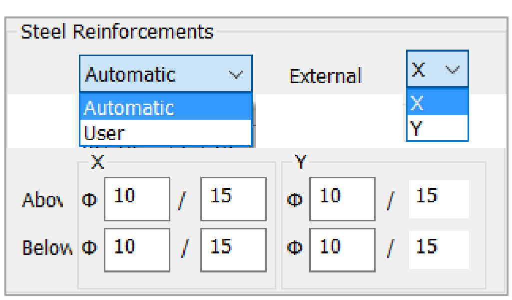
REENTRANT 4



REENTRANT 3



The Thickness and Cover of the Slab either considered Automatically or modified by the User, by typing the corresponding values for Thickness and Cover, upper and bottom, of the slab.

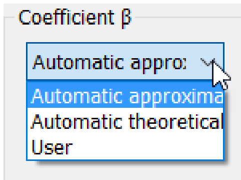


In Steel Reinforcements identifies the longitudinal reinforcement resulting from the calculation of flat slabs in the selected column region.

With the Automatic option taking into account the longitudinal reinforcement:

- Above for ΔN positive (+ΔN) (e.g. penthouse slab)
- Below for ΔN negative (-ΔN) (e.g. foundation slab)

Selection External X or Y determines the direction of the outer reinforcement of the slab longitudinal reinforcement mesh (either for the up or down mesh).



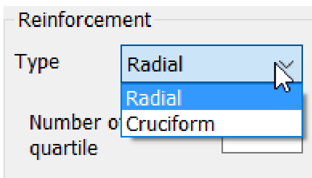
The Coefficient β for punching calculation, can be calculated automatically in two ways:

- Automatic approximation or
- Automatic theoretical.

⚠ The Approximation way is a function of the position of the loaded area and the ax, ay.

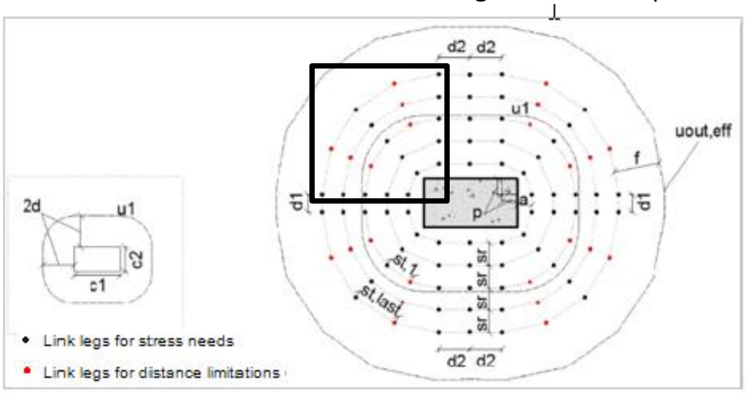
⚠ The Theoretical way is a function of the Moments My, Mz.

User selection allows the introduction of any value for the coefficient β.

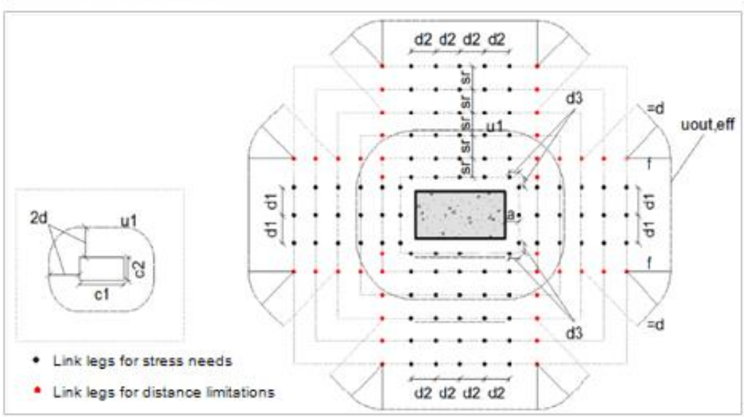


Finally, in Reinforcement set the layout type of punching reinforcement choosing between Radial and Cruciform.
For the Radial layout set the number of radii per quartile.

In the figure below the black box contains one quartile of the radial layout. The first perimeter has three radii of reinforcement, while there is an increase on the 3rd perimeter to 5 radii due to distance limitations set by the code. SCADA Pro automatically checks whether the distance limitations are satisfied on the first perimeter, and increases the number of radii when needed (even when the user’s choice is not enough for the first perimeter).



Radial reinforcement Layout



Cruciform reinforcement Layout

Calculation The Calculation command performs all the necessary checks to punch, taking into account all the above parameters.

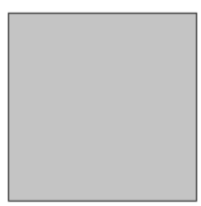
Results This command displays the results file:

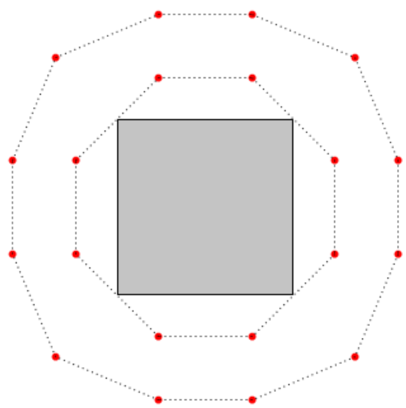
Input Data					
Description	**Value**	**Units**	**Description**	**Value**	**Units**
Level - Storey	1		Eccntr. factor (β) (EC2-6.4.3)	1.150	
# of node	40		Slab depth	40.6	(cm)
Combination	1		Cover of reinforcement	2.0	(cm)
Shear force ($V_{Ed,init.}$)	626.5	(kN)	Bar size (outer layer)	10	(mm)
Distributed load (p)	0.0	(kN/m^2)			
Reduced shear force ($V_{Ed,fin.}$)	626.5	(kN)	Spacing of bars (outer layer)	15.0	(cm)
Bending moment (M_x)	34.3	(kNm)	Bar size (second layer)	10	(mm)
Bending moment (M_y)	76.1	(kNm)			
Shape of loaded area	Rectangular		Spacing of bars (second layer)	15.0	(cm)
c_1 length (along x axis)	46.0	(cm)			
c_2 length (along y axis)	46.0	(cm)	Concrete (f_{ck})	20.0	(MPa)
c diameter		(cm)	Steel (f_{yk})	400.0	(MPa)
Position of loaded area	Interior		Reinforcement pattern	Radial	
Dist. of slab perim. along x (a_x)		(cm)	# of radii of reinforcement in a quadrant (circular pattern)	2	
Dist. of slab perim. along y (a_y)		(cm)			

Input Data: list of all elements identified in the previous window and required to punching check.

Punching Reinforcement Layout: according to predetermined parameters and if, there is reinforcement requirement.

⚠ In this example there is no requirement to punching reinforcement, so there is no reinforcement schematic arrangement (a random radial layout with two perimeters is also included just for observation).





In **Check results** there are two different checks:

If the 1st check presents *Sufficiency*, then the 2nd check specifies the requirement or not of punching reinforcement.

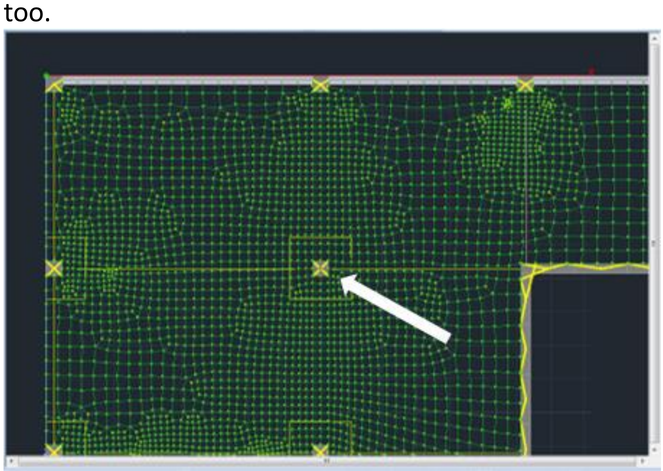
							Page : 2
Check results							
Description	**Value**	**Units**	**EC2**	**Description**	**Value**	**Units**	**EC2**
Effective depth of slab (d)	37.6	(cm)	(eq6.32)	Basic control perimeter (u_1)	657.0	(cm)	(fig6.15)
Perimeter of the loaded area (u_0)	184.0	(cm)	(eq6.53)	Design value of the shear stress at u_1 ($v_{Ed,1}$)	0.291	(MPa)	(eq6.38)
Design value of the shear stress at u_0 ($v_{Ed,0}$)	1.040	(MPa)	(eq6.38)	Punch. shear resistance without shear reinforcement ($v_{Rd,c}$)	0.356	(MPa)	(eq6.47)
Maximum punching shear resistance ($v_{Rd,max}$)	3.680	(MPa)	(eq6.53)	Constant (v_{min})	0.356	(MPa)	(eq6.3)
1st check: $v_{Ed,0} \leq v_{Rd,max}$	Sufficiency			2nd check: $v_{Ed,1} \leq v_{Rd,c}$	No reinforcement is required		

							Page : 2
Check results							
Description	**Value**	**Units**	**EC2**	**Description**	**Value**	**Units**	**EC2**
Effective depth of slab (d)	37.6	(cm)	(eq6.32)	Basic control perimeter (u_1)	657.0	(cm)	(fig6.15)
Perimeter of the loaded area (u_0)	184.0	(cm)	(eq6.53)	Design value of the shear stress at u_1 ($v_{Ed,1}$)	0.744	(MPa)	(eq6.38)
Design value of the shear stress at u_0 ($v_{Ed,0}$)	2.657	(MPa)	(eq6.38)	Punch. shear resistance without shear reinforcement ($v_{Rd,c}$)	0.356	(MPa)	(eq6.47)
Maximum punching shear resistance ($v_{Rd,max}$)	3.680	(MPa)	(eq6.53)	Constant (v_{min})	0.356	(MPa)	(eq6.3)
1st check: $v_{Ed,0} \leq v_{Rd,max}$	Sufficiency			2nd check: $v_{Ed,1} \leq v_{Rd,c}$	Reinforcement necessary: - add punching shear reinforcement - increase slab longitudinal reinforcement		

If the 1st check presents *Insufficiency*, then the 2nd check is not performed and some interventions are proposed for the slab.

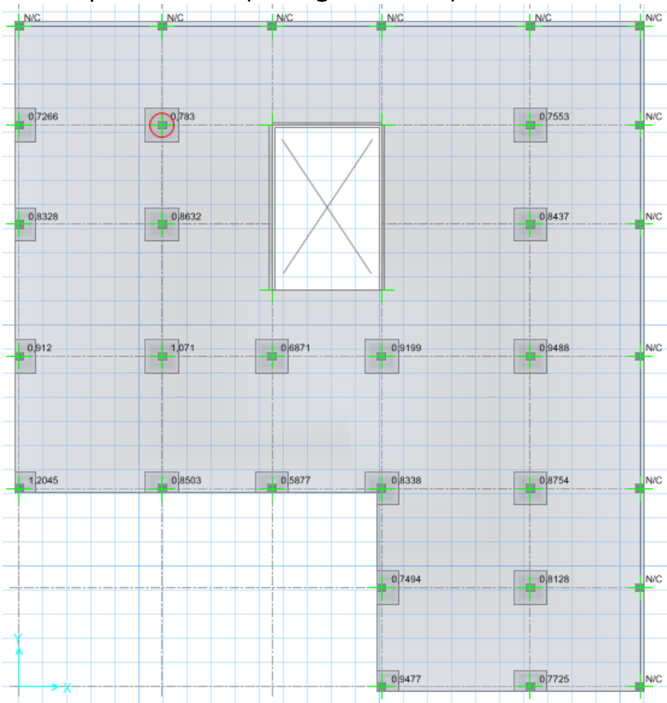
							Page : 2
Check results							
Description	**Value**	**Units**	**EC2**	**Description**	**Value**	**Units**	**EC2**
Effective depth of slab (d)	37.6	(cm)	(eq6.32)	Basic control perimeter (u_1)	657.0	(cm)	(fig6.15)
Perimeter of the loaded area (u_0)	184.0	(cm)	(eq6.53)	Design value of the shear stress at u_1 ($v_{Ed,1}$)	1.860	(MPa)	(eq6.38)
Design value of the shear stress at u_0 ($v_{Ed,0}$)	6.642	(MPa)	(eq6.38)	Punch. shear resistance without shear reinforcement ($v_{Rd,c}$)	0.356	(MPa)	(eq6.47)
Maximum punching shear resistance ($v_{Rd,max}$)	3.680	(MPa)	(eq6.53)	Constant (v_{min})	0.356	(MPa)	(eq6.3)
1st check: $v_{Ed,0} \leq v_{Rd,max}$	Insufficient. - increase size of the loaded area - increase slab depth - use of concrete of a higher quality			2nd check: $v_{Ed,1} \leq v_{Rd,c}$			

As mentioned before, in this example (node 40 shown in the figure below), the level of loading in not as high as to need punching shear reinforcement. In other words, the 2nd check is sufficient too.



The punching shear check for this column gives a capacity ratio equal to 0.291/0.356 = 0.81.

Safe 2014 also provides with a capacity ratio for the columns (figure below). For the current case the capacity ratio computed is 0.783 (difference between software of about 4%).
Let's consider the column situated two places below along Z (node 38). Safe calculates a capacity ratio equal to 1.071 (see figure below).

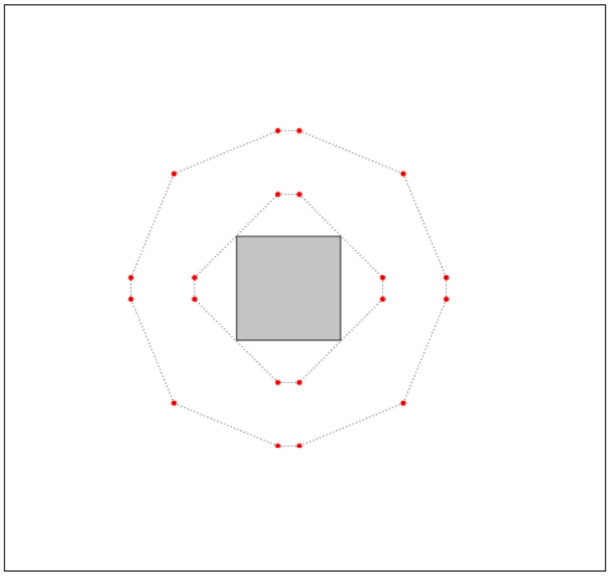


SCADA also computes a capacity ration above 1.0 (equal to 0.394/0.356 = 1.11) meaning that punching shear reinforcement is needed. The results are presented below:

Page : 1

Input Data

Description	Value	Units	Description	Value	Units
Level - Storey	1		Eccntr. factor (β) (EC2-6.4.3)	1.150	
# of node	38		Slab depth	40.6	(cm)
Combination	1		Cover of reinforcement	2.0	(cm)
Shear force ($V_{Ed,init}$)	846.7	(kN)	Bar size (outer layer)	10	(mm)
Distributed load (p)	0.0	(kN/m^2)			
Reduced shear force ($V_{Ed,fin.}$)	846.7	(kN)	Spacing of bars (outer layer)	15.0	(cm)
Bending moment (M_x)	-32.7	(kNm)	Bar size (second layer)	10	(mm)
Bending moment (M_y)	104.4	(kNm)			
Shape of loaded area	Rectangular		Spacing of bars (second layer)	15.0	(cm)
c_1 length (along x axis)	46.0	(cm)			
c_2 length (along y axis)	46.0	(cm)	Concrete (f_{ck})	20.0	(MPa)
c diameter		(cm)	Steel (f_{yk})	400.0	(MPa)
Position of loaded area	Interior		Reinforcement pattern	Radial	
Dist. of slab perim. along x (a_x)		(cm)	# of radii of reinforcement in a quadrant (circular pattern)	2	
Dist. of slab perim. along y (a_y)		(cm)			



The first page of the results shows the input data as well as the reinforcement layout. The choice here was radial arrangement with two radii per quartile.

Page : 2

Check results

Description	Value	Units	EC2	Description	Value	Units	EC2
Effective depth of slab (d)	37.6	(cm)	(eq6.32)	Basic control perimeter (u_1)	657.0	(cm)	(fig6.15)
Perimeter of the loaded area (u_0)	184.0	(cm)	(eq6.53)	Design value of the shear stress at u_1 ($v_{Ed,1}$)	0.394	(MPa)	(eq6.38)
Design value of the shear stress at u_0 ($v_{Ed,0}$)	1.406	(MPa)	(eq6.38)	Punch. shear resistance without shear reinforcement ($v_{Rd,c}$)	0.356	(MPa)	(eq6.47)
Maximum punching shear resistance ($v_{Rd,max}$)	3.680	(MPa)	(eq6.53)	Constant (v_{min})	0.356	(MPa)	(eq6.3)
1st check: $v_{Ed,0} \leq v_{Rd,max}$	Sufficiency			2nd check: $v_{Ed,1} \leq v_{Rd,c}$	Reinforcement necessary: - add punching shear reinforcement - increase slab longitudinal reinforcement		

Detailing results

Description	Value	Units	EC2
Perimeter $u_{out,ef}$	727.0	(cm)	(eq6.54)
(a) - Distance of 1st perimeter of reinforcement from the loaded area	18.8	(cm)	
Limit: $0.3 \cdot d \leq a \leq 0.5 \cdot d$	11.3<=a<=18.8		(9.4.3)
(f) - Distance of last perimeter of reinforcement from $u_{out,ef}$	56.6	(cm)	
Limit: $k \cdot d = 1.5 \cdot d$	56.5	(cm)	(6.4.5)
(s_r) - Radial distance of the perimeters of reinforcement	28.0	(cm)	
Limit: $0.75 \cdot d$	28.2	(cm)	(9.4.3)
($s_{t,1}$) - Tangential distance between link legs on the u_1 perimeter	91.8	(cm)	
Limit: $1.5 \cdot d$	56.5	(cm)	(9.4.3)
Distance (p)	18.1	(cm)	
Distance (d_1)	9.9	(cm)	
Distance (d_2)	9.9	(cm)	

Περιγραφή	Τιμή	Μονάδες	EC2
Distance (d_3)		(cm)	
Distance (d_4)		(cm)	
Angle (φ)	90.0	°	
($s_{t,last}$) - Tangential distance between link legs on the last perimeter	91.8	(cm)	
Limit: $2.0 \cdot d$	75.3	(cm)	(eq6.52)
($f_{ywd,ef}$) - Effective design strength of punching shear reinf.	344.1	(MPa)	(eq6.52)
($A_{sw,1}$) - Necessary area of a link leg	1.532	(cm^2)	(eq6.52)
($A_{sw,min}$) - Minimum area of a link leg	1.532	(cm^2)	(eq9.11)
Diameter of link leg chosen	14	(mm)	
Area of link leg chosen	1.539	(cm^2)	

Grouping of punching shear reinforcement

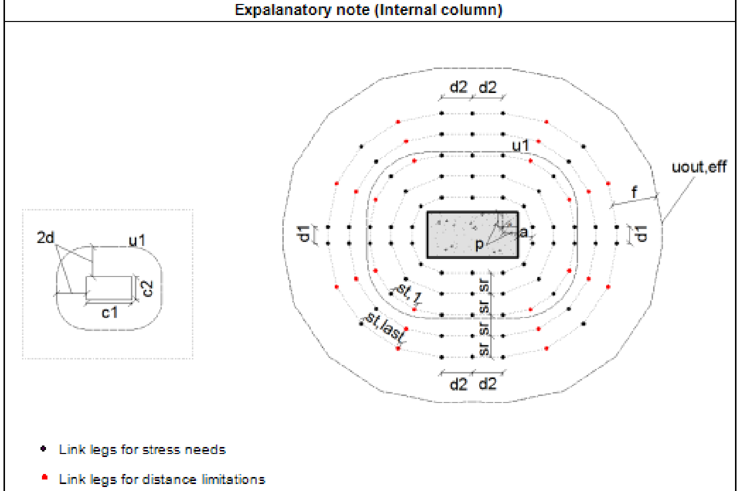
Group	Number of lines	Φ (mm)	Number of link legs on line	Height of link leg (cm)	Perimeter where the 1st link leg of the line stands	Distance of the 1st link leg from the loaded area
1	8	14	2	36.6	1	18.82
2	4	14	1	36.6	2	46.82

The second page shows the two checks as well as the detailing results.

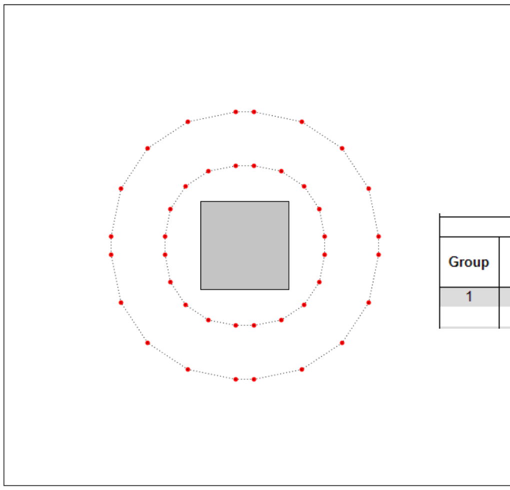
Legs Φ14 are computed (diameter of 14 mm) while in the table at the bottom of the page, the reinforcement is presented into groups of lines of reinforcement, with a certain number of link legs on line (this is a useful way of presentation in case a prefabricated system of reinforcement is chosen – stud-rail punching shear reinforcement).

In this case 8 lines with two link legs and four lines with one link leg are chosen. The two successive link legs in the eight lines are at a distance s_r (28 cm as the table shows).

Expalanatory note (Internal column)



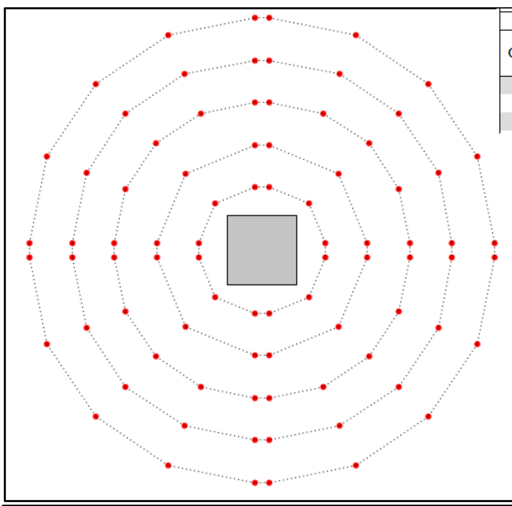
The 3rd page shows the Legend containing characteristics in accordance with the position of the Loaded Surface.



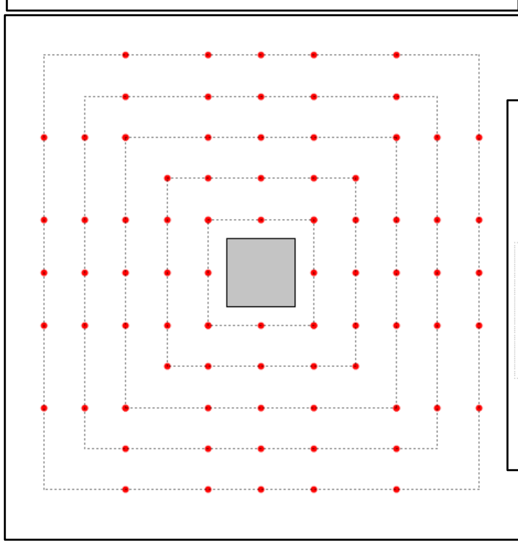
The smaller link leg may be proposed if the user chooses for more radii per quartile. For example, when five radii are chosen then the arrangement besides is proposed, and the table of reinforcement shows one group of lines containing 2 Φ8 link legs each, at a distance 28 cm.

Grouping of punching shear reinforcement						
Group	Number of lines	Φ (mm)	Number of link legs on line	Height of link leg (cm)	Perimeter where the 1st link leg of the line stands	Distance of the 1st link leg from the loaded area
1	20	8	2	36.6	1	18.82

For the same column, when the user decides to raise the load level and chooses a ΔN = 1400 kN (user value), then the radial arrangement with three radii per quartile gives Φ16 link legs (84 link legs).



Grouping of punching shear reinforcement						
Group	Number of lines	Φ (mm)	Number of link legs on line	Height of link leg (cm)	Perimeter where the 1st link leg of the line stands	Distance of the 1st link leg from the loaded area
1	12	16	5	36.6	1	18.82
2	8	16	3	36.6	3	74.82



Cruciform arrangement yields Φ16 link legs (80 link legs).

The Legend on the third page for this layout is: