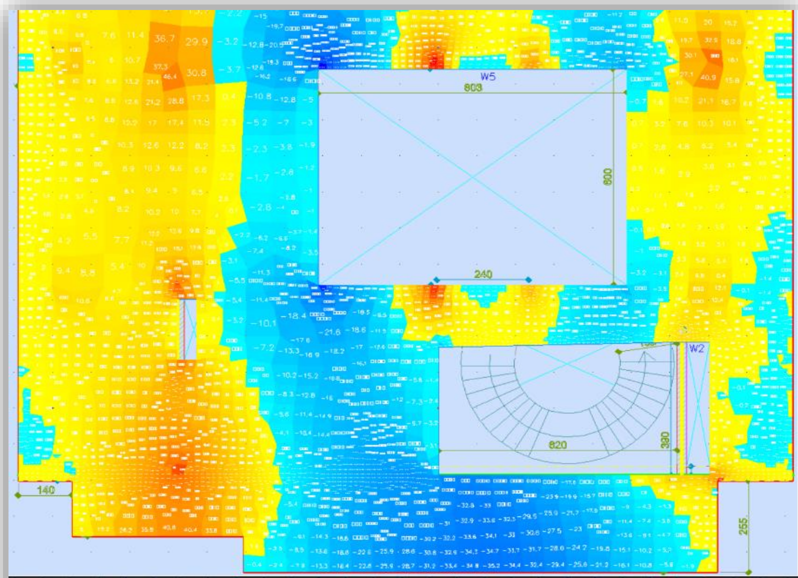
# SLABS EXPERT

## USERS MANUAL



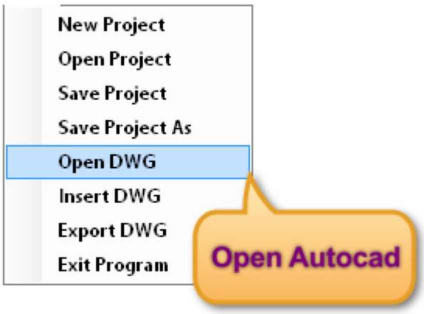
# Table Of Contents

# Chapter 1- Quick Start.

## Eager to start? Here are the steps you should follow.

1. Open the prepared AutoCAD drawing of the slab. It should be 2D, without blocks, and units should be in meters. It is recommended to just keep in the drawings what is related to the slab.
2. Save the project. A SLABS EXPERT project consists of a folder; it is not a single file. Inside this folder many files will be generated, all associated to the slab.
3. Adjust in the drafting settings the pointer snap and the orthogonal tracking. This will be of much help when modeling.
4. Create the structural layers. SLABS EXPERT associates to each layer name a structural object. This is how the software understands columns, beams, etc…These layers could be as well created in AutoCAD, before importing the drawing.
5. Use the user-friendly drafting commands that are identical to those in AutoCAD (Line, circle, copy, move, trim, extend….).

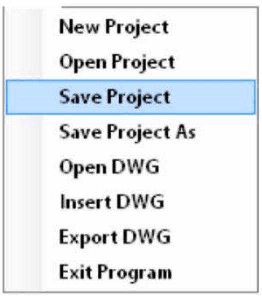
### STEP 1: FILE → Open Dwg.




Before opening the file, you should:

- Delete all layouts from the AutoCAD file.
- Delete all blocks (sections symbols, elevations symbols, attributes…)
- Units to be meters.

### STEP 2: FILE → Save Project.

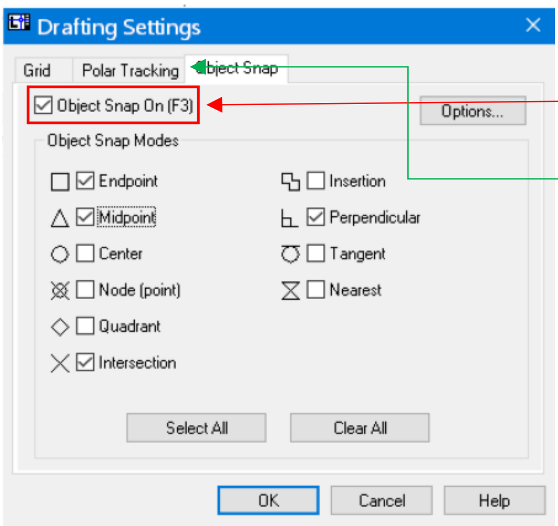



The project is a folder where the program stores its files. A new folder will be created. The name of the folder will be that of your project.

When you want to open a project, you select "Open Project" and you select the folder that you created previously.

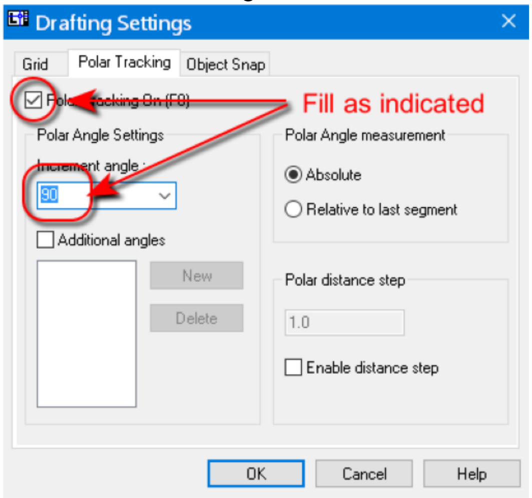
If the software is not able to save the project, it means that the AutoCAD that you opened needs to be cleaned (simplified). It is of good practice to keep in the AutoCAD just the required structural layers.

## STEP 3: TOOLS → Drafting Settings.




Check this item and the items listed below.

Go to “Polar Tracking” as shown below:

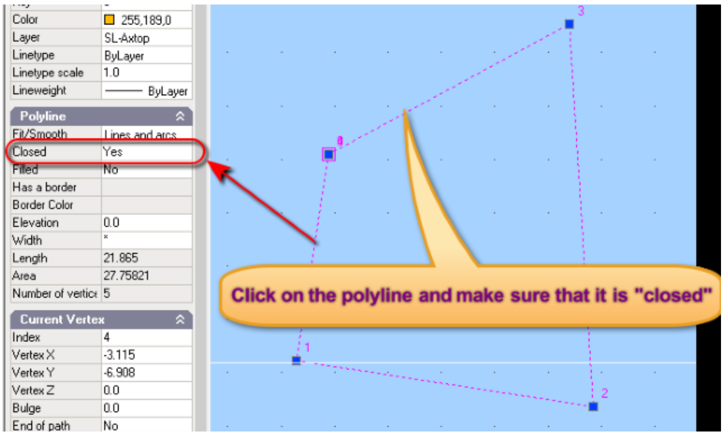



## STEP 4: Create the structural layers.

All layers to be created by the user are listed here-below.

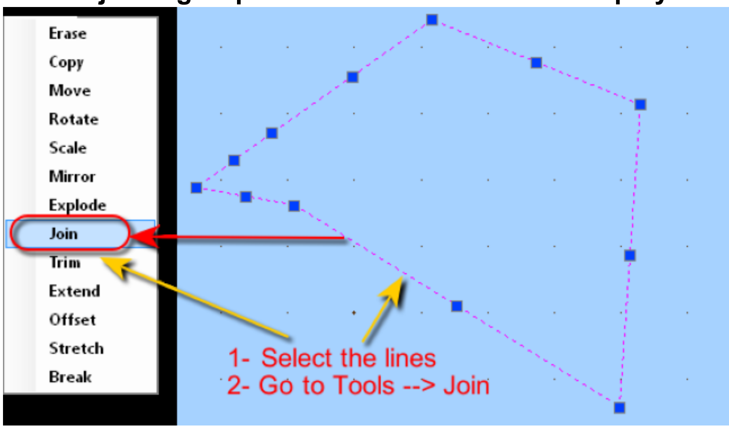
| LAYER NAME | FUNCTION | WHAT TO DRAW |
|---|---|---|
| SL-COLUMNS | Rectangular & circular columns | Rectangle (use command rectangle or circle). |
| SL-WALLS | Walls. Represented as line at the centerline of the wall. | Line |
| SL-SLABS | The main slabs | A closed polyline |
| SL-SUBSLAB | A slab inside the main slab: Could be a sunken, or a drop panel, etc… | A closed polyline |
| SL-CONTOURS | Lines used to put linear loads on them. | Line |
| SL-BEAMS | Beams. Represented as line at the centerline of the beam. | Line |
| SL-OPENINGS | An opening inside a slab. | A closed polyline |

Closed polylines in layer “subslab” represent sunken slabs or drop panels. These are supposed to be entirely comprised within the parent slab, with no contact with any edge of the parent slab.




*To be sure that the polyline is closed, click on it and check the "yes" that is written next to the "Closed" option.*

**How to join a group of lines to make one closed polyline:**



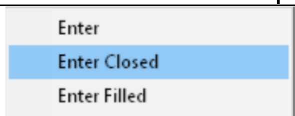
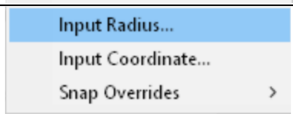
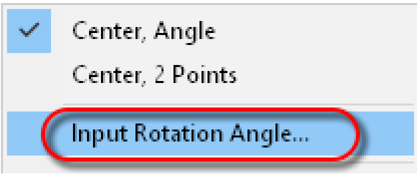
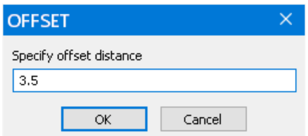

1- Draw the lines.
2- Select the lines with the mouse.
3- Go to "Tools" → "Join"

ATTENTION: All beams and walls should be lines. All slabs, openings and subslabs should be closed polylines!

Steps 1 to 4 are to be followed each time you begin a new project.

When you create your project in AutoCAD, the units should be in meters. Your slab should be close to the (0,0) reference point to avoid dealing with coordinates in kilometers!

**MOST COMMON DRAFTING COMMANDS:**

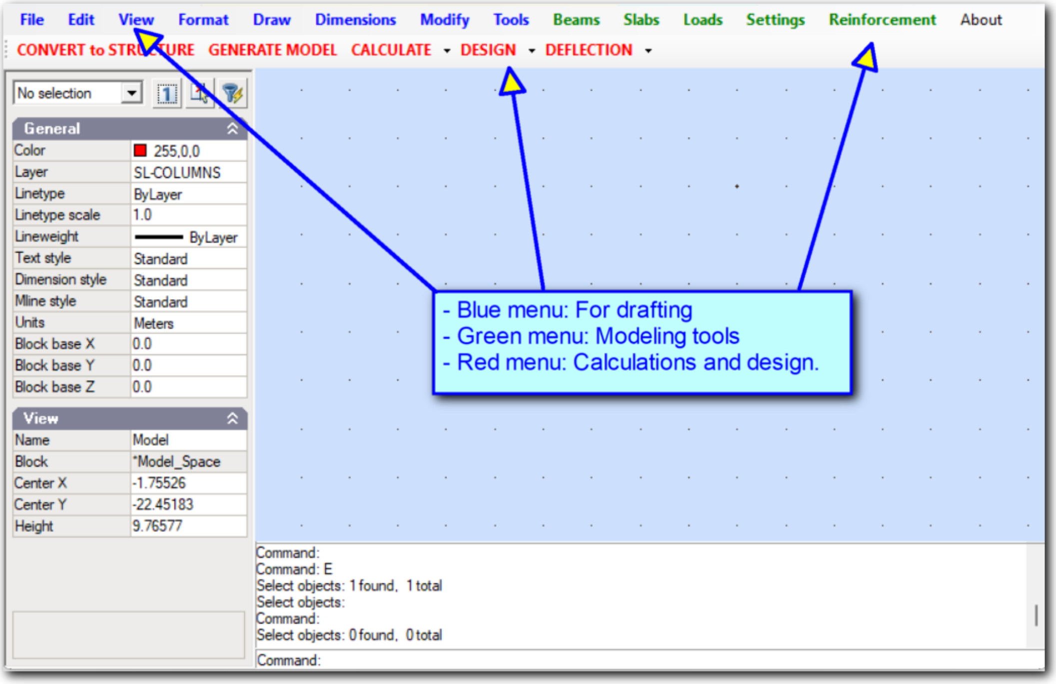
| COMMAND LINE | DESCRIPTION |
|---|---|
| “L” → Line | 1- Select the beginning point.<br>2- For the end point you can type “@3,0” |
| “PL” → Polyline |  Right-click and choose “Enter Closed”<br>To close the polyline |
| “C” → Circle |  Right-click to input the radius. <u>But before you have to select the center</u>. |
| Rectangle: with the menu “Draw”→ “2 Points” | 1- Select the beginning point<br>2- For the end point you can type “@3,2” |
| “C0” → Copy | 1- Select the beginning point<br>2- For the end point you can type “@3,2” |
| “M: → Move | 1- Select the beginning point<br>2- For the end point you can type “@3,2” |
| “RO” → Rotate | 1- Select the object<br>2- Selection the center of rotation<br>3-  Input angle (Right click) |
| “T” → Draw text | Follow the instructions in the displayed window |
| “TR” → Trim | 1- Select the cutting edge<br>2- Right-Click and select the part to cut |
| “EX” → Extend | 1- Select the boundary edge<br>2- Right-Click and select the line to be extended |
| “O” → Offset | 1- Select the line to offset<br> 2- Right-Click and select “distance”<br>3- Click on the screen the direction of the offset. |

## 1.1 Main Features

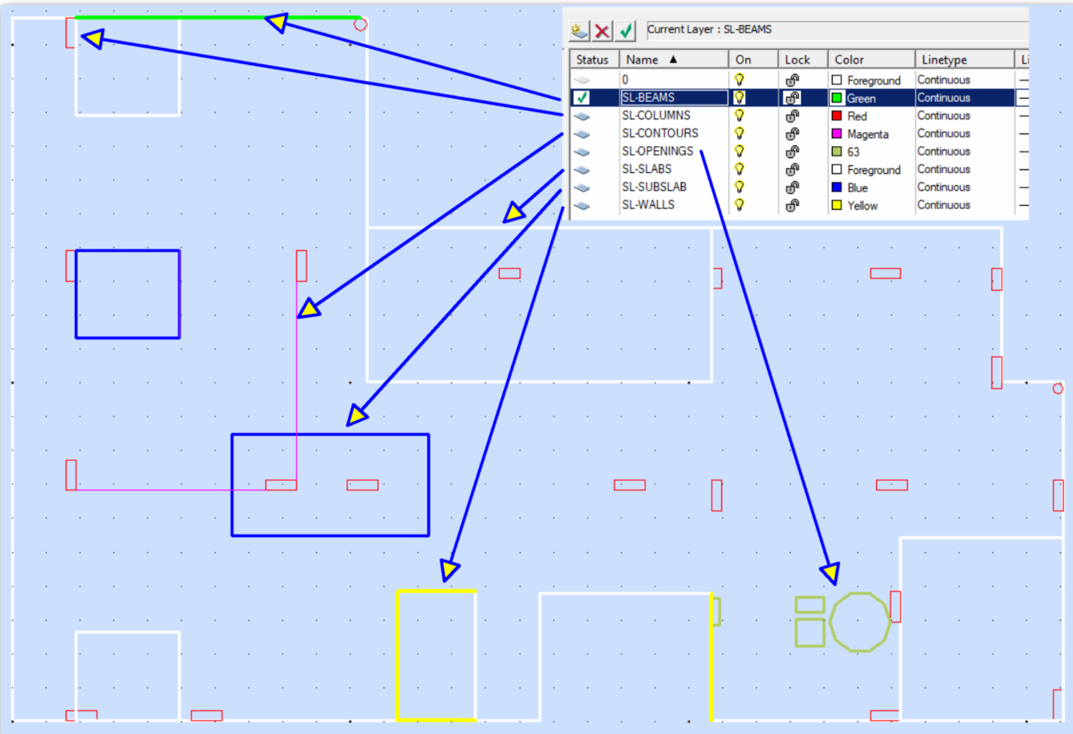
- Analysis and design of horizontal flat slabs (2D model).
- AutoCAD-like graphical interface allowing modeling with great ease.
- Direct import from AutoCAD
- State-of-the-art meshing engine that generates refined quadrangular elements, offering to the user high precision results for moments, shears, etc…
- Slab can be modeled with consideration to the stiffness of the vertical elements below and above (alike 3D models), by just introducing floor height.
- Rigid links at columns is a feature that considers physical dimensions of columns sections, leading to accurate results especially for moments and deflections. Most of the actually available software in the market model columns with their centerline only.
- It is possible to design the slab under service limit states combinations, by introducing crack width limits. The calculated reinforcement is done according to ACI 224.1R-07.
- The calculation of the long-term cracked deflection is performed with extreme accuracy. The user introduces sustained live loads, time at which partition works are completed, and the remaining part of dead loads to be done after the partitions. In addition, the user can introduce the provided reinforcement as per his drawings, at any location, which allows the program to accurately calculate the effective inertia for each finite element. The curvature of the slab is thus calculated in each principal direction, and the final deflections are obtained through an iteration process.
- Innovative design of the slab based on non-linear moment-curvature response curve that extends up to the nominal moment of the slab. The linear elastic method is traditionally used to derive the design moments from which the reinforcement is deduced. SLABS EXPERT proposes a second approach where the calculation of the moments is based on the real curvature of the slab, or in other words, on the effective stiffness. Once the user introduces the real provided reinforcement, the software calculates the moments and deduces a first value for the curvature, that once injected leads to a new distribution of moments. This iterative cycle is repeated till the moment's values do not change from one iteration to the other. This new equilibrium is hence achieved on the basis of the effective stiffness of each finite element. The program can display the corresponding ultimate moments and shears. Moreover, the ratio {ultimate moment / nominal moment} is displayed for each finite element, which corresponds to a working ratio map that helps the user verifying in a glance the adequacy of the provided reinforcement. Another benefit of this elasto-plastic calculation is the elimination of the moments high peaks at some singular meshes at the vicinity of the columns, since moment-redistribution is performed.
- The software designs mat foundations. The user can introduce the soil modulus of subgrade reaction or, for a more accurate representation, the soil elastic properties (Young modulus and Poisson ratio). In this latter case, the problem becomes that of a plate supported on infinite half-space.
- Design for punching shear according to ACI 318-25 and ACI 421.1R-20. A powerful algorithm is implemented that handles complex slab-opening configurations. Comprehensive report is generated. DXF drawings are generated, showing the layout of the punching shear areas.
- Full design of beams for flexure, shear and torsion. Long-term deflections are calculated based on effective inertia and shrinkage-creep effects. Shop drawings are generated with quantities and complete bar bending schedules.
- Generate with one click shop drawings for the full slab reinforcement. The program identifies the required reinforcement from the maps and convert them into real bars with correct length.

# Chapter 2- Introducing the software

SLABS EXPERT is a finite elements program that calculates, and designs reinforced concrete slabs and mat foundations according to ACI 318-25, metric edition.




One of the strong points of SLABS EXPERT is its graphical interface that is like that of AutoCAD, where comparable commands are implemented. The drafting menu (blue color) should be familiar to AutoCAD users. Chapter 1 of the present document gives a brief description for these commands. This figure illustrates the 7 layers the user can use to create the structural elements.



*"Menu **Format** → Create Default layers"* automatically creates all seven layers.

It is possible to draw the geometry of the slab either in AutoCAD or SLABS EXPERT. The command "File → Open DWG" imports the drawing from AutoCAD to SLABS EXPERT including layers. Once the drawing is opened, it is recommended to directly save the project. A SLABS EXPERT project is a **folder**, not one single file. When you save or open a project, your are in fact selecting a folder.

**Menus in blue color are used to draw the geometry of the slab (lines and polylines).**

**Menus in green color are used to assign structural properties to the elements:**

- Create and assign beams sections.
- Create and assign slabs sections.
- Create and assign loads.
- Modify the design parameters from the "Settings" menu.
- Assign the provided reinforcement to the slab for deflection calculations purpose.

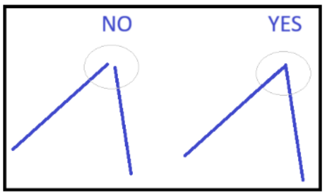
Columns are directly recognized by the software since they are drawn as rectangles or circles.

**Menus in red color allow the user to perform calculations and design:**

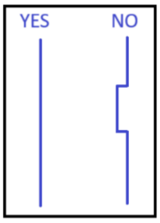
- The "Convert To Structure" button is of major importance. It converts the geometrical model into a structural model. Lines and polylines that belong to their corresponding layers are converted to beams, slabs, openings… The software performs some basic error checks to verify that the slabs are closed polylines, that slabs do not intersect, that openings are entirely located within slabs, etc.… This function should be used during the creation or modification process of a model.
- The "Generate Model" menu divides slabs into quadrangular finite elements. If the geometry of the slab is correctly defined according to the rules detailed hereafter, the meshing operation should succeed. In case of failure, meshes are not generated and the user should review the geometry.
- Menu "Calculate" is used to run calculations (combinations are automatically created) and to display results.
- Menu "Design" performs the reinforced concrete design.
  - Flexural reinforcement for the slab is calculated.
  - Design for punching shear in slabs
  - Design of beams subject to bending, shear and torsion
- Menu "Deflection" calculates the cracked long-term deflections. The software computes by itself the allowable limits (After five years, Net deflection for residential floors and that for parking).

# Chapter 3- Drawing the geometry of the slab

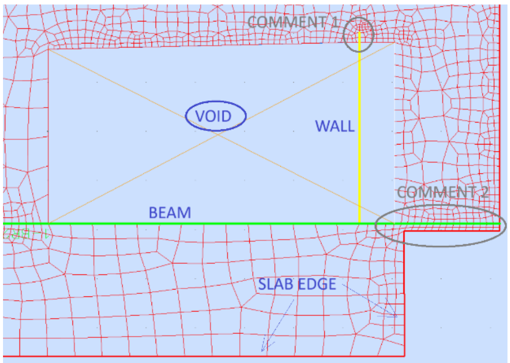
Seeming trivial at first impression, drawing the geometry of the slab is of utmost importance because of its direct consequences on the meshing and calculation steps. If not realized with great precision and care, the geometrical errors might prevent meshing from being generated.




Draw correct intersections. When the ends of two perpendicular lines are intended to intersect, they should. If not, then do not keep a gap less than 10cm. Distance of less than 10cm between two nodes will lead to dense meshing and consequent large number of meshes. Sometimes the software won't be able to generate the meshing plan.

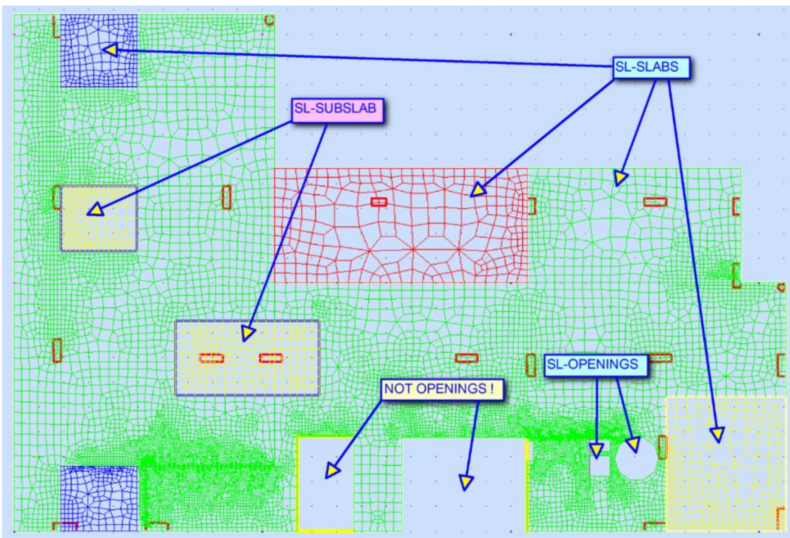



Simplify edges of slabs. Do not follow the architecture as it is. If the edge of a slab contains indentations of extrusions of amplitude less than 10cm, then do not consider them and adopt a straight edge instead. The dimensions of the irregularity in both directions should be more than 10cm, otherwise unnecessary meshing complications are to be expected.




This slab contains one big void, a wall at its extremity supporting the slab, and one beam. The first comment concerns the extension distance of the wall beyond the side of the void, as depicted in the figure. If the distance between the end of the wall and the edge of the void (or slab) is less than 10cm, a high concentration of meshes is expected. The second comment concerns the distance between the beam and the slab edge. In general, when two lines are parallel, the distance between them should not be less than 10cm. Otherwise, it is advised to superpose them.

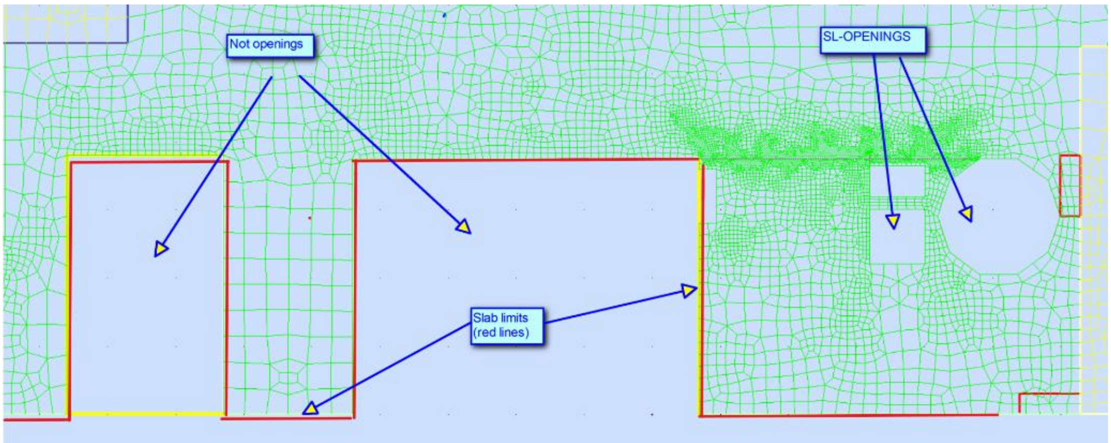
If the above-mentioned gaps are very small (less than 1cm), which means consequent to drafting error, the program might not be able to perform the meshing generation task!



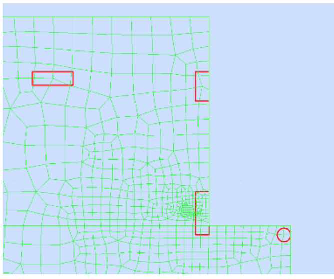

*Slabs, subslabs and openings represented in their corresponding layers*

The figure in the previous page illustrates the following:

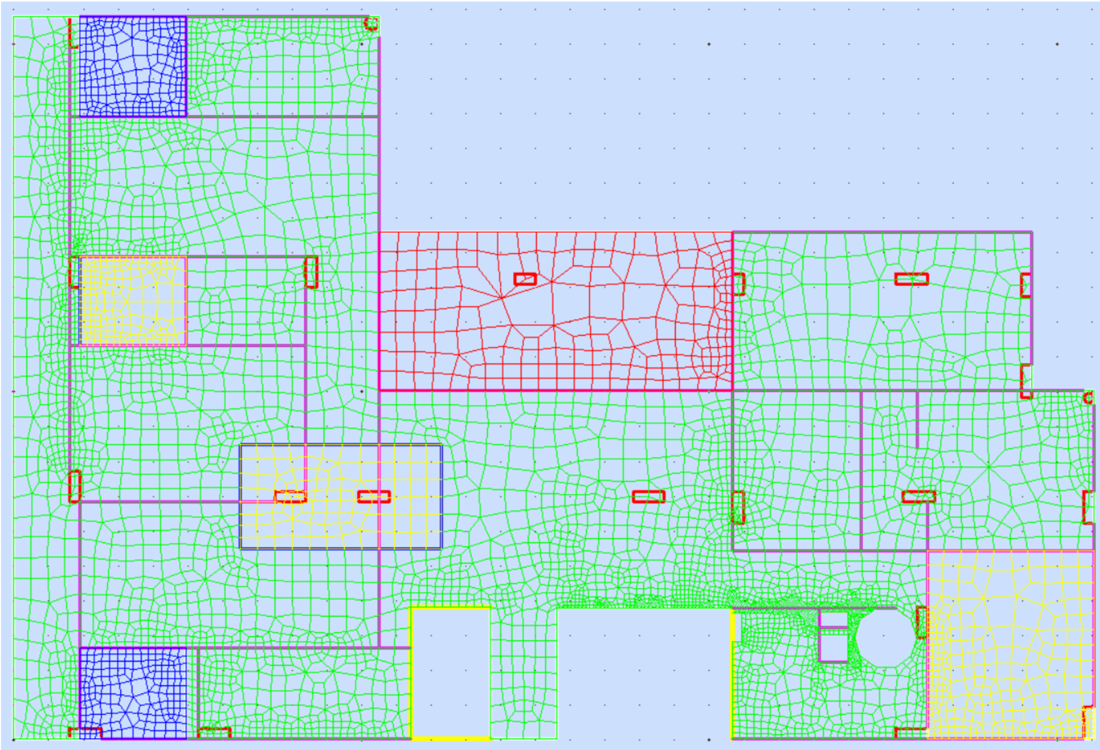
- Slabs are closed polylines located in layers “sl-slabs”. If we have a look at the slab in red color (for instance), it cannot be considered as a subslab because it is not entirely enclosed within the main green slab. Only island slabs that are totally comprised within the main slab must be drawn in the layer “sl-subslab”. It is important to understand the difference between a subslab and a slab. Sunken slabs and drop panels that are entirely surrounded by meshes that belong to the parent slab are placed in “sl-subslab” layer.
- Openings in slabs often occur at the edges. In this case, it is not recommended to keep a small strip between the edge of the slab and that of the void. It is more efficient to simply adjust the limits of the slab to match those of the edge opening, as shown in the figure here-below. The three openings at the right are totally enclosed within the slab, and they should belong to layer “sl-openings”. Note that openings, like slabs, are modeled with closed polylines. A circular opening should be discretized into a series of segments.




*How to model edge and central openings.*

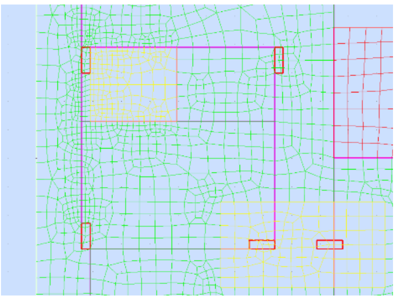


Columns located at the edge of the slab should have their edges superposed to that of the slab. The reason is that in case the option “Consider rigid links at columns” is selected, the interior of the columns will be meshed like a small plate, with big thickness affected to these meshes. This is how the interior of the columns is considered in the software. If a small gap is kept between the side of the column and the edge of the slab, the software might not be able to mesh it. In case the rigid link option is not used, then the only constraint concerning edge columns is limited to the distance between the center of the column and the edge of the slab. Here again the 10cm minimal distance rule applies. It is not recommended to keep the center of the column within a few centimeters of the edge of the slab. The circular column, as shown in the figure, is tangent to the edge of the slab. The rectangular column just to its left is located at the corner. Note that the distance between the corner of the slab and the corner of the column is intentionally kept larger than 10cm, to respect the minimal distance rule between two nodes.

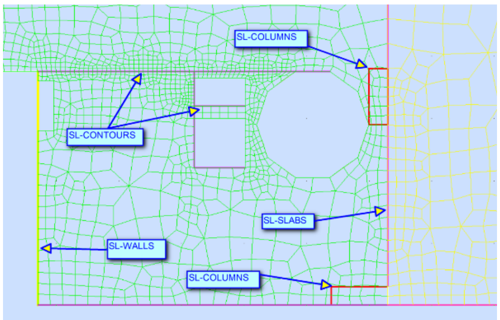


*Internal partitions (lines in magenta), located in layer "sl-contours"*

This figure shows the internal partitions of the floor, that are modeled as lines, where linear loads will be applied. The main purpose of the layer "sl-contours" is to allow linear load application on lines that are present in this layer. These lines are attached to the meshes and therefore they abide to the same rules mentioned before.



This close view shows that lines in layer "sl-contours" are preferably drawn from the corners of the columns, where a node is already present. If not, then they are far enough from this corner to respect the 10cm minimal distance rule. When they are close to the sides of a subslab (in our case the sunken slab of a toilet is colored in yellow, and we are modeling the partition walls), the block wall and the slab edge are exactly superposed to avoid ending with adjacent lines parallel to each other's, where eventual meshing problems could arise.




The structural wall at the left (in yellow color) is 100% superposed to the edge of the slab. Furthermore, in the "Y" direction, its length is adjusted to match that of the slab edge, avoiding extending inside the slab, unless it has to, but in this case, it should be for more than 10cm depth. The partitions are adjusted exactly at the edges of the central openings, and also their lengths match those of the opening's sides. Finally, at the right, we note that the edges of the green and yellow slabs are aligned with the top and bottom columns sides. The emphasis on the accuracy and "cleanness" of the geometrical layout of the slab is intended to eliminate the biggest part of the meshing problems that user might encounter. The geometry of the slab can be done in AutoCAD and imported to SLABS EXPERT.

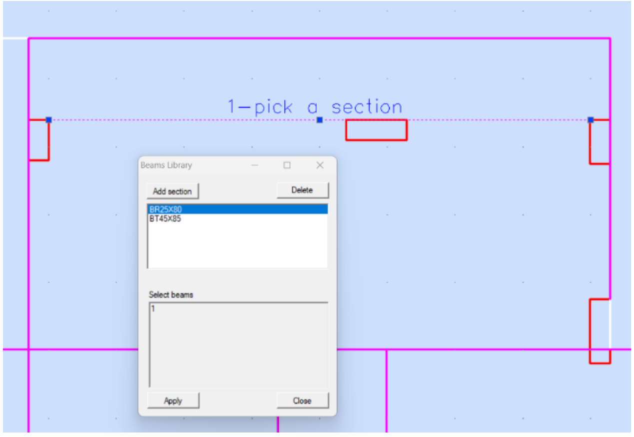
# Chapter 4- Assigning the structural properties

## 4.1 Modeling structural elements

The green menus are used to assign to the lines and polylines their elastic characteristics.

When the ***"Convert to structure"*** menu (in red color, at the top left of the screen) is clicked, the software verifies if all lines and closed polylines that belong to their corresponding layers are affected with structural properties. A label will appear for each entity, describing its structural section, if it exists. Otherwise, the label will show a sentence inviting the user to assign a beam or slab section.

### 1. "Beams" Menus

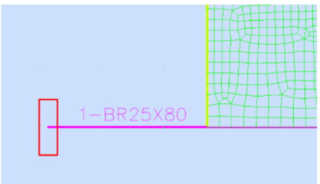



A label will appear for each line that belongs to layer "sl-beams". Lines without properties will just have a numerical number, with a message "pick a section" next to it. "Beams" menu opens the sections manager where the user can create the desired sections. Rectangular and "T" sections are available. The created sections appear as a list, as shown in this figure... The user must select the desired lines by making a window with the mouse in the graphical screen, so their number will appear at the bottom half of the "beams library" window, as shown in this figure. Even if among the selected lines there are some that belong to other layers, it is not a problem. The software will filter only the lines in layer "sl-beams". Before clicking on the "Apply" button, the user should select from the list the section to assign; it will be highlighted in blue (refer to the figure).

***Tips for drawing beams***

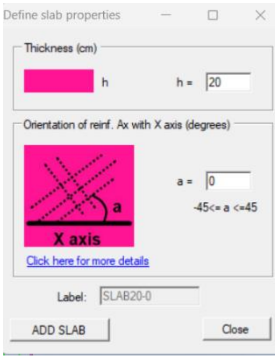
Commonly, a beam is modeled as a bar element (line) which ends should match columns or walls centerline. In the figure here-above, the beam is intended to connect three columns. Two cases are considered:

a) If the rigid links option (refer to the "General settings", inside the "Settings" menu) is not retained, then the beam should be connected to the column's centerlines.
b) When the rigid link option is used, the column becomes like a panel, where the meshes located inside its perimeter are of large thickness. In this case, the beam does not have to reach the centerline of the column. Reaching any point of the column's side is adequate, although it is possible to draw the beam like described in case a). When using the rigid links option, care to be considered when connecting the beam to the faces of the column: Try either directly connecting to the corners or not too close to them, to avoid meshes concentration. In the figure here-above, the beam is intentionally connecting the corners, to simplify the meshing process.




This figure shows the case where a column is located outside of the slab perimeter (or inside the void of an opening). The right end of the beam is connected to one mesh, which is Ok. However, the left end, although connected to the column, is in fact considered as floating. The reason is that the column is not attached to a mesh, nor the left end of the beam. The beam will be assumed as a cantilever, like if there is no column. Always make sure that the beams ends and the columns are attached to the meshes.

## 2. “Slabs” Menus




When defining a new slab, two types of entries are requested. The first one is the thickness. The second one is the angle between the reinforcement “Ax” and the “X” axis which is always horizontal. Reinforcement of the slab consists of “Ax” and “Ay” orthogonal mesh, placed at the top and bottom faces. “Ax” is supposedly the reinforcement parallel to the “X” direction, but in many cases the configuration of the slab leads to the consideration of an inclined reinforcement mesh. This is why the user can specify the angle “a” that “Ax” makes with the “X” axis. When the reinforcement maps are displayed, one should keep in mind that “Ax” map shows the reinforcement that is oriented according to the introduced “a” angle, in degrees. “Ay” is always perpendicular to “Ax”.

After defining the different slabs, the user should assign them to the closed polylines that belong to layers “sl-slabs” and “sl-subslab”. Selection is graphically performed, with the mouse. Even if other objects are selected in the process, only contours belonging to layers “sl-slabs” and “sl-subslab” are filtered.

## 3. Columns

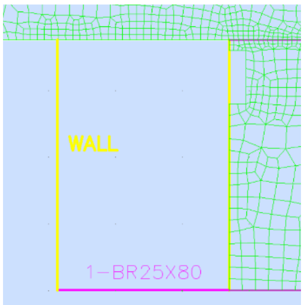
Definition of columns is straight-forward. Rectangular and circular columns are automatically recognized by the software. Use the command “Rectangle” for rectangular columns or draw a closed polyline. The risk of this last option is that the sides of the closed polygon should be 100% perpendicular to each other’s, to make a perfect rectangle. The software identifies the rectangles in layer “sl-columns” and labels them with their dimensions. Columns work as pinned supports (no moment interaction with the slab) if the user keeps the story height zero in the “General settings” menu. If a value is given for the story below or above (or both), then the flexural stiffness of the column is incorporated to the model.

It is also possible to activate the rigid links option (located in the “General settings” menu) for a more accurate representation of the columns, when they have large dimensions.

❖ The centroid of a column should always be attached to a mesh node. The figure of the previous page illustrates a case where the presence of the column will be ignored.

## 4. Walls

Walls are **line** elements in layer “sl-walls”. They do not have structural properties. Thickness of walls is not to be introduced because the software always assumes a hinged connection between the slab and the wall (no moment transmission). The purpose of the wall is to provide a vertical support for the slab.




This figure shows the case of a beam supported on a wall. The upper end of the wall and the right end of the beam are attached to meshes nodes. Since the wall is located outside the meshed plan, it will not be recognized as a support, except for its upper end. This wall is interpreted as a single nodal support only. The beam is considered a cantilever. Only nodes attached to the wall are considered as supports. If the bottom node of the wall is connected to a mesh node, then a support will be generated, and the beam is supported at both ends.

## 4.2 Introducing loads

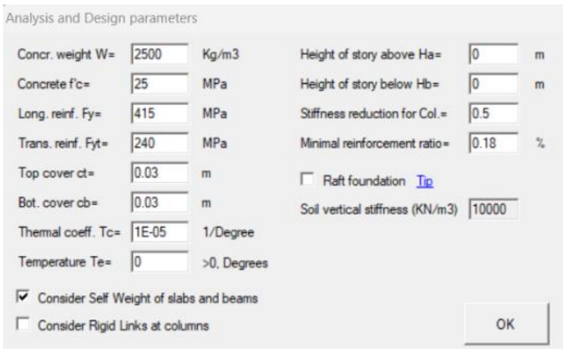
**Loads menu**

- Surface loads can be applied to the slabs (vertical loads only, in KPa).
- Linear loads are applied to beams (layer "sl-beams") and lines (layer "sl-contours"). Units are KN/m for the vertical forces and KN.m/m for the moments. These latter are introduced about the global axis. Mx is the moment about the "X" horizontal axis.
- It is not possible to introduce concentrated forces. Moments and shears under singular loads tend to undefined values. Loads induced by implanted columns can be modeled as linear loads, by drawing a line along the length of the column. Substituting nodal load by a linear load, even over a small distance, is more accurate.
- The option "loads on columns" is used in the case of mat foundations only. When this option is selected in the "General settings" menu, the software considers the slab resting on elastic supports. The purpose of the introduced columns becomes solely to introduce concentrated forces at their centerlines. Using the rigid link option allows to distribute each force over the area of the column.

The software considers gravity loads only: Dead and Live. It is not possible to introduce lateral loads on the slab. Combinations are hence automatically generated. Thermal loads can be considered as well, as shown hereafter.

# 4.3 The Settings menu

## 4.3.1 General settings




Most of the required parameters are self-explanatory. Transversal reinforcement consists of vertical links to resist shear and punching shear forces. The design for one-way and two-way shear is not yet implemented in the software.

Thermal loads are considered by introducing the temperature variation Tc, to be >0.

**How thermal loads are considered.**

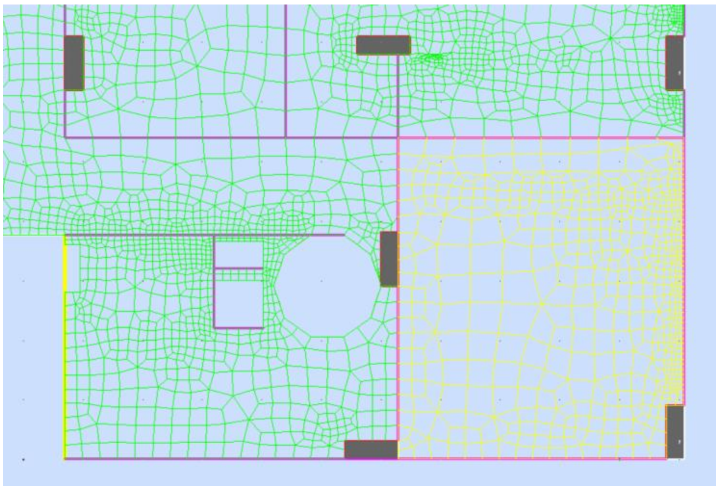
Self-straining forces are always challenging when it comes to their correct consideration. The main reason is that in a linear elastic analysis stresses and strains are proportional. Due to energy conservation theorems, an imposed strain to a structure produces a work that should be entirely converted to internal stresses. When the stiffness of the vertical members is important, high stress concentration is expected at some key locations. Usually, the consequent reinforcement that is required to equilibrate these stresses is abnormally high. The elastic approach is not adequate to design self-straining forces. The main reason is that in real case, apparition of cracks in concrete releases the biggest part of the strain work, whereas common elastic analysis does not account for these cracks that are decreasing the applied imposed deformation.

SLABS EXPERT calculates the required tension reinforcement as follows:

- **If crack control ("Cracks control" menu) are not of concern**, then the ultimate state governs the design. Vertical cracks are supposed to develop all across the thickness of the slab. Tension-stiffening between two adjacent cracks is neglected. Between the two lips of the crack, the reinforcement is resisting the forces consequent to thermal strains. Stress in reinforcement becomes: $\sigma$=Es*$\varepsilon$, where Es=200 000 MPa and $\varepsilon$=Tc*Te (same notation as in the window). $\sigma$=F/(As+At), where As is the area of the top and bottom reinforcement in the considered direction, and At that to be added to resist thermal stresses. The force F should be resisted by additional reinforcement since the existing one is intended for flexural strength. The required reinforcement to resist thermal loads is At=F/fy, where fy=420 Mpa. After few manipulations, we obtain the following expression for the required additional reinforcement At:
  - $A_t = \frac{A_s}{\frac{\varepsilon_y}{\varepsilon} - 1}$ where $\varepsilon_y$=0.002 (yield strain) and $\varepsilon$=Tc*Te. It is important to note that the strength reduction factor $\Phi$=0.9 divides At. The additional reinforcement consequent to thermal gradient is At/0.9. The software will add it to that required for flexure.
- **If the user imposed a crack width to be respected**, then the design is likely to be governed by the service limit state. The working stress of reinforcement is limited to 0.6*fy. We suppose that cracks are not fully developed, and that the stiffness of concrete is resisting the imposed strains. $\sigma$=Ec*$\varepsilon$, where Ec is the elastic modulus of concrete and $\varepsilon$=Tc*Te. $\sigma$=F/Ac, where Ac is the cross section of the concrete slab, per meter width. At=F/(0.6*fy). To be noted that in SLS combinations, ASCE 7-16 factors self-straining loads with 0.75.
  - At = 0.75*Ac*$\varepsilon$*Ec / (0.6*fy). The software adds the reinforcement At to that required for flexure.

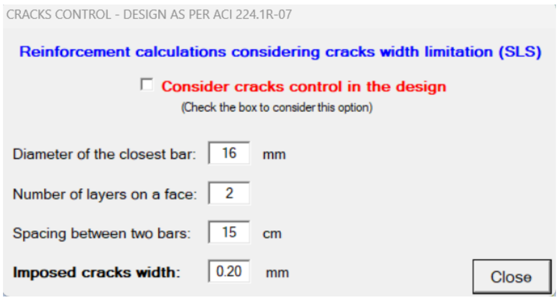
To access the crack control window: click on the "Settings" menu.

- Clear cover is considered at SLS (when crack control is activated) and also at ULS to determine the effective depth of the slab.
- When the height of the story below is more than zero, the bending stiffness of the columns intervenes. The axial stiffness of the columns and walls will be as well incorporated, which will lead to a more accurate distribution of the vertical reactions. The flexural stiffness of the walls is not considered (pinned supports).
- When the height of the story above is more than zero, the bending stiffness of the columns above the floor will be considered. If already the floor below exists, then the flexural stiffness of the columns is therefore increased. The presence of the columns decreases both negative and positive moments of the slab, since additional stiffness is injected to the model. It is recommended to consider a part of this stiffness by introducing a reduction stiffness factor <1.
- The minimal reinforcement ratio ensures that the software will not display null reinforcement areas. For each direction, and for top and bottom reinforcement, this minimal ratio will be respected. Another role for this ratio is its consideration in the calculation of the time-dependent deflections. The software supposes that inside each finite element the minimal reinforcement corresponds to the introduced ratio.
- When the soil subgrade modulus (Winkler coefficient) is introduced and the adjacent checkbox checked, the slab is considered on grade (mat foundation). In this case the introduced columns are supposed to be located above the slab, and it is possible to introduce the corresponding concentrated forces at their nodes. The introduction of walls is not of interest. If the user wishes to consider the flexural stiffness of the walls, it is recommended to model them as beams with height equal to one or two stories.
- The rigid links at columns option is very useful when the length of the columns becomes significantly big with regards to the spans. For instance, if the length of the column exceeds tenth the span, it is advised to consider the rigid links option. Practically, this option assigns to the interior area of the column meshes of large thickness, colored in grey. The number of meshes becomes larger and calculation time a bit longer, but it deserves consideration for a more realistic model. As a direct consequence, the deflection becomes smaller.



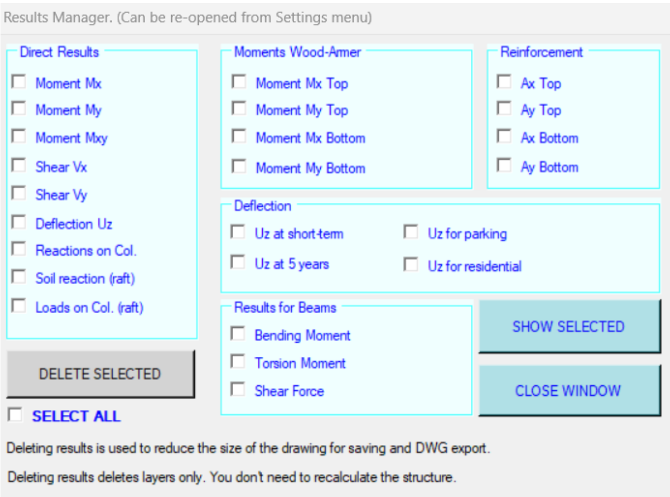
*Rigid links at columns:*

## 4.3.2 Cracks control

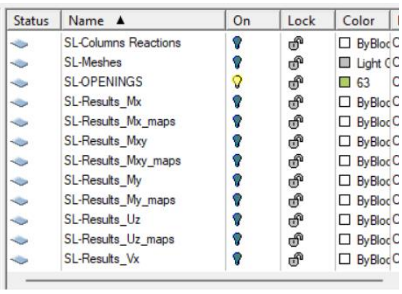


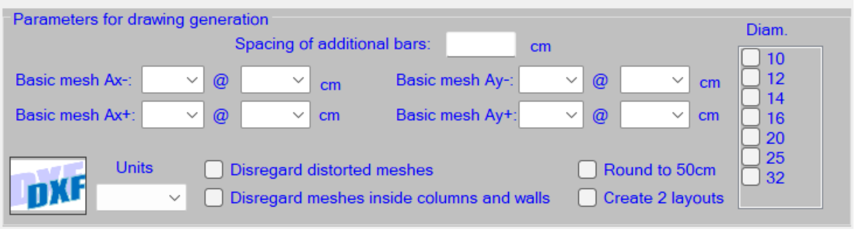

When the slab is exposed to environmental conditions that require limitations of crack width, the design should be performed according to the Service Limits State. The imposed cracks width should be respected by controlling the allowable stress of the reinforcement. The adopted procedure is that of ACI 224.1R-07. The software will calculate the flexural moments under D+L combination and will provide the consequent reinforcement with respect to the maximal value of cracks width.

## 4.3.3 Results manager




This window is of prime importance. It is either automatically opened after calculations are performed or it can be manually called from the “Settings” menu. It shows all the results grouped inside one form. If reinforced concrete design is performed, the “Reinforcement” results are available. Same for the deflection and other types of results. Once the “Show selected” button is clicked, the result is loaded from the memory to the layers. So, the purpose of this window is to create the corresponding layers and visualize them on the screen as colored maps, with values displayed. If no results are selected and the “Show selected” button is clicked, the screen is cleared. The software automatically turns off the corresponding layer. One can as well access the layers from the “Format” menu and sight the correspondence between the “on” layers and the checked result in the Results manager. For instance, if the “Moment Mx” is selected in the window above, layers “SL-Results_Mx” and “SL-Results_Mx_maps” are automatically turned on in the layers manager. Each result in the Results manager is associated with two layers in the Layers manager. One for the maps and the other for the labels. The button “Delete selected” does not delete the results. There is no need to re-run the calculations. It just deletes the associated layers in the Layers manager to reduce the size of the drawing. The results are still available, but they are deleted from the drawing. The purpose is ease the export of the drawing (including all layers) to AutoCAD. If all layers are present and filled with their maps and values, the size of the drawing becomes very big, and it cannot be exported to AutoCAD. The “Delete selected” button empties and deletes the selected results (in fact it does so for their layers) and decreases the size of the drawing. It is possible now to export the drawing (menu “File” and then “Export Dwg”) while keeping only the necessary results. Several drawings can be exported like that if the user wishes to convert to dwg all the results.



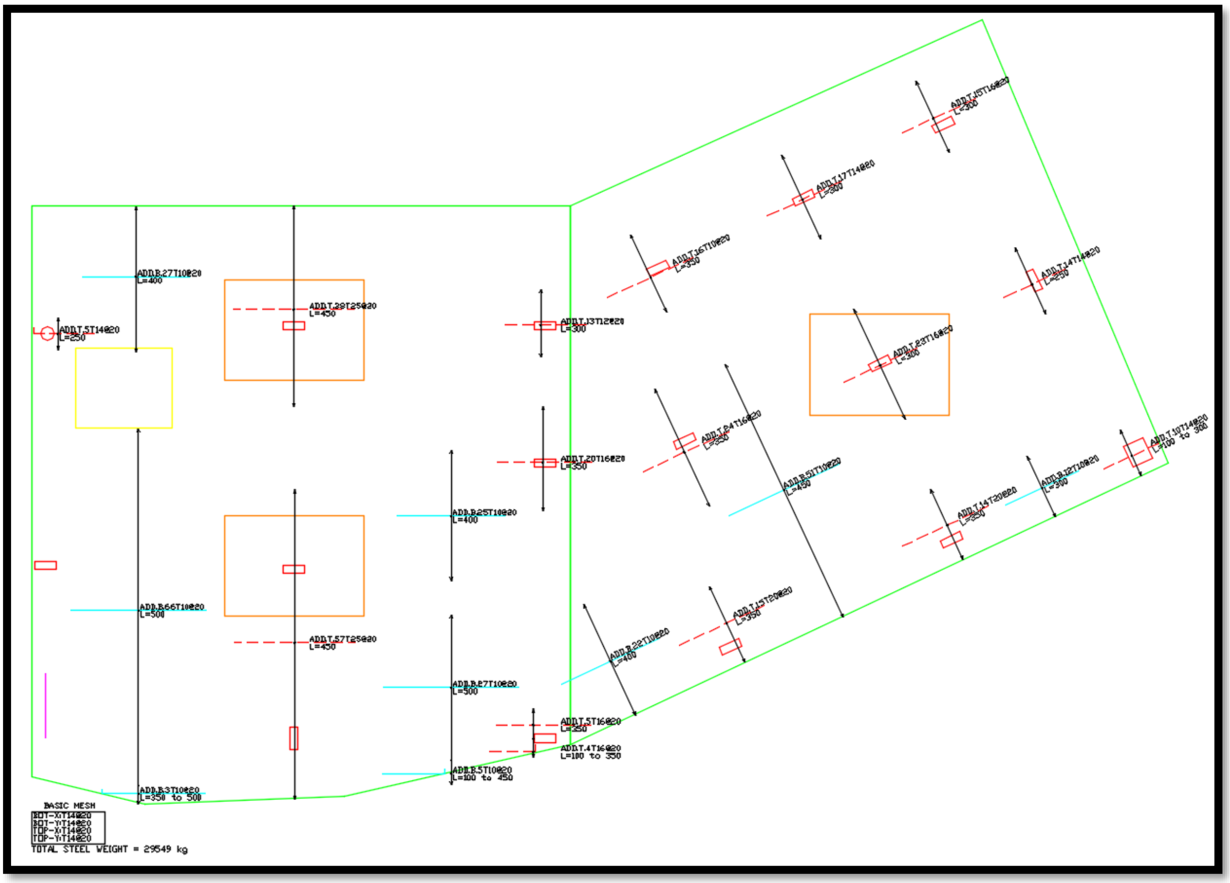



The program generates detailed drawings for longitudinal reinforcement, with consideration to slab edges and openings. The user first should specify the basic mesh.

It is recommended for the basic mesh to at least cover the minimal code requirement (As,min) to avoid obtaining additional reinforcement over the whole area of the slab. The spacing of the additional reinforcement should be as well introduced. Incoherent meshes -also called distorted- which have an angle on a vertex less than 15º or a sides ratio>3 can be disregarded due to their lack of reliability. Furthermore, meshes located within the perimeter of columns and walls can be as well ignored.

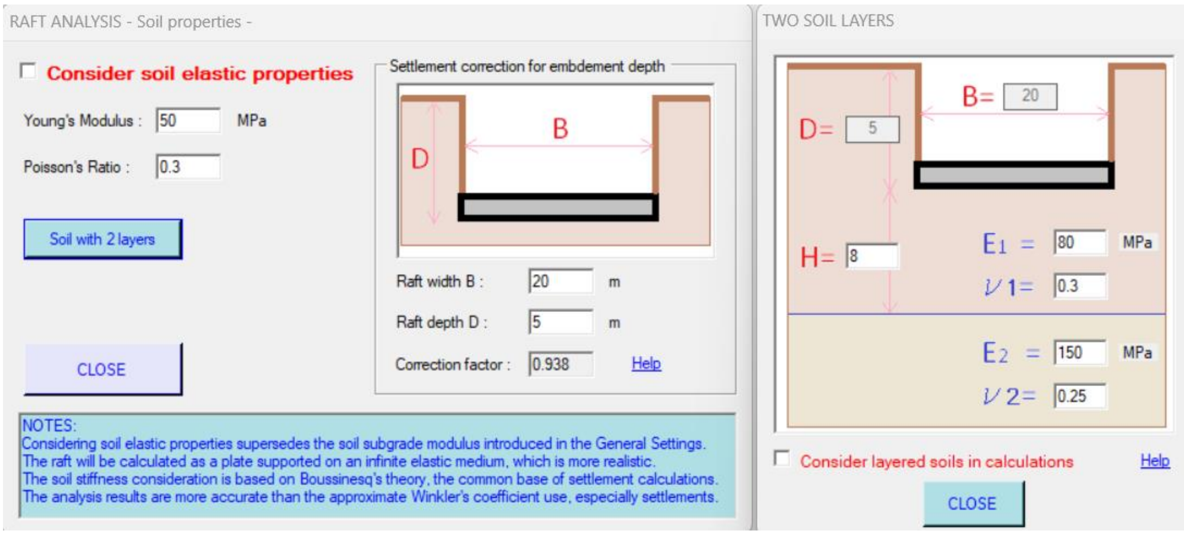
- Bars length can be rounded up to 50cm
- Reinforcement in direction X and direction Y can be grouped in two separate layouts.

A powerful algorithm translates finite element maps into practical reinforcement layouts using several weighting criteria. The result is a human-like drawing generation where bars are hooked where they should, correctly placed around openings, and grouped by length at inclined edges.




*Sample drawing generated by the program*

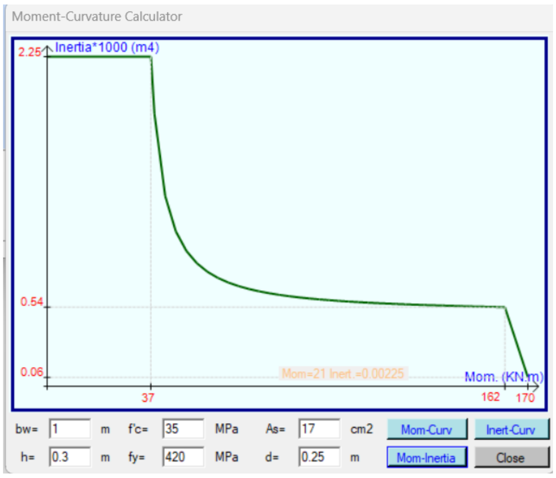
### 4.3.4 Advanced soil properties




If the "Raft foundation" checkbox in the "Settings" menu is selected, then the software assigns to the nodes springs which are uncoupled or coupled.

- The Winkler model (modulus of subgrade reaction "K") consists of individual springs assigned to the nodes of the slab, which stiffness is "K". These springs are uncoupled in the sense that they all have the same stiffness and there is no interaction between the stiffness at node "i" and that at the adjacent node "i+1". It is a simplified representation of the soil response.
- If the user considers soil elastic properties (by selecting the checkbox here-above) the software uses Boussinesq theory to describe the soil response. If a concentrated force is applied on a half-space (disregarding the stiffness of the raft), the spring that is located at the application point of the force displaces, but not alone. All the surrounding nodes -even if unloaded- will move as well according to the equations of elasticity. The total displacement of a node is equal to the sum of: the part induced by the load right above, and the part induced by the loaded nodes that are around. In the Winkler model, a node displaces only if it is loaded. There is no interaction between the movement of one node and another, unless the stiffness of the raft is considered.
- The elastic properties of the soil are required. The use of elastic properties implies that the soil is supposed to be granular, and that consolidation settlement is not of concern. Otherwise, the calculated settlement will not be exact. The calculation of settlements requires the introduction of the excavation depth and width to derive the overburden pressure. A reduction factor for settlement is calculated according to Meyerhof, 1965. It is possible to consider a layered soil (two layers). The equivalent stiffness of the springs is calculated according to Bowles, 1987.

### 4.3.5 Moment-curvature calculator




This window proposes a simple tool that calculates the evolution of the moments curve in function of the curvature, from the uncracked section till its nominal strength. Furthermore, it provides inertia-curvature and moment-inertia curves, which are very useful when assessing the effective stiffness of a beam in function of the applied moment.

The software can perform an elasto-plastic analysis based on the moment-curvature approach. This calculator tool based on the same engine.

## 4.4 The Reinforcement menu

This menu allows the user to define the provided reinforcement as per his execution drawings. The purpose is to perform an accurate long-term deflection analysis and non-linear analysis at ULS.

### 4.4.1 Basic mesh for slabs

The user can introduce the provided basic mesh for each slab. It is introduced as reinforcement areas, for directions X,Y top and bottom.

If not introduced, the software assumes that the minimal reinforcement ratio (refer to section 4.3.1) applies. This reinforcement is used when calculating the long-term deflection. It intervenes in the determination of the effective inertia, where the provided reinforcement is a major variable. The user can introduce an inclination angle for the mesh in case the provided reinforcement is not aligned with eh global X,Y reference.

### 4.4.2 Additional reinforcement for slabs

One of the major limitations encountered when calculating deflections with commercial software is the consideration of the provided reinforcement, not the calculated one. An accurate determination of the deflection should be based on the provided reinforcement at different locations within the slab. For instance, at some columns the user can specify additional top reinforcement to the already defined basic mesh. This can be done as well at mid-spans, for the bottom reinforcement.

The user can thus emulate his execution drawings on the numerical model.

SLABS EXPERT provides powerful means that help in the allocation of the provided reinforcement to the finite elements meshes. The user can create layers named as follows (sample):

SL-AxBot_25-AxTop_9-AyBot_0-AyTop_16.5-Angle_0

Each of these layers contains variables to be modified by the user, indicated in red color as shown. The text above indicates that the additional bottom reinforcement in direction X is 25cm$^2$/m, the additional top reinforcement in direction X is 9cm$^2$/m, no additional reinforcement for bottom reinforcement direction Y, and 16.5cm$^2$/m is the additional top reinforcement in direction Y. The angle of this additional mesh toward the X axis is zero degree. Only numbers in red color can be modified. This reinforcement will be added to the existing basic mesh that was defined in the previous section.

The total reinforcement will be used to estimate the effective inertia of each finite element in both principal directions. The user can create as many layers as required.

Once the layer is created, the user can draw rectangles or closed polygons at the concerned areas, without being exactly restrained by the contour of the slab. The created polygons can extend outside the limits of the slab; edges are not obliged to superpose.

# Chapter 5- Calculations and Design

## 5.1 Converting lines to structural members

The “Convert to structure” menu (in red color) converts the drafting model into a structural model.

- Lines in layer “sl-beams” are labeled and require to be assigned a section.
- Lines in layer “sl-walls” are converted to supports.
- Lines in layer “sl-contours” are labeled and ready to receive linear loads.
- Rectangles in layer “sl-columns” are labeled and transformed to columns.
- Closed polylines in layer “sl-slabs” and “sl-subslab” are labeled and require to be assigned a section.
- Closed polylines in layer “sl-openings” are converted into voids.

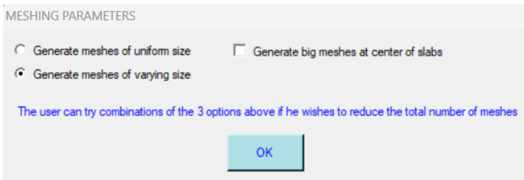
Furthermore, several integrity checks are performed, like isolated columns, superposed contours…

It is important to note that it is up to the user to correctly realize a drafting model without ambiguity, since the capacity of the software to detect some type of errors is limited.

It is recommended to frequently use this function during the modeling process.

## 5.2 Meshing the slabs

The “Generate model” menu (in red color) converts the drafting model into a structural model.



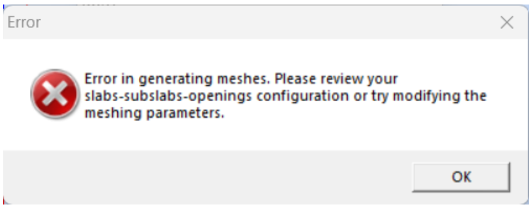

This window offers two sets of meshing with the choice of using the option of generating big meshes at center of slabs, for each. The “uniform size” leads to small meshes of square shapes. Bigger meshes of rectangular size are obtained if the “varying size” is selected. In both cases, the meshes away from the columns can increase in size, which leads to smaller numerical models, and thus quicker to run and to exploit.

It is always recommended to try several combinations of options. Depending on the topology of the slab, some type of meshing might be more convenient than expected. In general, the user should adopt the meshing with least numbers. Re-generate the model each time loads, geometry or reinforcement are modified! All the related parameters are stored in the meshes.

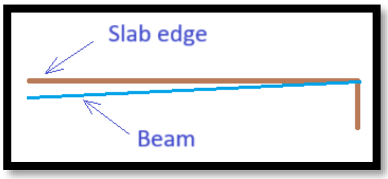
### 5.2.1 Why meshing cannot be generated?

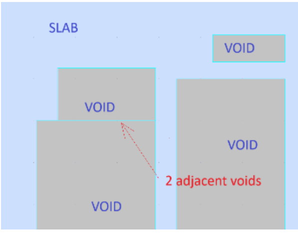
This topic is of utmost importance. The user is invited to first read chapters 3 and 4 of this manual. Several rules and hints are provided.

- When meshing fails and the displayed error message contains explicit indications about the problem (like the coordinates of the culprit element), the user should check on the message.
- When the message concerns two overlapped slabs, and the user is sure that the mentioned slabs (or subslabs) are not effectively overlapped, then one of them should be shifted either in the X or in the Y direction about 5 or 10cm. The reason is that the checking algorithm might in few particular cases generate a false warning.




When this message is displayed, it means there is a drafting problem. The user should check the sides of the slabs and openings especially with regards to very small unwanted angles that might arise between two lines that are supposed to be superposed. This error is illustrated in the following figure, where a beam line is supposed to be drawn directly over the edge of the slab. Due to a drafting error, there is a tiny angle between these two lines that leads to the generation of distorted meshes. The algorithm cannot generate such meshes. The user must visually inspect the drawing if the error is related to such cases. However, it is possible also to try different meshing options (varying size, uniform size…) to be sure at first that the problem is effectively related to drafting, and not to some inconvenient meshing option.



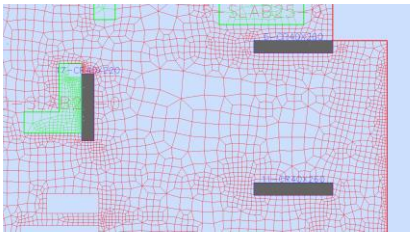




The error message shown here-above can also be triggered by the presence of two openings sharing the same side. It is very important to note that an opening should be surrounded by slab meshes, like an island. In the case of the figure at the left, the user should draw one opening than englobes both voids. <u>The closed polylines of two separate openings cannot share common sides.</u>

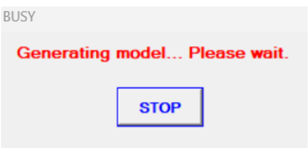
**How to solve the "Error in generating meshes" message?**

- Make sure the recommendations of chapters 3 and 4 are respected.
- Visual inspection of the meshing, looking for very close nodes or tiny angles between lines.
- Turn off the rigid links option. If the meshing works, it means that definitely the error is related to lines making tiny angles with one side of the column, or lines very close to one of its corners.
- Narrow the error investigation by filtering the geometry: Select the doubtful element and put it in layer zero. This way the element is not deleted from the drawing, but it will not be considered in the structural model. Each time an element is stored in layer zero (or any layer that does not begin with "SL-") the "Generate model" command should be launched. Once the meshing is successfully generated, it will be easy for the user to detect and solve the problem.

The rigid link option meshes the area of the column, similarly to a slab. If the sides of the columns are not well aligned or not superposed with the adjacent openings, slabs or subslabs, meshing might not be generated. It advised to turn off the rigid link option as a first trial, in case of meshing error.



**Case where meshes do not appear on the screen – with no error messages displayed.**

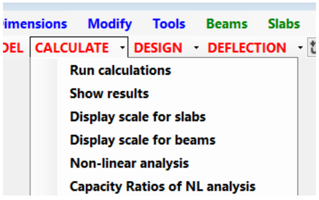



This message is displayed during the meshing generation process. When this message disappears, the meshing should appear on the screen. In some rare cases, if this does not happen, then the error is related to the presence of **columns** inside openings or outside the slab. It is recommended to turn off rigid links option in the case of such errors.

## 5.3 The Calculate menu.

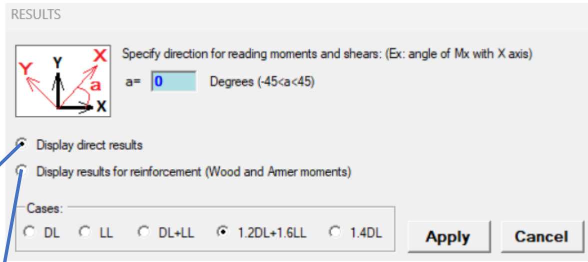
This menu should be used prior to calculating reinforcement or long-term deflections. It performs the analysis of the structure, after successfully generating the meshing. The "loads", "settings" and "reinforcement" menus should be visited before proceeding to the analysis.

### 5.3.1 Running calculations

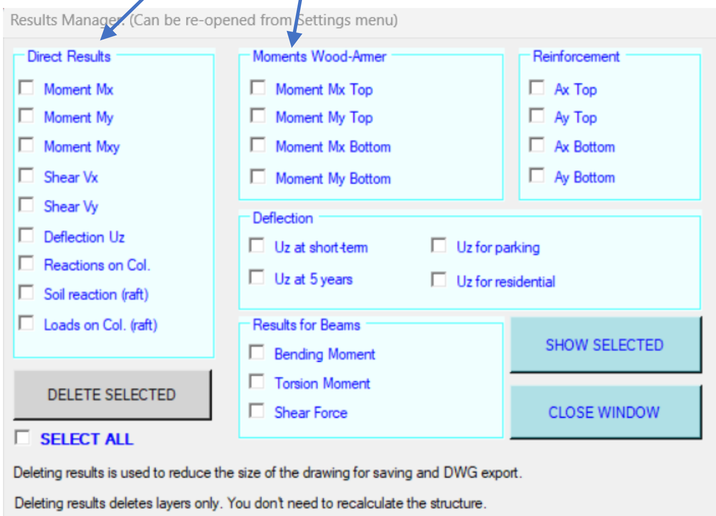



The "Run calculations" calculates the direct results, to know: Moments, shears, reactions and instantaneous uncracked deflections. If an error occurred during calculations, it means the meshing of the slab contains distorted meshes. It is recommended in this case to change the meshing type. Once calculations are performed, the user should select "Show results" to display them graphically. If modifications are operated to the geometry or loads or materials, the user must regenerate the model and rerun the calculations.

If the calculations are run, the user can at any moment select "Show results" to display them. This menu allows the user to select the desired load case (five load cases are available) and to either visualize the direct results of the Wood & Armer moments.

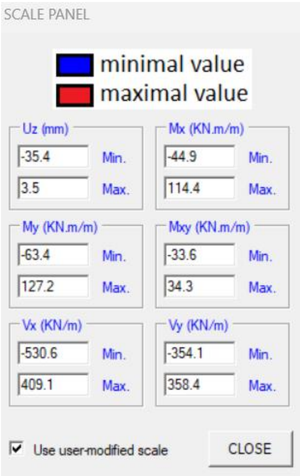



*The "Show results" menu can be called anytime.*




If the "Results manager" command is selected from the main "Settings" menu, the results that will be displayed on the screen are imposed by the latest options that have been selected in the "Show results" window. If in this window the user selected "Display direct results" under load case "1.2DL+1.6LL", then the "Results manager" will only allow these results to be displayed. If the user wants to display Wood & Armer moments, then he should select again the "Show results" and choose the Wood & Armer option with the appropriate combination. The "Results manager" shows the results that have been selected in the "Show results" menu. Results for beams are accessible within the "Results manager" window if the "Display direct results" option has been selected in the "Show results" windows.

Refer to chapter 4.3.3 for more information on the "Results manager" window.

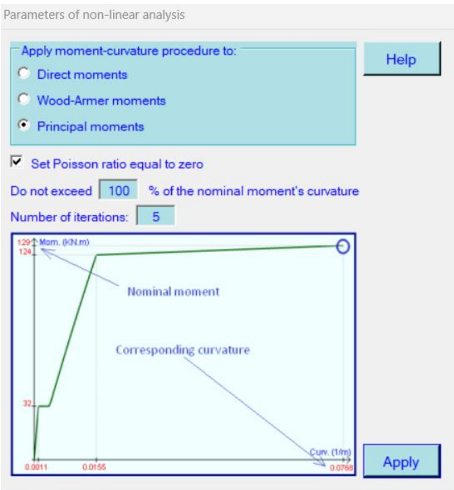



It is possible to adjust the scale color of the results for a better visualization of the colors palette.

The colors palette is distributed within the interval of the results, between the minimum and the maximum values. In the case of very high maximum or minimum, the colors range will be distributed near these areas and the remaining meshes will seem to have the same color. It is recommended in this case to introduce a new interval for results, with smaller maximal and minimal values.

In the maps, meshes with negative values are colored in blue shades, whereas meshes with positive values are colored with red shades. The user can introduce any new maxima's or minima's, select the "Use user-modified scale" checkbox, and then reselect "Show results" menu in order to refresh the colors on the screen. The new distribution of the color palette will be much more indicative.

### 5.3.2 Non-linear analysis




This innovative feature performs an elasto-plastic analysis based on the moment-curvature curve that is generated for each finite element mesh. Chapter 4.4 describes the importance of the "Reinforcement" menu where the user defines the provided reinforcement in the slab, prior to performing this analysis. This analysis is done at the ULS, for combinations 1.4D and 1.2D+1.6L.

The provided reinforcement is used to calculate the effective inertias of each mesh of the slab, about the principal directions, direct, or Wood & Armer moments (as per the user selection). The program calculates the curvatures and associates the corresponding stiffness. This latter is injected in the model in order to obtain new moments distribution, used to re-estimate the curvature to which corresponds new stiffnesses. This cycle is repeated till convergence is achieved. The user can select a number of iterations at which end the software informs the user about the state of convergence and validity of the results.

The software can go to the end of the moment-curvature curve, in the mean that it can calculate the curvature at the ultimate nominal moment. The user can impose to the software to not exceed let's say 90% of the nominal curvature (refer to the corresponding filed in the figure).

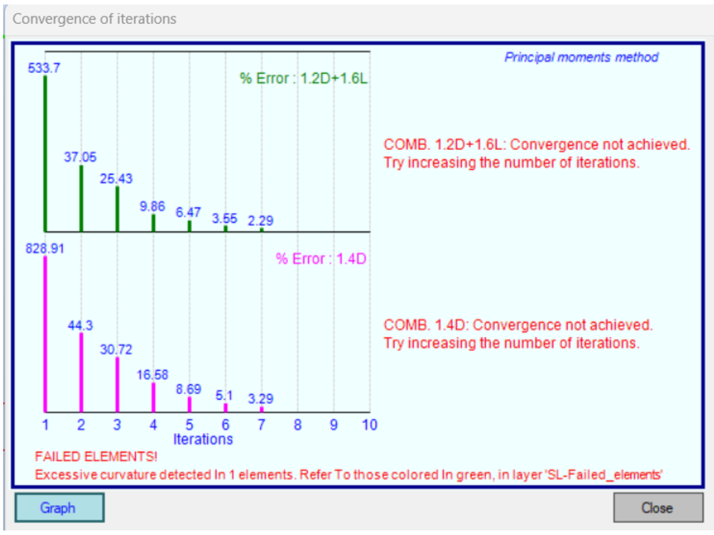
In general, considering the principal moments gives the best results for the moment-curvature. Setting Poisson ratio equal to zero illustrates the cracking state of the slab.

**What is the purpose of such analysis?**

The classical linear analysis is based on the Lower Bound Theorem where the obtained moments are compared to the nominal capacity of the section. Thie exercise is performed whilst considering the linearity between the applied forces and the obtained moments. In other words, the material is assumed elastic all long the loading path. This method is safe and justified by the above-mentioned theorem. However, it contains some drawbacks, to know:

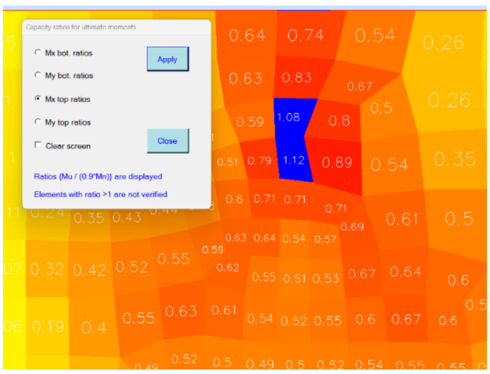
- Design at ULS supposes the sections are yielding, which means the linear response is lost.
- The inertia of the slab is constant since the gross one is considered. In fact, it is directly related to the existing reinforcement.
- The moment-curvature curve shows that a small increase in moments leads to big difference in rotations, once the yielding moment is exceeded. This ductility is not depicted in the linear analysis where rotations and moments are proportional.

The consideration of the true moment-rotation law has a direct impact on the results: The peak negative moments at the location of the columns will be softened since the adjacent meshes will share the excess in moments. Instead of obtaining the common peaks which are usually disturbing and require interpretation from the Engineer, the meshes will be more uniformly loaded over a bigger area, depending on the provided reinforcement over the whole area of the slab. Providing excess reinforcement at mid-span will lead to a bigger redistribution of the moments where those at the columns will be smaller and those at mid-span bigger. The user can therefore control the moments distribution - to an extent, indeed- instead of standing idle at the solution.
Another advantage of this analysis is that it provides a better assessment for the areas where the elastic analysis indicated a failure. Let's suppose that the classical analysis leads to high negative moments at a support, more than the nominal moment of the section, even with the use of maximal reinforcement. In such cases drop panels are required, or another solution that involves geometrical modifications. Once the provided reinforcement is placed, the non-linear analysis gives much bigger curvatures at this support, which means bigger rotations. Consequently, and to respect equilibrium, the moments at big span will be bigger, those at the support will decrease. The final results might be surprising.




*Sample of the non-linear analysis log at seven iterations*

Once the non-linear analysis is performed, a log window appears, showing the evolution of the convergence for the two considered cases: 1.4D and 1.2D+1.6L. In this sample, seven iterations were set and still the results of the last two iterations differ by about 2 to 3%, which is not acceptable. More iterations are required, as the messages in red color are indicating. At the bottom of the window a warning might appear in case the curvature that corresponds to the ultimate nominal moment is exceeded. Finite elements which curvature is excessive are colored in green once the maps are displayed. Such elements fail because of extreme yielding. The reason is insufficient reinforcement at their location. Normally, the curvature over the whole area of the slab should not exceed the maximal limits.



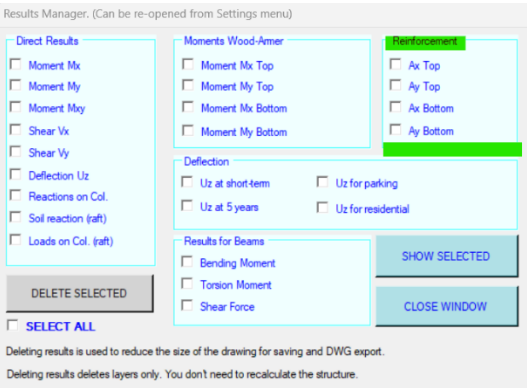
In the "Calculate" menu, the command "Capacity ratios of NL analysis" displays an innovative type of results. The ratio $M_u/0.9M_n$ is shown, where $M_u$ are the ultimate moments calculated with Wood & Armer method and $M_n$ the nominal moment that is based on the provided reinforcement. This ratio is visualized for top and bottom Wood & Armer moments as obtained from the non-linear analysis. When the ratio is bigger than 1, the mesh is colored blue. It means the inequality $M_u<\Phi M_n$ is not verified and the reinforcement should be increased. It is an efficient way to verify the provided reinforcement of the slab.

# 5.4 The Design menu.

## 5.4.1 Flexural design for slabs

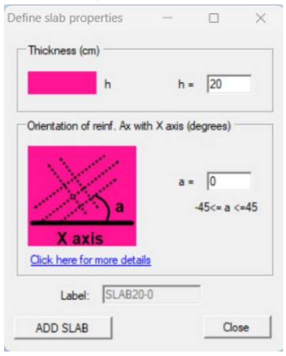
The longitudinal (flexural) reinforcement of the slab is calculated with this menu, based on the linear analysis results. It is important to note that the non-linear elasto-plastic analysis provides capacity ratios, based on the provided reinforcement by the user. The design menu calculates the required reinforcement based on the ultimate combinations for the moments. Always Wood & Armer moments are used for this purpose.

The software respects the minimal reinforcement ratio as set in the "Settings" menu. It also calculates the maximal reinforcement for flexure and if exceeded, the concerned meshes are colored blue.

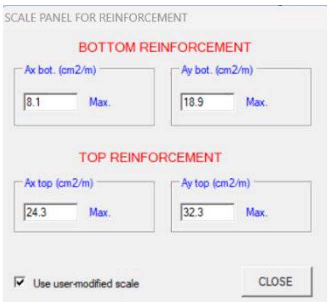



It is possible then to display the results by selecting the "Results manager" menu. To recall the features of this window, refer to chapter 4.3.3.

When displaying the results for Ax, it is important to recall the orientation of the "x" axis. When creating the slab, the angle "a" between Ax and the global "X" axis is introduced. Each slab can be defined with its proper "a" angle.

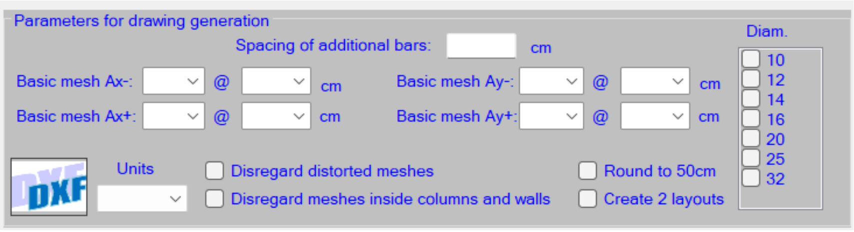



When the reinforcement maps are displayed, one should keep in mind that "Ax" map shows the reinforcement that is oriented according to the introduced "a" angle, in degrees. "Ay" is always perpendicular to "Ax". The label of each slab is composed of two numbers: One represents the thickness and the second one the orientation angle "a".




It is possible to adjust the scale color of the results for a better visualization of the colors palette.

The colors palette is distributed within the interval of the results, between the minimum and the maximum values of the reinforcement. In the case of very high maximum or minimum, the colors range will be distributed near these areas and the remaining meshes will seem to have the same color. It is recommended in this case to introduce a new interval for results, with smaller maximal and minimal values. The user should recalculate the reinforcement to update the scale.

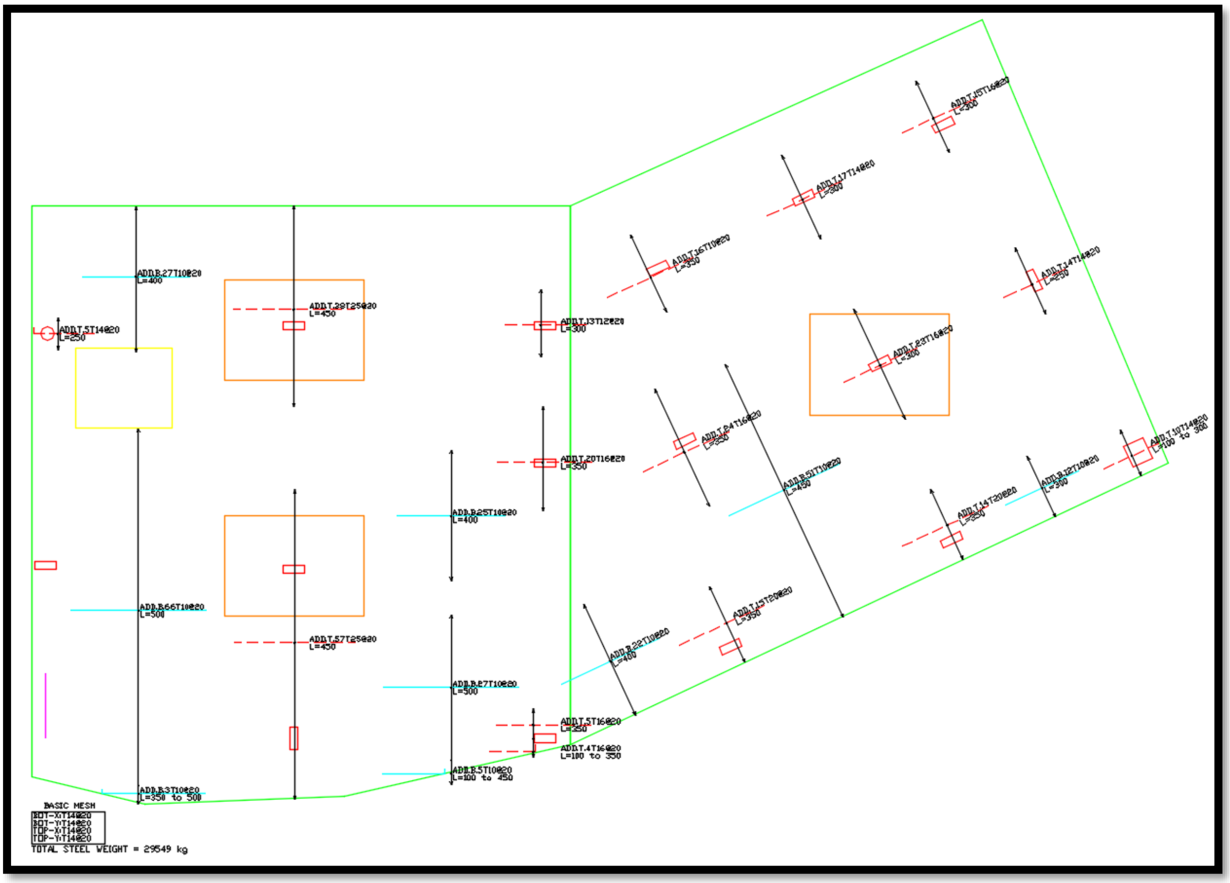



The program generates detailed drawings for longitudinal reinforcement, with consideration to slab edges and openings. The user first should specify the basic mesh.

It is recommended for the basic mesh to at least cover the minimal code requirement (As,min) to avoid obtaining additional reinforcement over the whole area of the slab. The spacing of the additional reinforcement should be as well introduced. Incoherent meshes -also called distorted- which have an angle on a vertex less than 15º or a sides ratio>3 can be disregarded due to their lack of reliability. Furthermore, meshes located within the perimeter of columns and walls can be as well ignored.

- Bars length can be rounded up to 50cm
- Reinforcement in direction X and direction Y can be grouped in two separate layouts.

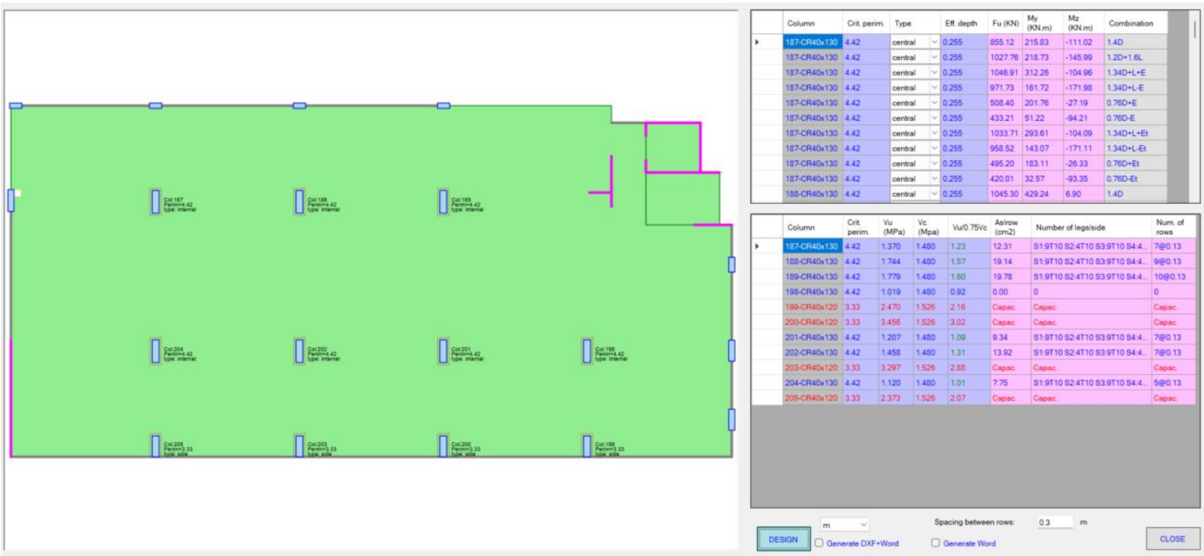
A powerful algorithm translates finite element maps into practical reinforcement layouts using several weighting criteria. The result is a human-like drawing generation where bars are hooked where they should, correctly placed around openings, and grouped by length at inclined edges.



*Sample drawing generated by the program*

### 5.4.2 Punching shear design for slabs

Punching shear is performed for envelope of ultimate combinations 1.4D and 1.2D+1.6L.



Punching shear design follows the requirements of ACI 318-25 and ACI 421.1R-20. Central, side and corner columns are automatically identified; the inner critical perimeter (located at d/2 from that of the column) is generated with regards to slab and opening edges. The upper table shows the input data that is used for the design. The slab layout shows the critical perimeters (figure here-above). Columns attached to beams or walls are not designed for punching shear.

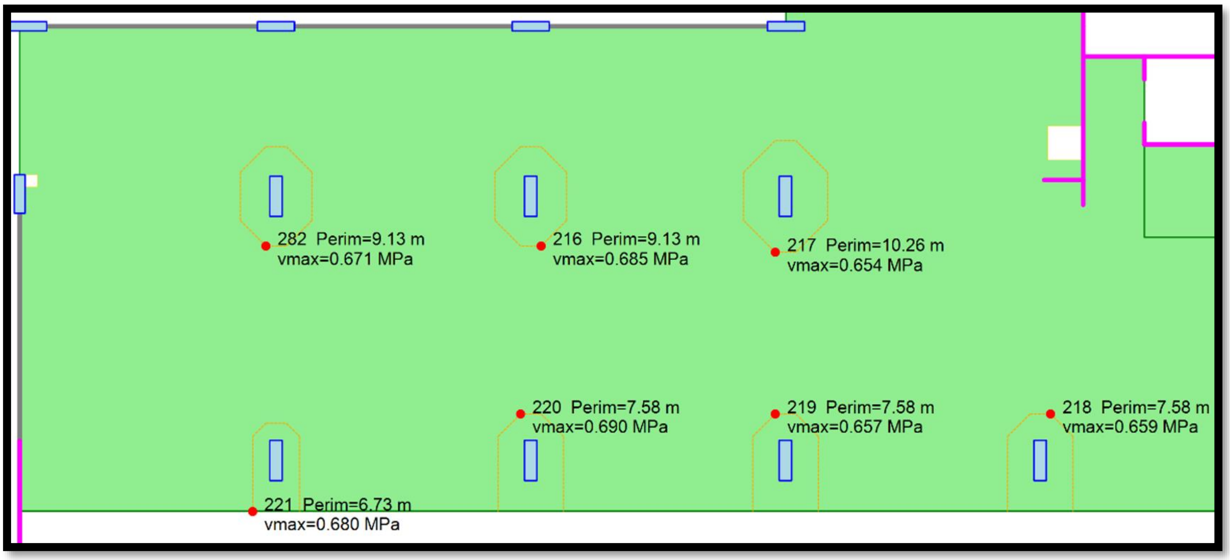
Based on the shear force and the unbalanced moments, the program calculates the maximal shear stress $V_u$ at the corners of the critical perimeter. If it exceeds the ultimate strength of the section ($V_u > \Phi V_n = \Phi * 0.5 * (f'c)^{0.5}$), the slab thickness shall be increased. In this case, the column's results are highlighted in red, with the word "Capac.", signaling the failure case. Eventually, the user can modify the forces shown in the upper table and investigate further the design.

If $V_u > \Phi V_c$ , where $V_c$ is the shear stress capacity of concrete, then reinforcement is required, provided as peripheral rows of ties; each row holding the same number of ties. The ratio $V_u/\Phi V_c$ is in this case shown in the table with green color. The program displays for each side of the critical perimeter (numbered S1 to S4 in case four sides are involved) the required number of ties to be placed.

The number of rows is deduced from the outer critical perimeter (octagonal shape). The program iterates on the perimeters to reach that where the shear stress is less than the limit specified by the Code. The required number of rows is shown in the last column of the lower table.

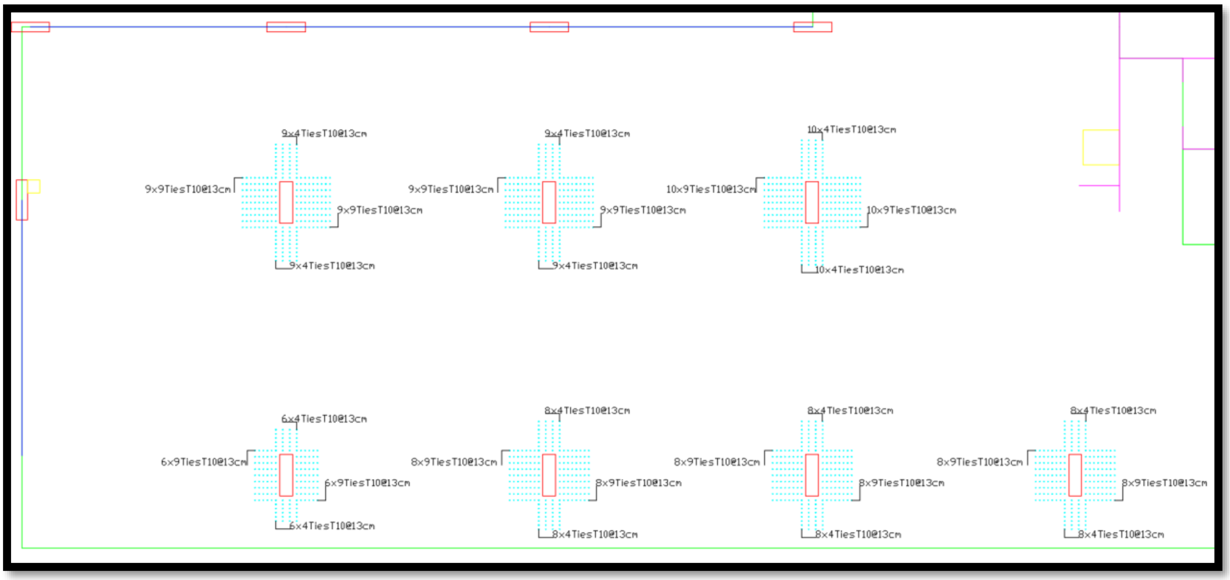
Alike the inner perimeter, the program displays the outer perimeter with consideration to the slab geometry and openings. The maximal shear stress is indicated together with the perimeter length.

- A comprehensive calculation note is generated, detailing the results at each column.



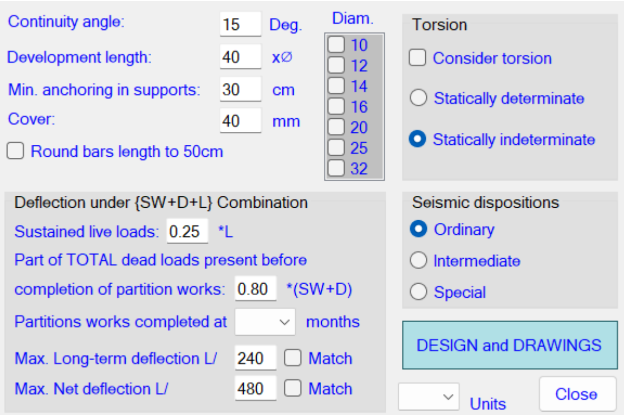

*Extract from the design report: Outer perimeters with their length and maximal shear stress*

It is possible to generate a DXF drawing that shows the layout of the punching shear zones.




*Drawing output showing the layout of the links and their disposition*

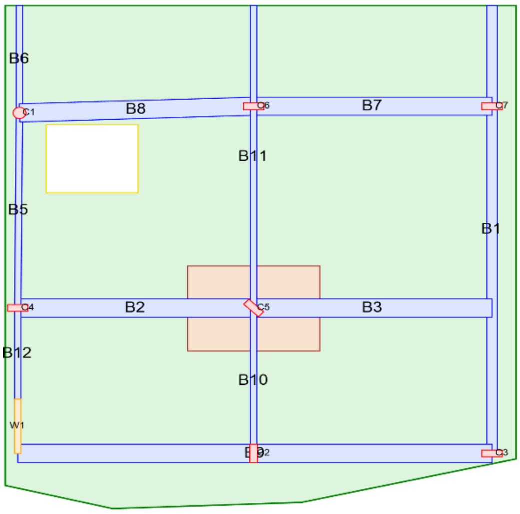
### 5.4.3 Design of beams




Design of beams is performed at ULS in the actual version of the program.

The program considers the envelope of ultimate combinations 1.4D and 1.2D+1.6L to calculate the moments are shears required for the design. For deflection design, the required service combinations are constructed from the primary simple cases which are dead and live.

Finally, a professional report (Word format) and shop drawings (DXF format) are generated.




Beams are first translated from the model to the mechanical configuration, with identification of their supports (columns, walls and supporting beams).

If a chain of beams with a continuity angle less than that specified (for instance 15º as per the input in the form at the left) the chain is translated into a continuous beam of several spans. If the intersecting angle exceeds 15º, beams are dissociated.

A key plan is generated, showing the labels of the beams, columns and walls. These labels are important for the identification of the beams and their supports.

The user can specify the development length which is mainly used to anchor reinforcement at end supports and for splicing. At intermediate supports, if seismic dispositions are not selected, the development length of bottom reinforcement is calculated based on the ratio required reinforcement / provided reinforcement, with a minimum anchoring length of 30cm or as specified by the user. When seismic dispositions are selected, the bars are fully developed at all supports.

The program starts with the diagrams of the ultimate moments and shear and adapts them based on the location of the stations of interest, as follows:

- Bending moments: the demand is located within the clear span.
- Torsional moments: the demand is located within the clear span.
- Shear forces diagram considers the demand that starts from a distance equivalent to the span effective depth from face of supports.

The above is reflected in the corresponding diagrams that are displayed in the Word report.

**Bending reinforcement**

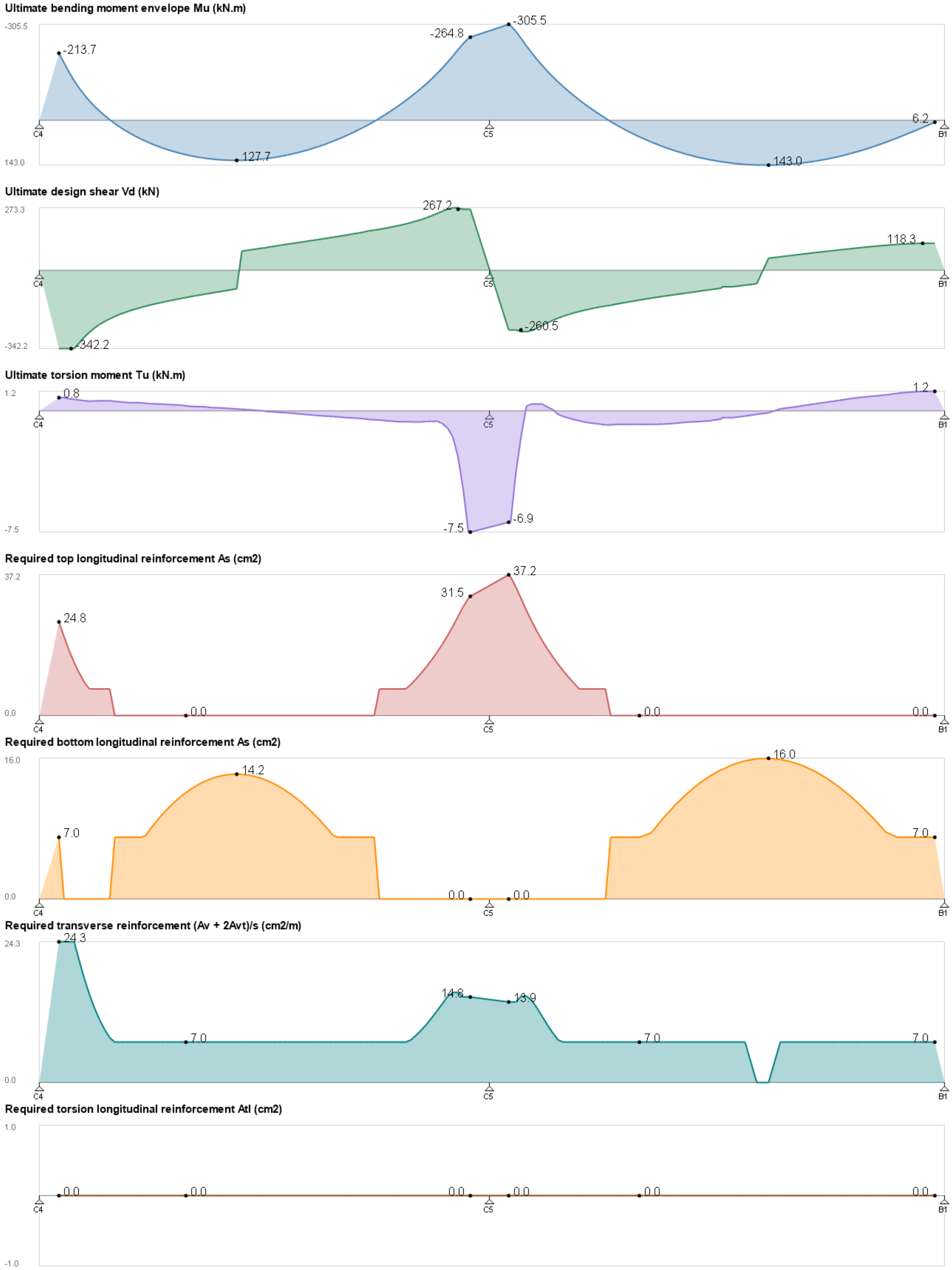
Top and bottom areas ($cm^2$) of longitudinal reinforcement are calculated and curtailed as per the requirements of ACI 318-25, complying with the minimum reinforcement condition. When the maximal reinforcement ratio is exceeded, the report shows in red color the stations concerned.

**Shear and torsion reinforcement**

Transversal shear reinforcement (Av/s) is computed as a density per linear meter of transversal reinforcement ($cm^2/m$). The program automatically considers the Code-minimal reinforcement requirements, depending on the value of the shear force. When torsion moments are included, the required transversal reinforcement Avt/s is added to that consequent to shear as follows: Av+2*Avt. The required longitudinal reinforcement Atl ($cm^2$) is placed at the faces of the beam, like skin reinforcement, and displayed in the report as a separate diagram. It corresponds to the required longitudinal reinforcement.

- It is important to note that torsional moments have a direct impact on transversal reinforcement. Torsional strength is provided by additional transversal and longitudinal reinforcement.
- When the intermediate or special moment-resisting frames option is selected, the program adds to the shear that is present in seismic combinations the part computed from the free-body diagram, assuming nominal moments at the ends of the beam (intermediate frame) or probabilistic moments (special frame). The related Code sections are 18.4.2.3 and 18.6.5. Seismic dispositions affect only shear reinforcement.
- If torsional stiffness is required for equilibrium (statically determinate structure), then the actual ultimate torsion moment diagram governs the design.
- When the structure can adapt through forces redistribution (most of the cases), the structure is classified as statically indeterminate and the cracking torsion Tcr is considered for design, instead of the actual demand.
- When the beam fails under shear or torsion, the concerned stations are shown in red color.

Typically, forces and reinforcement values are shown at faces of supports and at mid-spans. Except for shear where the demand and the reinforcement area are shown at a distance equal to the section effective depth from the face of support.

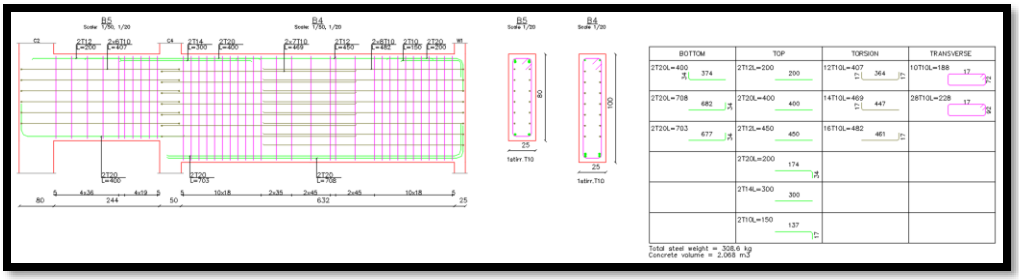


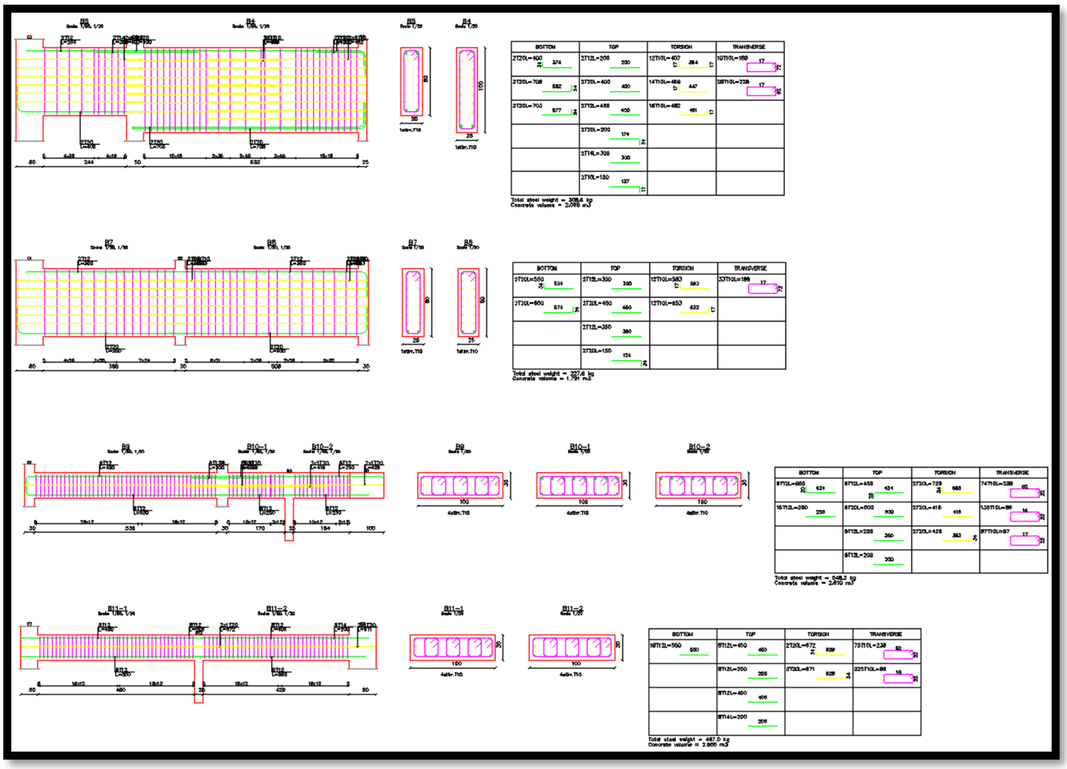

*Sample beam output for ULS design*

**Drawings generation**

High-quality shop drawings are generated for all beams based on

- At end supports, if the straight development length does not fit, standard 90º-hook is used. If the height of the beam cannot accommodate it, it is replaced with a 180º-hook.
- At intermediate supports, the program calculates the required development length for bottom reinforcement, with a minimum of 30cm. If seismic dispositions are set, the bars are fully developed.
- Longitudinal torsion reinforcement is always developed as per Code requirements. If the straight development length does not fit within the support, a horizontal 90º-hook is used.
- Bottom and top reinforcement are curtailed as per the Code provisions. Lengths of the second and higher-order layers are hence calculated and do not follow approximate rules.
- Skin reinforcement is provided when the depth of the beam exceeds 90cm.
- Transversal reinforcement follows the demand diagram. The program starts by selecting the smallest diameter ($\Phi$10mm) and calculates the required spacing between rows. If this spacing is too small, a bigger diameter is selected. When seismic dispositions are required, the program follows the minimal spacing requirements of the Code.
- For each beam, an elevation with section on each span is generated. A detailed bar bending schedule displays, for each type of reinforcement, total and partial lengths. Steel and concrete quantities are shown at the bottom of the table.





#### 5.4.3.1 Deflection calculations

The “Design and Drawings” perform the ULS design and generates drawings. Furthermore, calculations of cracked time-dependent deflections are performed and detailed in a Word report that shows diagrams and tabulated results.
The provided reinforcement (top & bottom) and the service moment are used to compute the effective inertia at each station. The double integration of the curvature over the whole beam gives the deflection diagram, with a new redistribution of the service moments. This iterative process will converge to the final instantaneous “cracked” deflection. In the case where the supports are beams, the corresponding displacement is not nil. The program calculates in this case the relative deflection to a line that joins the deflected state of the supports.

Several combinations are considered:

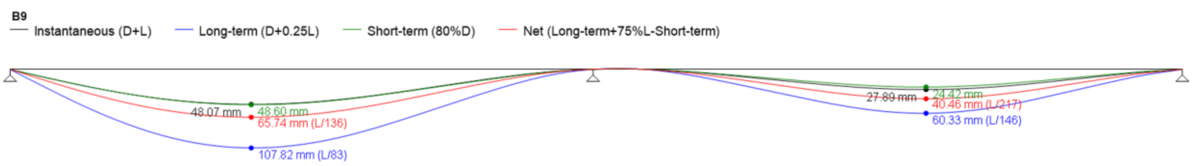
- Comb1: “SW+D+L”
- Comb2: “SW+D+$\alpha$L”
- Comb3: “$\beta$(SW+D)”

- The coefficient $\alpha$ represents the fraction of live loads that is sustained.
- The coefficient $\beta$ represents the fraction of the total dead loads (SW+D) that is executed at the time where partition works are completed.
- The user needs to specify the time “$t_1$” at which partition works are completed.
  - At 1 month, creep coefficient is 0.5
  - At 2 months, creep coefficient is 0.75
  - At 3 months, creep coefficient is 1
- The program assumes that after five years, the creep coefficient reaches a maximum of 2.

- Case LT: Deflection at 5 years (long-term) is calculated with Comb2 at t=5 years.
- Case ST: Short-term deflection corresponds to Comb3 at $t_1$.
- Net deflection: Case LT - Case ST + {Comb1-Comb2}.

Long-term deflection is usually compared to the Span/240 limit. In the presence of brittle partitions, the net deflection should be compared to Span/480.

The program displays the obtained deflection diagrams with the maximum values for each span.




The ratio of compression reinforcement $\rho'$ is considered when computing the time-dependent amplification coefficient $\lambda = \frac{\zeta}{1+50\rho'}$. Compression reinforcement is always considered in the calculation of $\lambda$.

**Control of deflections**

If the “Match” checkbox is active and the obtained maximal deflection is less than the specified limit, the ULS provided reinforcement (tension and compression) is increased to improve the flexural stiffness of the concerned span. This process is automated by the program. The drawings generated reflect the final state of the reinforcement provided.

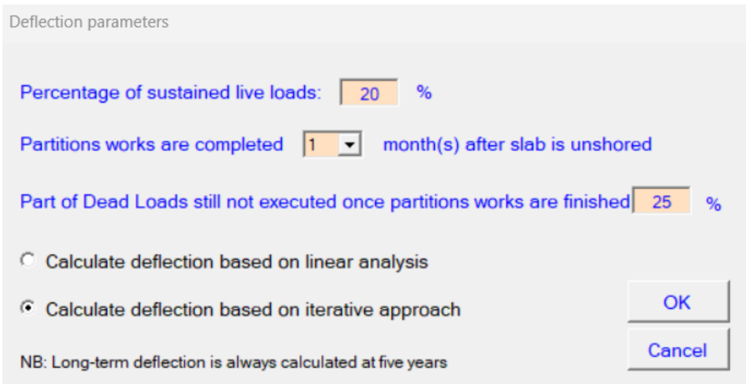
The adjustment process consists of increasing the diameters of the original reinforcement that is obtained from ULS design and adding new layers. This does not guarantee the obtention of deflections satisfying the imposed criteria.

## 5.5 The slab deflection menu.

SLABS EXPERT provides a powerful tool for the calculation of the cracked long-term deflection for the slab. Combined with its refined meshing capacities, staging of loads, the exact provided reinforcement and the moment-curvature algorithm, the user benefits from the best instruments to attain the most accurate deflection results.
Before calculations, the user should introduce the provided reinforcement if he wishes to obtain a precise deflections map. This can be done with the “Reinforcement” menu. The calculated reinforcement (required) is not considered by the software when determining the cracked inertias.

> If no basic mesh is defined and no additional reinforcement specified, the software supposes that the reinforcement is equal to the minimal ratio that is set in the “Settings menu”. This will definitely lead to exaggerated deflection values.

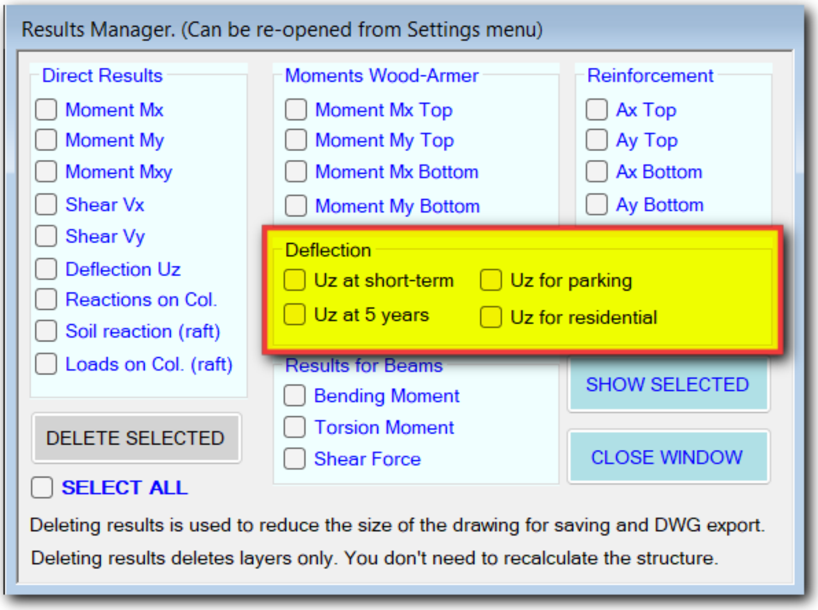
### 5.5.1 Calculate slabs deflection.




*Parameters used for deflection calculations*

- The percentage of sustained live loads designates the part of live loads that is permanently present on the slab. This part will be added to dead loads and will be magnified by the creep function $\lambda(t)$.
- When the props under the slab are removed, instantaneous deflection applies, and the creep effect starts due to the presence of compression stresses. If partition works -among others- are completed one month after, there is a tiny part of creep that already occurred. A plastic deflection occurs, equal to $\lambda(1)*f_0$ where $f_0$ is the instantaneous deflection due to all loads already present on the slab.
- It is possible to specify the remaining part of total dead loads (where the self-weight of the slab is accounted as a part of this total) that is still not present when the partition works were completed. It could be, for instance, a part of entire tiling works. The short-term deflection (that calculated at the completion of partition works) considers only the executed part. For instance, supposing that as per the figure here-above 75% of the total dead loads (as mentioned, total dead loads comprise the self-weight) are executed when the partition works are finished, one month after the slab is unpropped. The instantaneous defection is due to 75% of these dead loads. After one month, the time-related deflection is $\Delta_1$=f(0.75D)*(1+$\lambda$(1)). This deflection is labeled in the “Results manager” as **“Uz at short-term”**. The words short-term refer to the consideration of a small part of the creep, since just one month is accounted.

- **“Uz at 5 years”** displays the deflection five years after the slab has been unpropped. It is the long-term deflection.
    - It considers the creep that is done by the executed portion of dead loads during the first month $f_0(0.75D)^*\lambda(1)$ where for illustration 75% is the assumed portion, and one month is the duration for the completion of partition works
    - It considers also the creep over the duration of 5 years minus one month $\{\lambda(60)-\lambda(1)\}$ that affects the deflection of sustained loads, which are in the current example D+0.2L. The literal expression for the long-term deflection becomes:
        - $\Delta_{60} = f(0.75D)^*\lambda(1) + f(D+0.2L)^* \{1+\lambda(60)-\lambda(1)\}$




*Four stages of deflections can be displayed.*

Usually, long-term deflection is not bound to a limit. It is however recommended to avoid exceeding the span/240, whatever is the usage of this slab (roof, residential, parking….). The deflection is an indirect indicator for vibration susceptibility. Excessive long-term deflections are usually accompanied with resonance at low frequencies. From the esthetical point of view, it is advised to provide a camber.

- **“Uz for residential”** is the net deflection that is of big importance when brittle partitions are attached to the slab. It should not exceed span/480.
    - Uz for residential = Uz at five years – Uz at short term + Instantaneous Live. This latter represents the deflection of 100%-20%=80% of the live loads considered alone, supposing that 20% of the live loads were considered sustained.

- **“Uz for parking”** is the instantaneous deflection of the non-sustained part of the live loads, considered alone. In the case of cars parking, ACI 318-25 recommends comparing this deflection to span/380.

In the “Deflection parameters” window, a quick (linear analysis) and lengthier one (iterative) are proposed. The iterative approach is more accurate. In both cases the provided reinforcement should be specified.

END OF DOCUMENT