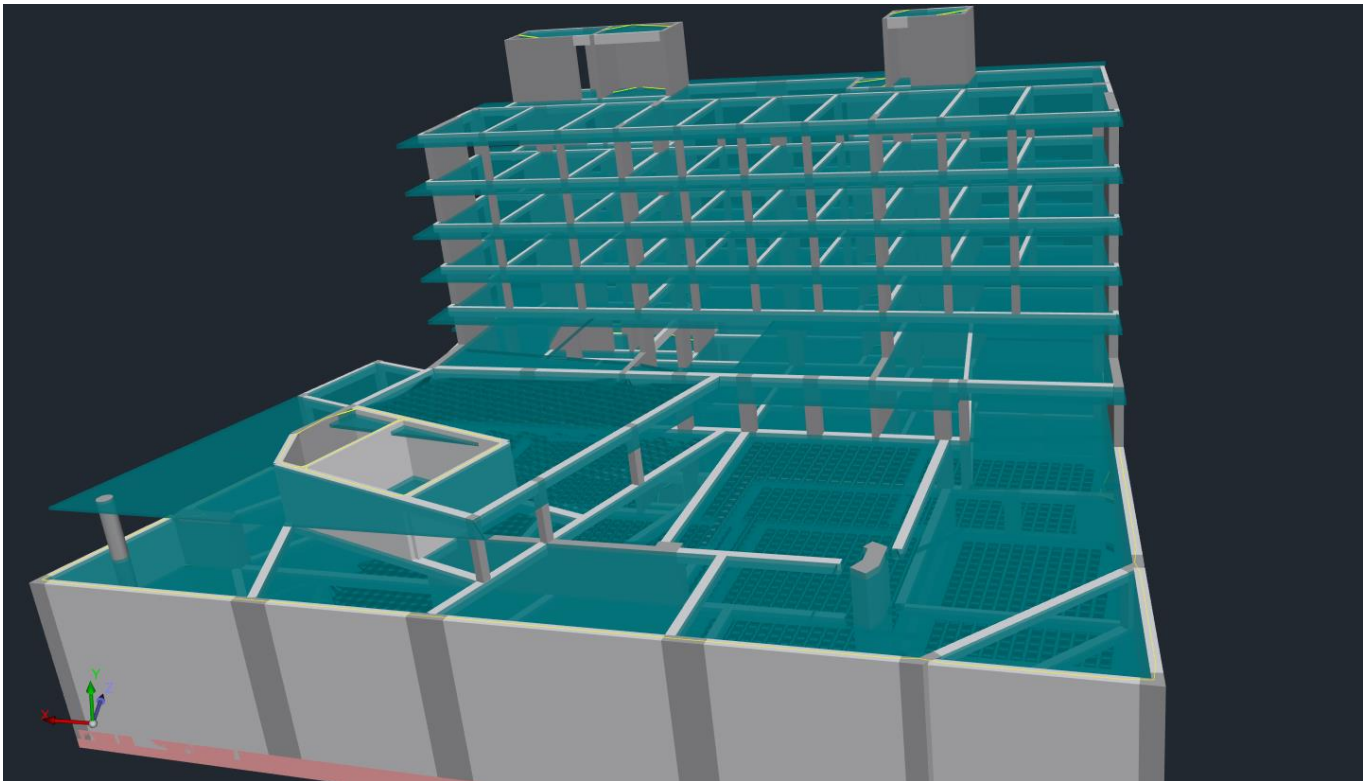




**SCADA Pro**<sup>™</sup>  
Structural Analysis & Design

# User's Manual

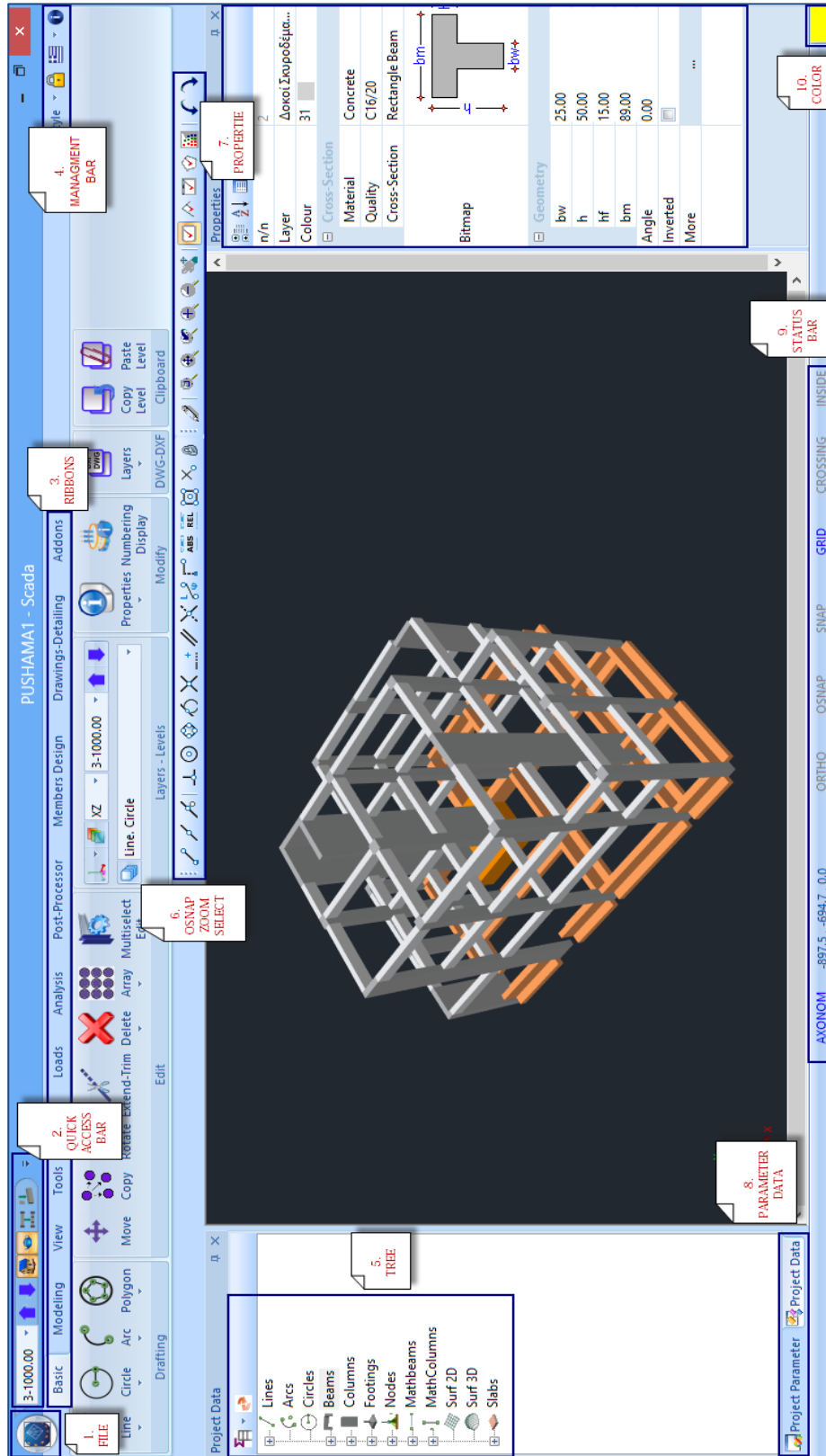
## 5.SLABS



## CONTENTS

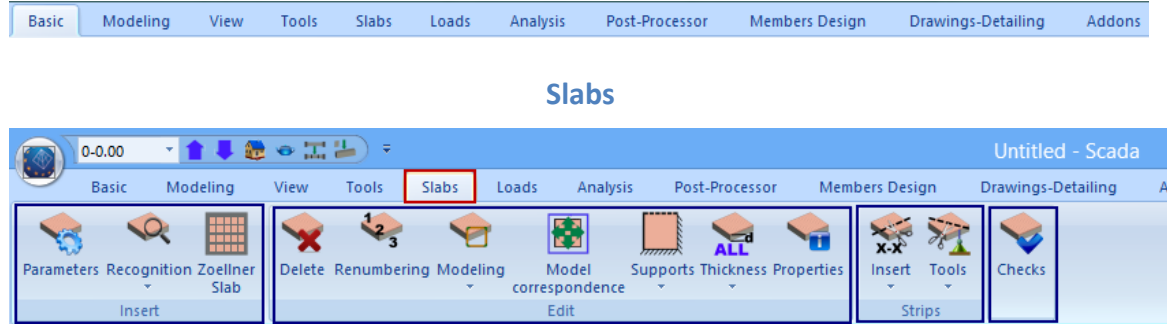
<b>I.</b>	<b>THE NEW UPGRADED INTERFACE OF SCADA PRO .....</b>	<b>3</b>
<b>II.</b>	<b>DETAILED DESCRIPTION OF THE NEW INTERFACE .....</b>	<b>4</b>
	SLABS.....	4
1.	INSERT.....	5
1.1	PARAMETERS.....	5
1.2	IDENTIFICATION.....	6
1.2.1	OVERALL.....	6
1.2.2	BY PICK .....	6
1.2.3	FROM MEMBERS.....	7
1.3	ZOELLNER SLAB .....	7
2.	EDIT .....	9
2.1	DELETE .....	10
2.2	RENUMBERING .....	10
2.3	MODELING.....	10
2.3.1	RECTANGLE .....	11
2.3.2	INCLINED RECTANGLE .....	11
2.3.3	TRIANGLE.....	12
2.4	MODEL CORRESPONDENCE .....	12
2.5	SLABS SUPPORTS .....	14
2.5.1	OVERALL.....	14
2.5.2	BY PICK .....	14
2.5.3	EDIT.....	14
2.6	THICKNESS .....	15
2.6.1	OVERALL.....	15
2.6.2	BY PICK .....	15
2.6.3	EDIT.....	15
2.6.4	MINIMUM THICKNESS.....	16
2.7	PROPERTIES.....	16
3.	STRIPS .....	20
3.1	INSERT .....	20
3.2	TOOLS.....	20
4.	CHECKS.....	21

## I. THE NEW UPGRADED INTERFACE of SCADA Pro



## II. DETAILED DESCRIPTION OF THE NEW INTERFACE

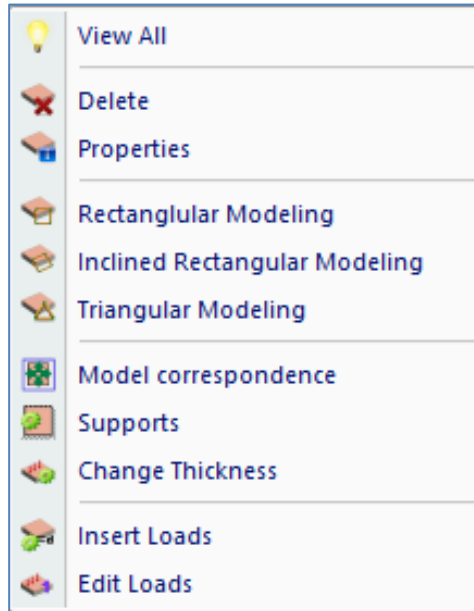
In the new upgraded SCADA Pro, all commands are grouped in 12 units.



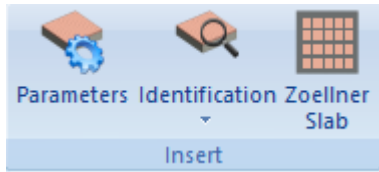
The 5<sup>th</sup> Unit called “Slabs” includes the following four groups of commands:

1. **Insert**
2. **Edit**
3. **Strips**
4. **Checks**

*When the Ribbon "SLABS" is activated, you can use "right click" mouse button to display slabs' main commands.*



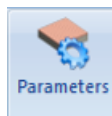
### 1. Insert



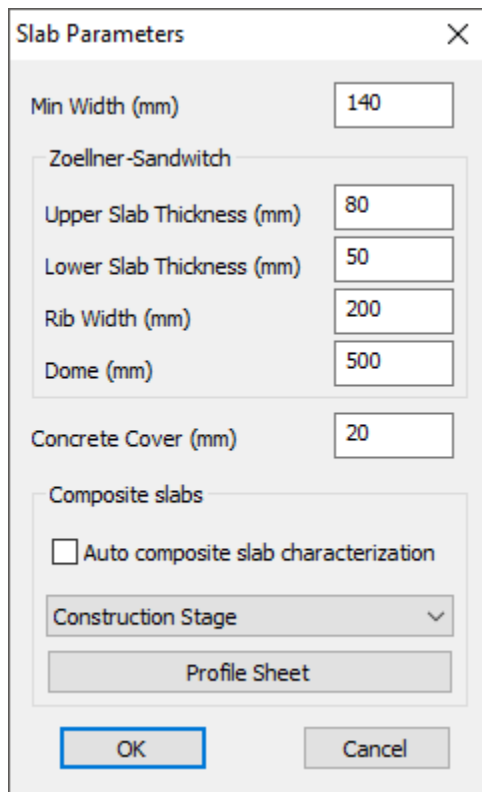
The command group “Insert” contains commands for inserting slabs.

- ⚠ *A precondition for inserting slabs is the existence of the mathematical model.*
- ⚠ *For the application of “Slab” commands, the 2D representation is proposed (per level).*

#### 1.1 Parameters

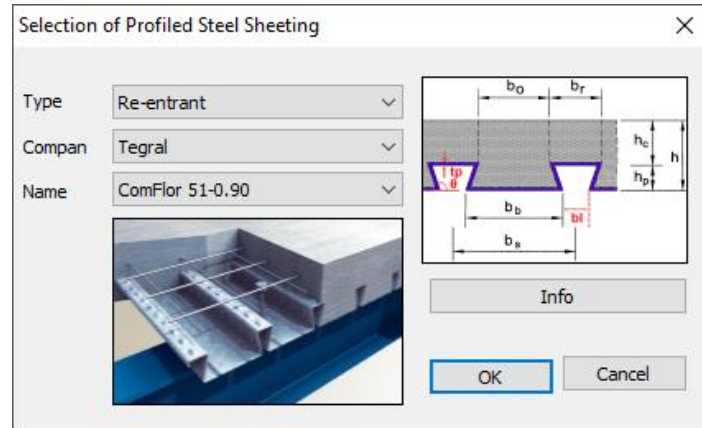


Select the command and the following dialog box is displayed:



Type the minimum slab’s width and the concrete cover (mm). In case of Zoellner slabs, type also the corresponding geometric features.

The bottom part of the window regards the Composite Slabs and you can define the characterization and the Profile Sheet. Detailed information you will find in the relative manual.

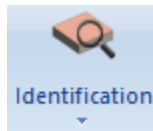


⚠ During the identification of the slabs, the program considers the value for the Min Width given in "Parameters", calculates the required (based on the Greek norm) and comparing the two values, keeps the maximum.

Min Width (mm)

⚠ Using Eurocodes, the **final thickness** of the slab results after the deflection checks, which however can not be proposed in the initial recognition of the slab because the calculation requires the reinforcement.

## 1.2 Identification



In SCADA Pro the identification of the slabs can be implemented in two way:

- automatically (overall active level or by picking plates) or
- by defining the corresponding members.

### 1.2.1 Overall

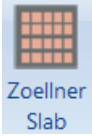
Solid slabs automatic identification in the closed boundaries of the active floor.

### 1.2.2 Pick

Solid slabs identification using left click inside the closed boundary, selected one by one.



After inserting a slab, the circular symbol with the corresponding information is displayed; the number and the thickness in cm (the greater value between the minimum you set and the one as result from the bending resistance check), in a circle. Around the circle, lines are displayed representing the slab's support conditions:

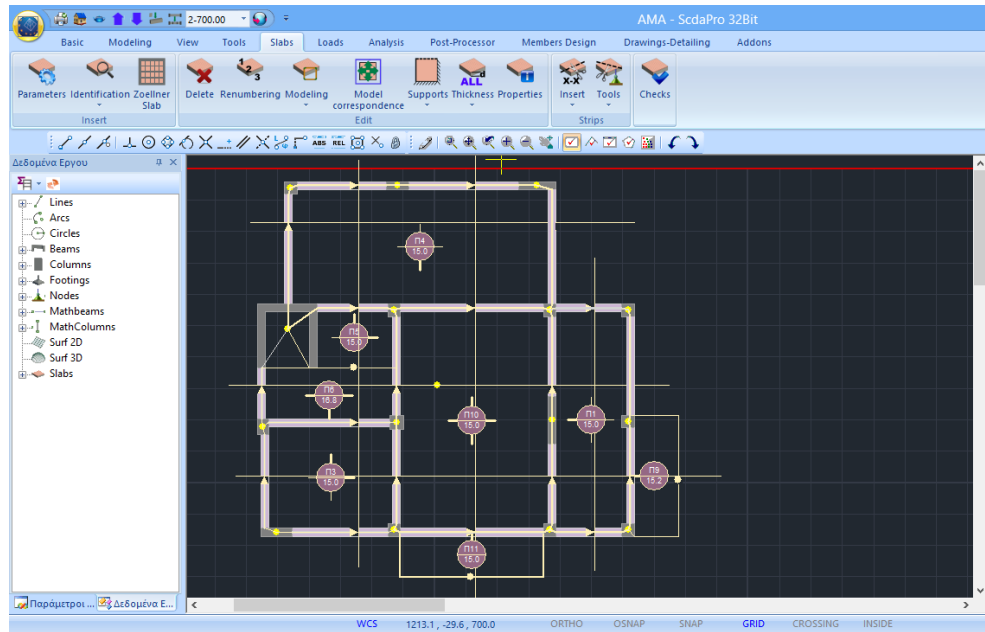


- Thick Line: slab continuity → fixed.
- Thin Line: slab discontinuity → joint.
- No Line: free end (case of balconies).

⚠ *The sign “?”* in the symbol of the slab, indicates that the slab has not rendered correctly and needs "Modeling".



The program automatically inserts solid slabs. The user can modify them and change the type.



### 1.2.3 From Members

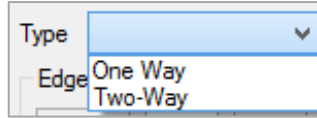
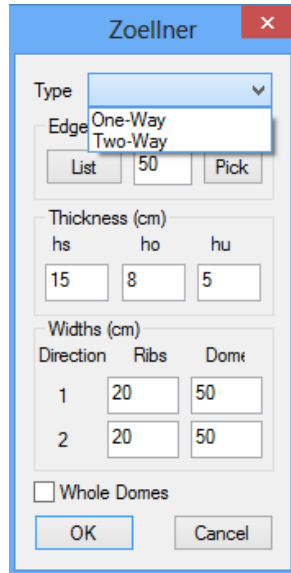
This command is used for the manual insertion of solid slabs. Select successively, one by one, the elements surrounding the slab (beams, columns, lines). Right, click to exit command.

Manual insertion is required, where the edges of the slabs are composed of many members thus it is difficult for the program to identify them.

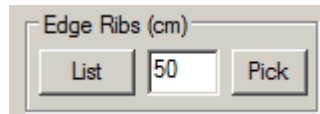
### 1.3 Zoellner slab

This command is used to convert a solid slab to a Zoellner or Sandwich slab.

Select the command, left click inside the slab and the corresponding dialog box is displayed:



In the drop-down list “Type”, select if the slab is connected in one or two directions.

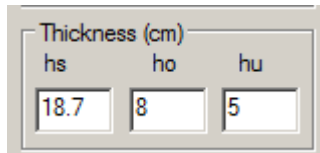


Then define solid zones’ widths (cm). Type the number and click the button “Pick”. Left click on the side of the beam considered as an outline of the slab. Then, the boundary of the solid zone will be placed in parallel with the beam at a distance equal to the width that was defined previously. The line is drawn (boundary of the solid zone) and with left click, the direction is indicated.

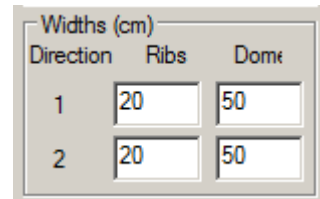
- ⚠ *The solid zones should be defined in a row and circularly, clockwise or counterclockwise.*
- ⚠ *If you press right click during the definition of the solid zone, the “Zoellner” dialog box opens, so that you can define different widths for the next solid zone.*

After defining the latest solid zone, right click to open the window and complete it by typing thickness, ribs, and domes widths.

In the next section, the thickness of the slab is defined.



In “hs” type the slab’s total thickness (cm).  
 In “ho” type the up side solid slab’s thickness (cm).  
 In “hu” type the down side solid slab’s thickness (cm) for Sandwich slabs. Otherwise, type 0.



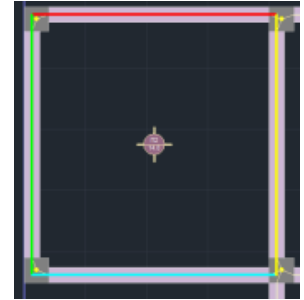
Define ribs and domes widths:  
 Type the widths in each direction.

- Direction 1 is the direction parallel to the slab’s side that you will define when the program will ask you to do so. For Zoellner slabs on one direction, Direction 1 is the main slab’s direction.
- Direction 2 is the other one (vertical).

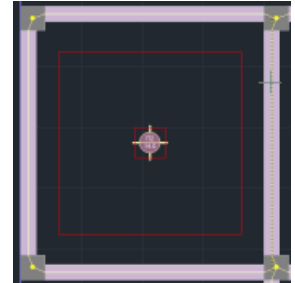
Activate the checkbox  Whole Domes to receive only whole domes.



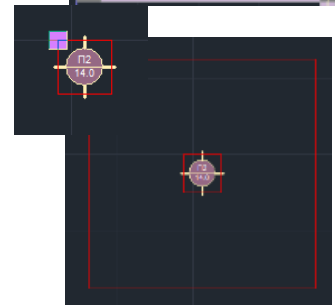
Click “OK” to display the mathematical model of the selected slab. Then the program asks you to define Direction 1 (the side of the slab, which will be parallel to the beam of the first direction).



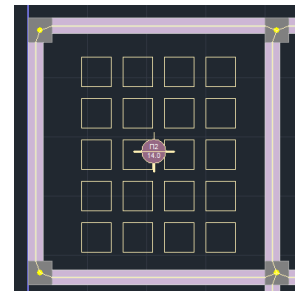
Select the side of the slab’s model and the gap with the defined geometry is placed automatically in the center.



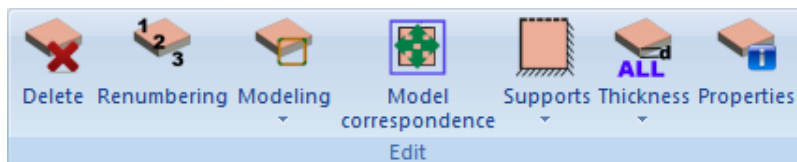
To define where to start putting the domes, first, click on a dome’s vertex and then on a slab’s vertex.



Automatically the solid slab is converted to a Zoellner slab:



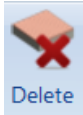
## 2. Edit

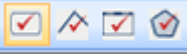


By using the commands group “Edit”, and after the insertion of the slab, you can use the following commands:

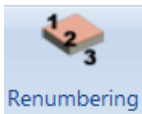
- Delete
- Renumbering
- Modeling
- Model Correspondence
- Support
- Thickness
- Properties

## 2.1 Delete

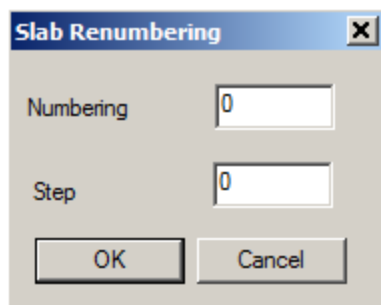


This command is used to delete one or more slabs. Select the command, use a command from the selection toolbar  and then right click on the slab or slabs. Right click to exit the command.

## 2.2 Renumbering

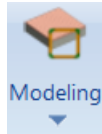


Select the command to renumber the slabs placed and numbered automatically by the program. The following dialog box is displayed:



Type the starting number and step. Left click on the slabs, one by one.

## 2.3 Modeling



Slabs with arbitrary shape need to be modeled using an equivalent rectangular, inclined rectangular or triangular slab i.e. defining a rectangular or triangular slab similar to the real one.



The sign “?” in the symbol of the slab, indicates that the slab has not been rendered correctly and needs to be modeled (“Modeling” command group).

### 2.3.1 Rectangle

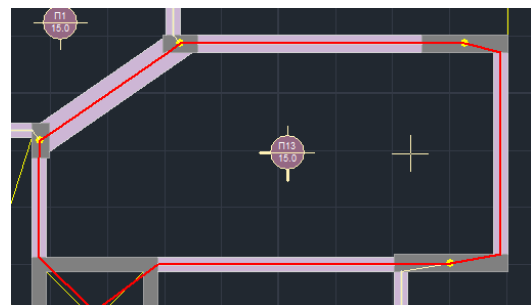
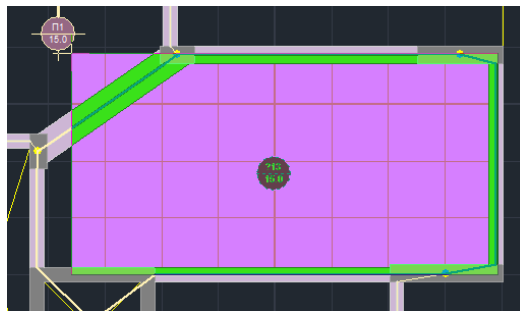


This command is used to model arbitrarily shaped slabs with the equivalent rectangular slabs. The sides of the slab are parallel to the Global X and Z Axes.

Select the command and left click into the surface of the slab that will be modeled.

To define the equivalent rectangular:

- Left click inside the arbitrary area of the slab
- Left click on the first top, move the mouse diagonally up to the second top drawing a rectangle and left click again.



### 2.3.2 Inclined Rectangle



This command is used to model arbitrary slabs with equivalent inclined rectangular slabs. The sides of the slab will be inclined to the Global X and Z Axes.

Select the command and left click into the surface of the slab that will be modeled.

To define the equivalent inclined rectangle:

- Left click inside the arbitrary area of the slab
- Left click on a slab side to define the direction of the equivalent inclined rectangular slab
- Left click on the first top, move the mouse diagonally up to the second top drawing a rectangle and left click again.

### 2.3.3 Triangle



This command is used to model arbitrary slabs with equivalent triangular slabs. Select the command and left click into the surface of the slab that will be modeled.

To define the equivalent triangle:

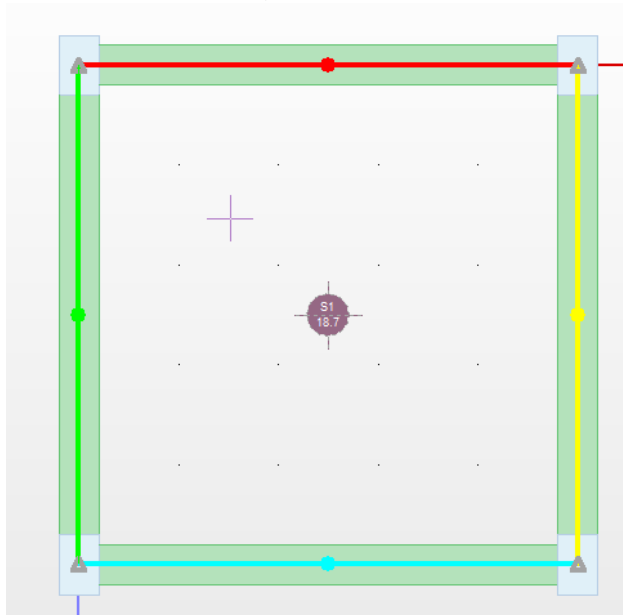
- Left click inside the arbitrary area of the slab
- Left click on the three sides of the equivalent triangular slab.
- Triangular slab always needs to be modeled even if the sign “?” is not displayed in their symbol.

### 2.4 Model correspondence

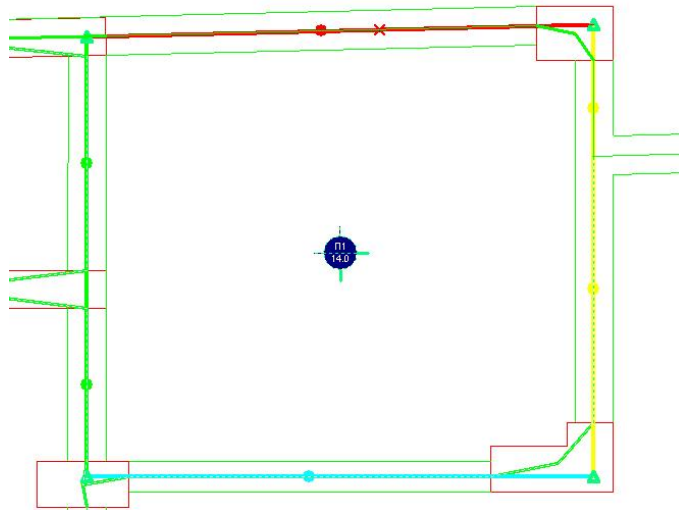
After the modeling of the arbitrary form of slabs, define the correspondence between the sides and the tops of the equivalent slab and the ones of the real slab. The members and the lengths of the sides of the physical model match the ones of the mathematical model with this process.



- Select the command
- Left click inside the slab to display the equivalent rectangular or triangular slab (slab's mathematical model)

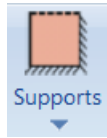


- Select one side of the mathematical model of the slab. An X appears on it.
- Left click on the corresponding physical model member (in the middle of the member a circle that takes the color of the corresponding mathematical member is displayed).
- Right click to complete and left click again to continue with the rest sides of the mathematical model of the slab.



- Finally, assign to each vertex of the equivalent slab (symbolized with a triangle) the corresponding physical point to reduce the length of the sides of the physical to the mathematical model. Consequently, loads of the equivalent slab will be distributed to the real lengths of the physical members.
  - For the assignment first, select the top of the mathematical model and then left click on the new location. Repeat the process for the remaining three vertices of the mathematical model without using the right mouse button.
- ⚠ *To do the matching of the mathematical model's sides to the physical ones you don't have to choose the columns.*
- ⚠ *If before the matching of the sides, the mathematical model's sides appear with the same color as the physical members, you don't have to do the matching, unless you wish to make changes.*

## 2.5 Slabs support

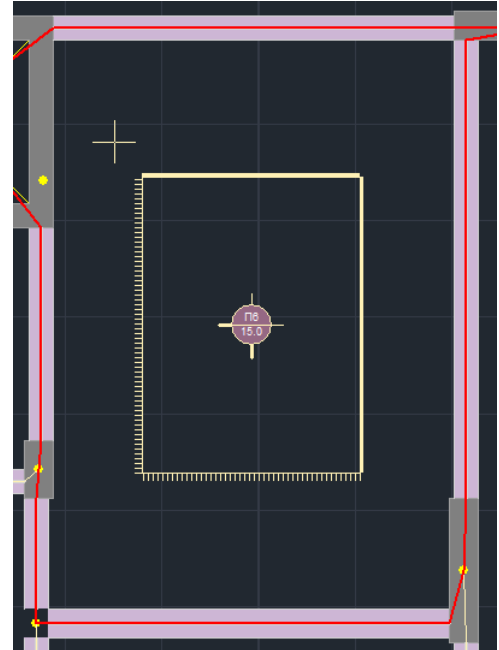


This command is used for the automatic definition of the support conditions of the slabs. First, they are identified and then they are modeled.

The symbolism used by the program is:

	:	fixed
	:	joint
	:	free

Initially, the program assumes free ends at the edges, where there is no continuity with another slab and fixed supports where there is continuity with another slab.



### 2.5.1 Overall



Select the command to define slab’s support conditions automatically for all the modeled slabs of the current level.

### 2.5.2 Pick



Select the command to define slab supports, but not automatically. Left click inside the area or the slabs.

### 2.5.3 Edit

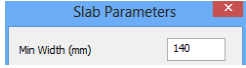


Select the command to modify slab’s supports. The supports of the slabs are displayed on the screen. Left click on the support to change it. Each click changes the support following the series: fixed → joint/free

## 2.6 Thickness



This option checks the slenderness of the identified slabs and determines the required thickness of slabs in cm automatically.

⚠ The program takes into account the minimum thickness  (defined on Parameters) and chooses the greater value.

⚠ This formula determines the required thickness:  $hf = d + \text{cover}$  according to the Greek norm. Select:

### 2.6.1 Overall



to calculate the thickness of all slabs of the current level automatically.

### 2.6.2 Pick



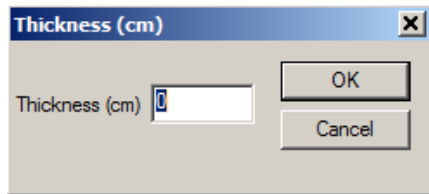
to calculate the thickness of the slabs that you select by clicking on them.

### 2.6.3 Edit



to modify slab's thickness defined previously.

Select the command and the following dialog box is displayed:



Type the new value, then press the button “OK” and left click inside the slabs. Then thickness is displayed in the slabs’ symbol.

### 2.6.4 Minimum Thickness



if by using the command "Edit", you change the thickness value and give a higher value the command "Minimum Thickness" will restore the original value of the thickness of the selected slab.

#### ATTENTION

⚠ *Using Eurocodes, the **final thickness** of the slab results after the deflection checks, which however can not be proposed in the initial recognition of the slab because the calculation requires the reinforcement.*

In the new version of SCADA Pro, there are also the deflection checks of the slabs.

The Deflection Checks based on 7.4.2 and 7.4.3 of EC2 and presented at the end of the results of each slab.

The results of both checks are displayed separately.

```

+-----DEFLECTION CONTROL (EC2 7.4.2 & 7.4.3)-----+
| 1/d | 1/d |Suf.|Suggested.min| |Max. M | du1 | a |1/a (perm.)|Suf.|
| | perm. | |thick. hs (mm)| | (kNm) | (mm) | | (mm) | |
+-----+-----+-----+-----+-----+-----+-----+
| 46.15| 146.64|YES | 61 | | -3.69| 2.25 | 250 | 24.00 |YES |
+-----+-----+-----+-----+-----+-----+-----+
  
```

From the first check, emerge the minimum suggested a thickness of the slab, which cannot be proposed during the initial recognition of the slab because for the calculation the reinforcement is required. In case of insufficiency, you must change the slab thickness.

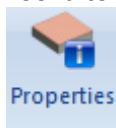
In the first check, the intensive forces are not involved, while the second check involves the serviceability combination(s).

⚠ *In case of Insufficiency, the user should modify the initial thickness, using the command Edit and set in place the **Suggested min thick. hs (mm)***

### 2.7 Properties

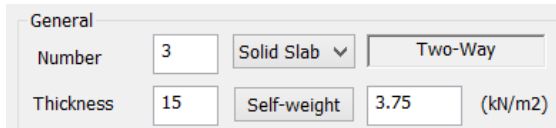
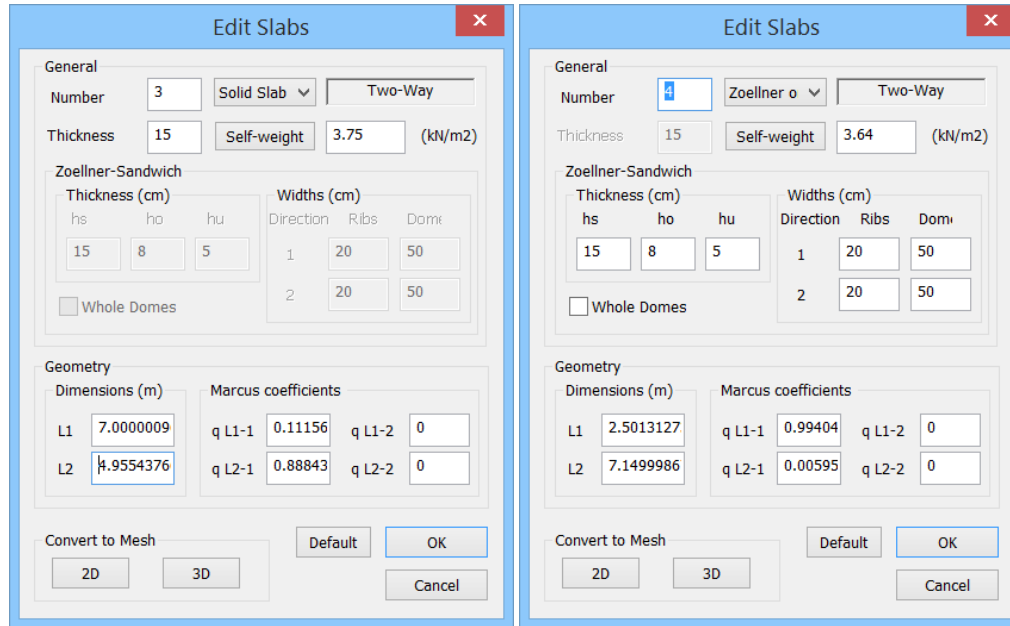
With this command you can see and edit the details of the slabs that have been identified and modeled, and modify the type of the slab:

- . Solid to Zoellner
- . Zoellner to Solid
- . Solid to Waffle



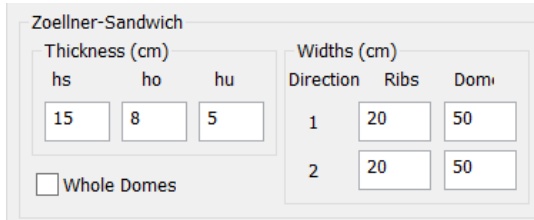
Select the command and click inside a slab. Then, the following dialog box is displayed:



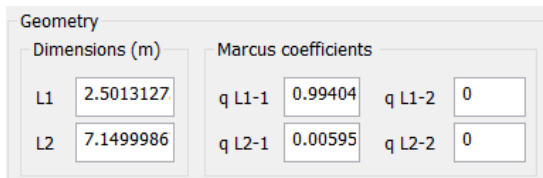


The first section "General" contains information about the slab, like the numbering, the thickness, the self weight and the type (Solid or Zoellner).

**!** You can convert a Zoellner slab to a solid slab, but you can't change the type (i.e. Two Ways). It is a read only field.



The “Zoellner-Sandwich” section is activated for Zoellner slabs and contains information about the total, upside and down side slab’s thickness, ribs and domes widths, which are editable.



“Geometry”: This field contains the dimensions L1 and L2 as well as the load distribution coefficients qL1-1 and qL2-1 (Marcus coefficient).

**!** The fields “qL1-2” and “qL2-2” is related only to the three way slabs. In all other cases are 0.

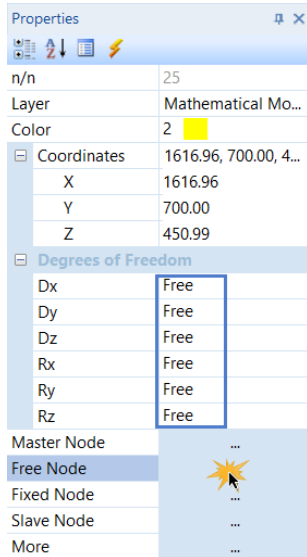
**!** Also, you can convert a slab to a 2D and 3D mesh. Select the "2D" or "3D" and define the mesh.





**EXAMPLE**

**Slab conversion using surface finite elements**



Start by setting free the nodes of the perimeter of the slab, since they probably belong to a diaphragm.

You can do this for each node separately concerning its data, by clicking the button next to "Free Node".

Otherwise, use the Multiselect Edit command



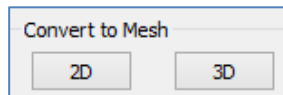
and select all the nodes.

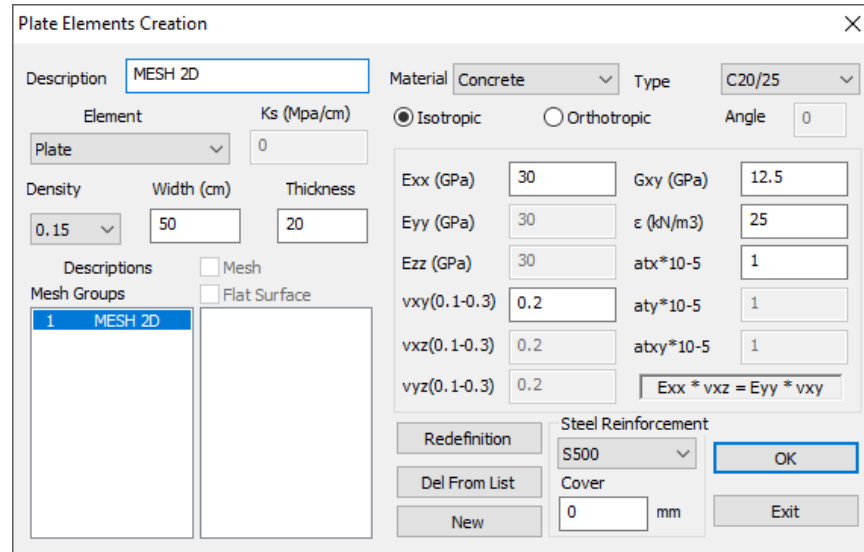


Select the command “Properties” and left click inside the slab:



Select “Convert to Mesh” and then click “OK”.

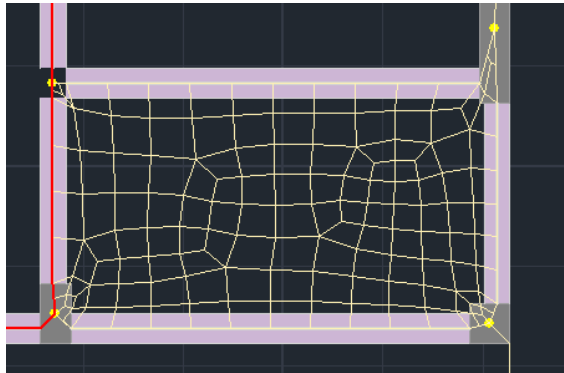




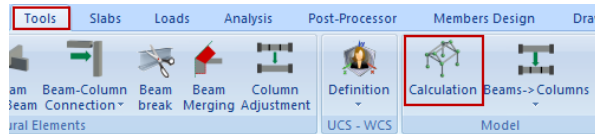
In the dialog box, define the following parameters:

- Description, Element, Thickness equal to the thickness of the slab
- The density that depends on the form of the slab (usually a value around 0.15 is sufficient).

Also, if it is necessary, change the type of the concrete. Select “New” and then “OK”.



- The program creates the mesh of the surface finite elements.



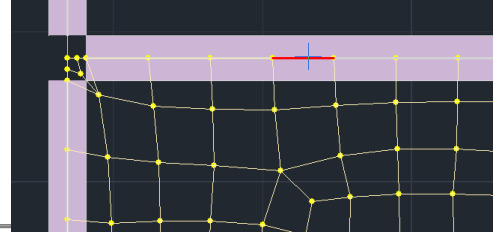
- Then you need to calculate the model.

- The last step is the connection of the beams members with the surface elements by using

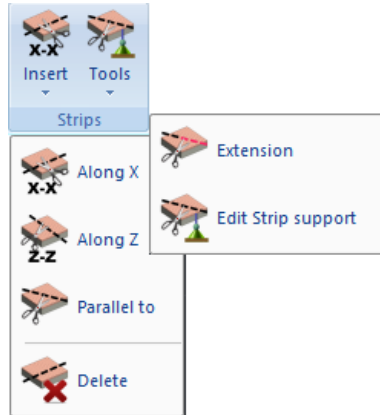
“Tools > Members> Beam-Plate Connection”



**!** Zoom to see that the beams members are broken and connected with the surface elements.



### 3. Strips



Strips' input is essential to the analysis and design slabs' steel reinforcement. They are the "guides" for the design of the steel reinforcement and the diagrams display. Depending on the type and form of the slab, one or more strips are required in one or more directions.

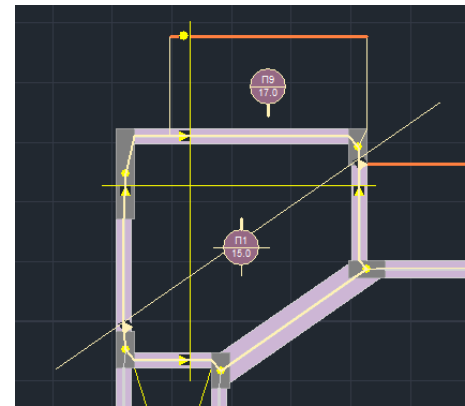
#### 3.1 Insert



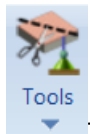
Select "x Strip" and "z Strip" and enter the slab strips from side to side, parallel to the global X-X and Z-Z axes, respectively.

The command "Parallel to" is used to insert strips parallel to a Line that you have previously selected.

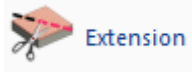
**!** To delete one or more strips, select the command "Strip Delete" and left click on the strip line that you want to delete.



#### 3.2 Tools



This command contains a list of sub-commands for strips editing; to extend strips or change support conditions.



**Extension**

: select the edge of the strip you want to expand and define the new position.



**Edit Strip support**

: select the command and left click on the support to modify the support conditions. Click on the support conditions' symbol to change them sequentially.

⚠ The symbols used by the program are:

- : free edge
- ▲ : simple support
- : fixed

⚠ The change of the support type is achieved by left clicking on the support. The change from one type to another is done sequentially.

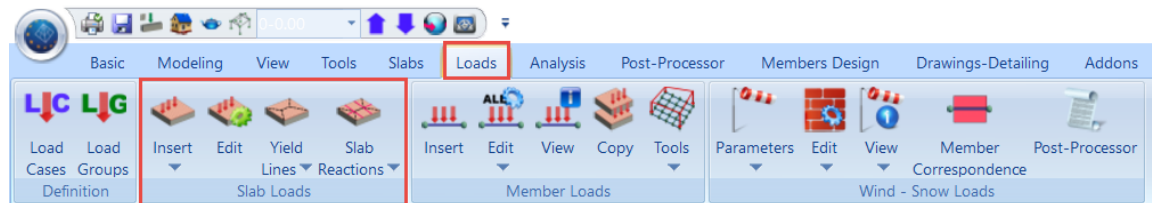
#### 4. Checks



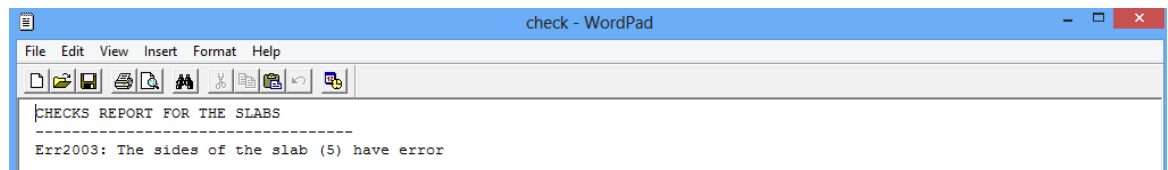
**Checks**

Checks command should always be performed to avoid possible errors.

⚠ To avoid alerts related to the slabs reactions, it is best first to attribute loads of the slabs, in the next section "Loads" >> "Loads Slab" and then to run the Checks command.



Select the command “Checks” and a TXT file opens, which contains the results of the slab’s design checks for all slabs of the current level.



⚠ If the file is empty there are no errors. Otherwise, correct all the errors related to the slabs before you proceed to the "Loads".