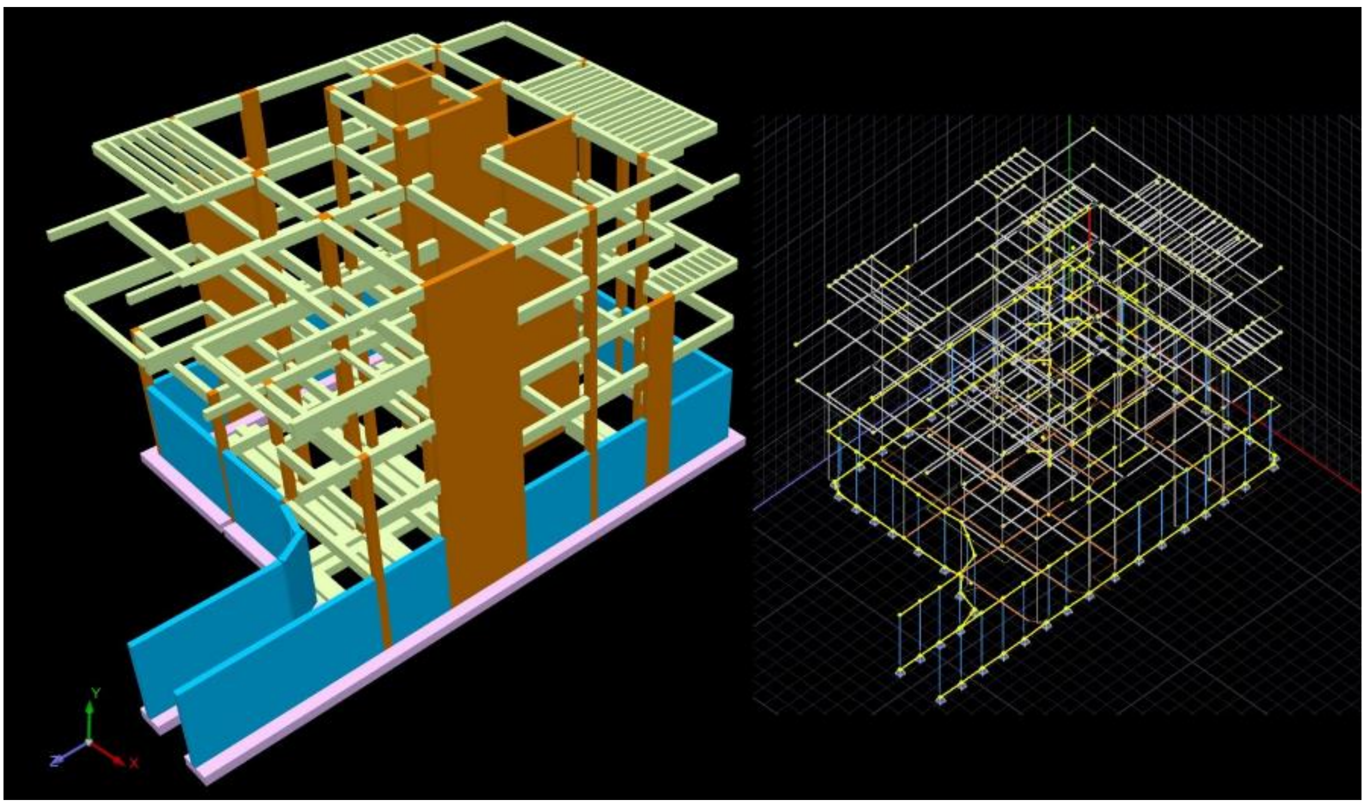
SCADA Pro™

Structural Analysis & Design

User's Manual

2.MODELING

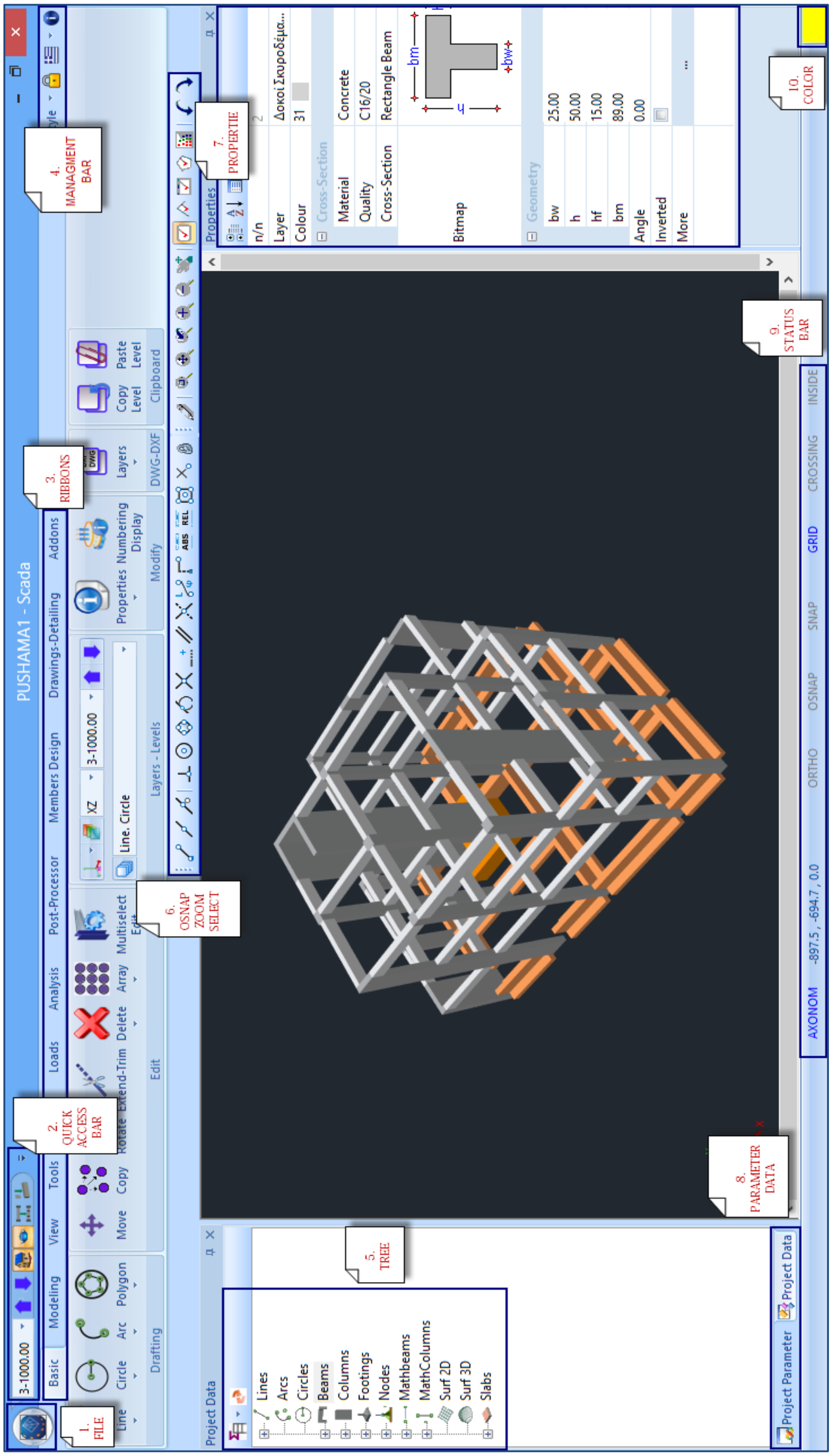


ACE HELLAS

INTEGRATED SOLUTIONS

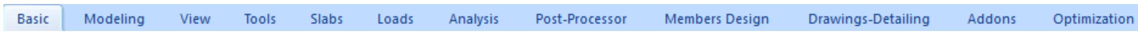
Contents

I. THE NEW UPGRADED INTERFACE of SCADA Pro

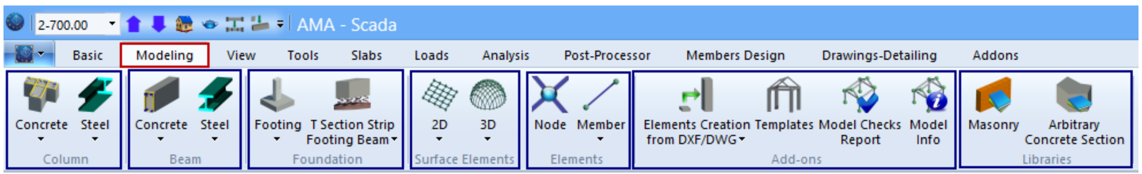


II. DETAILED DESCRIPTION OF THE NEW INTERFACE

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



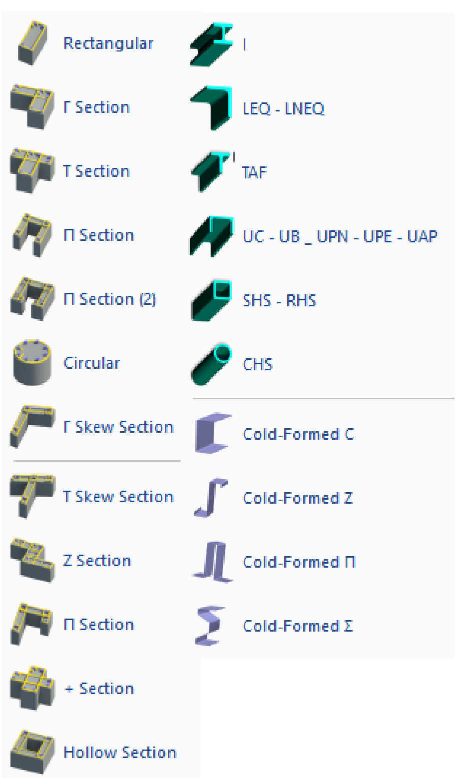
MODELING

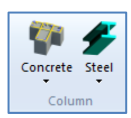


The 2nd Unit entitled "Modeling" includes the following seven command groups:

1. **Column**
2. **Beam**
3. **Foundation**
4. **Surface elements**
5. **Elements**
6. **Add-ons**
7. **Libraries**

1. Columns





The "Column" group command includes commands for modeling columns with the following cross- section types of material:

Concrete

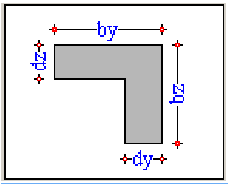
Steel (Cold formed & Cold Rolled)

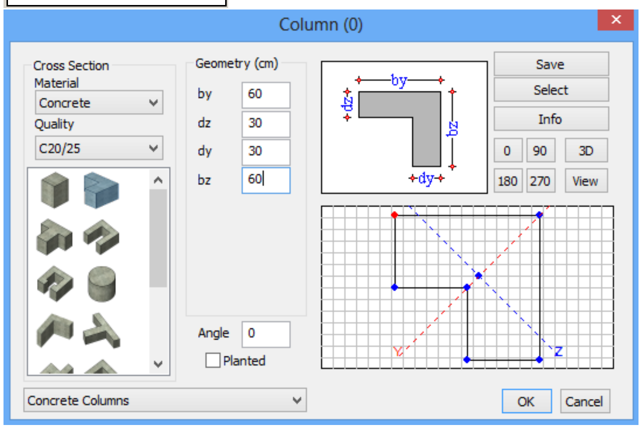
Timber

Each one contains relative section commands.

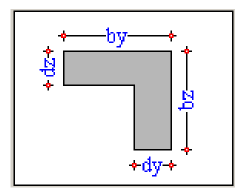
Define columns' **parameters**. More specifically:

1.1 Concrete columns



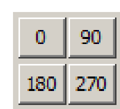


"**Cross Section**": select the material (Concrete /Steel) and then the type. Choose a standard section from all the sections that appear, depending on the type of the material.

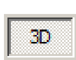


"**Geometry**": type the dimensions of the section according to the figure on the right.

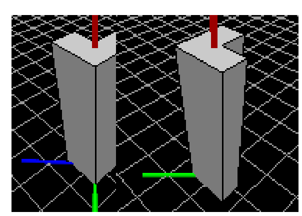
The four buttons under the figure define the placement angle in degrees. The column can be rotated by 90, 180 or 270 degrees.

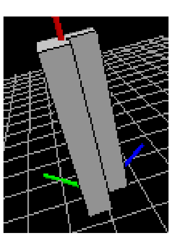


"**Angle**": type the angle's value manually.



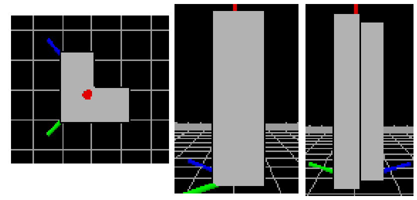
Button: offers a 3D representation of the column cross-section. Use the degrees buttons to rotate column.



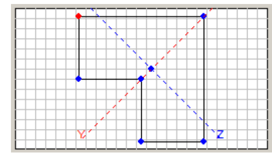


Press and hold the left mouse button and move the mouse to rotate the column along the respective axis.

Press the button  to see all column views.

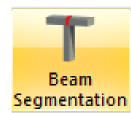


In the window below, you can see the formed section according to the geometry and the dimensions selected. You can also see the local axes yy and zz and you can change the insertion point (red point) by choosing a blue one.

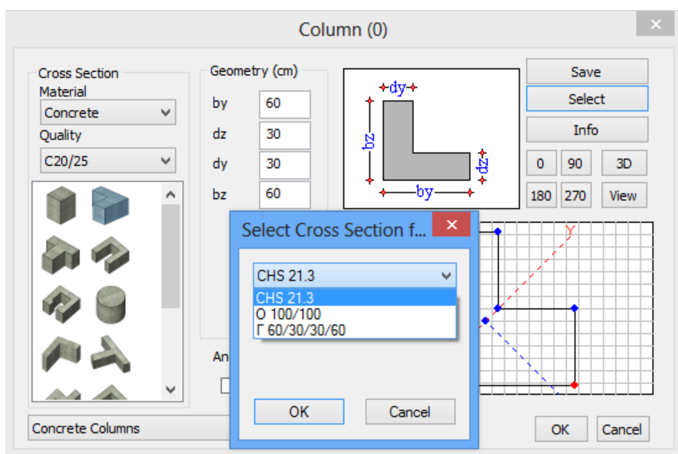


“**Planted**”: This command is used to insert a planted column. Activate the following checkbox  and place the section in the current level, while at the next upper level insert the same section <u>without</u> activating the checkbox "Planted".

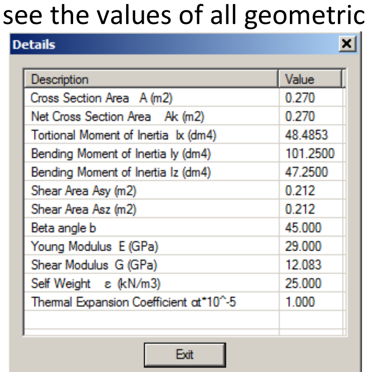
Remember to connect the planted column to the corresponding beam: Select "**Tools** >> **Beam Segmentation** " and then left click on the beam.



"**Save/Select**": This command is used for the creation of a section’s library through the command "**Save**", related to the user’s need. The user can open it, at any time, through the command "**Select**", without the need of defining the same sections again.



“**Info**”: This command is used to see the values of all geometric and inertial data of this section.



Details

Description	Value
Cross Section Area A (m2)	0.270
Net Cross Section Area Ak (m2)	0.270
Tortional Moment of Inertia Ix (dm4)	48.4853
Bending Moment of Inertia Iy (dm4)	101.2500
Bending Moment of Inertia Iz (dm4)	47.2500
Shear Area Asy (m2)	0.212
Shear Area Asz (m2)	0.212
Beta angle b	45.000
Young Modulus E (GPa)	29.000
Shear Modulus G (GPa)	12.083
Self Weight ε (kN/m3)	25.000
Thermal Expansion Coefficient αt*10^-5	1.000

Exit

Concrete section’s list includes standard, parametric and arbitrary sections.

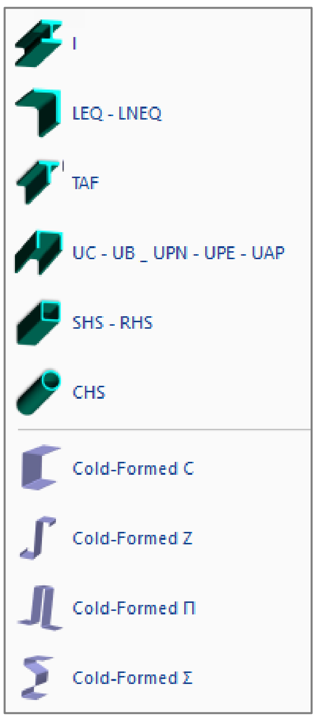
- ⚠ *To define an arbitrary concrete section use the command “Modeling>>Libraries>>Arbitrary Concrete Section”,* described in detail in the corresponding chapter.

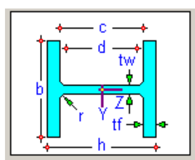
- ⚠ *L & T Parametric sections are designed automatically.*

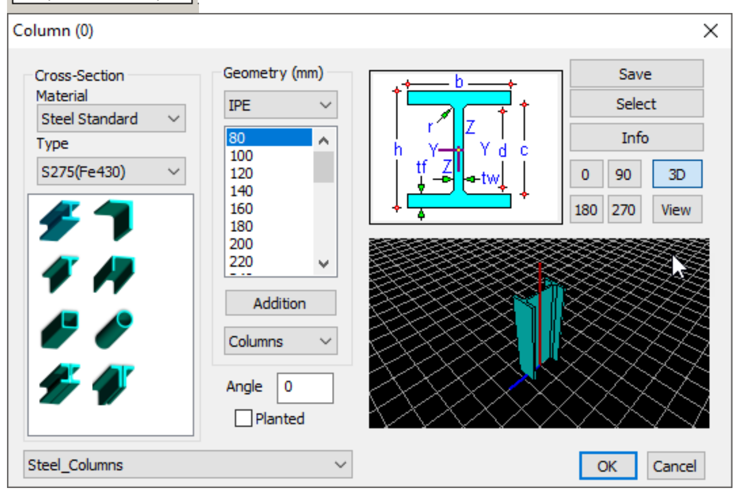
- ⚠ *All the other parametric sections cannot be* designed automatically, *but if the user set reinforcement in the section, the program can check the section’s resistance.*

1.2 Steel columns

For steel sections, set the material type and the angle. Geometry includes the number of cross sections both Hot Formed and Cold Rolled.

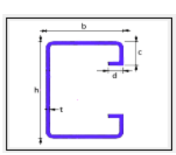


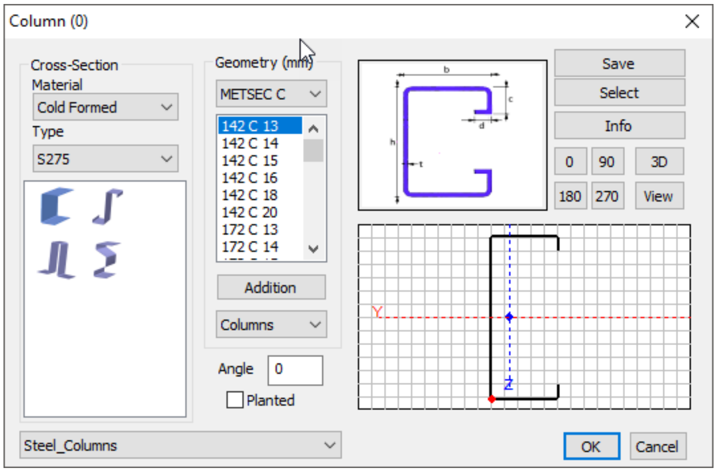




The three-dimensional display helps to select the right angle versus the local axes, and the "Save" button to create sections’ library.

The insertion of the cold rolled elements is similar to the hot rolled. For both beams and columns, cross sections have been placed in a new category called "Cold Rolled".



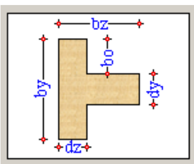


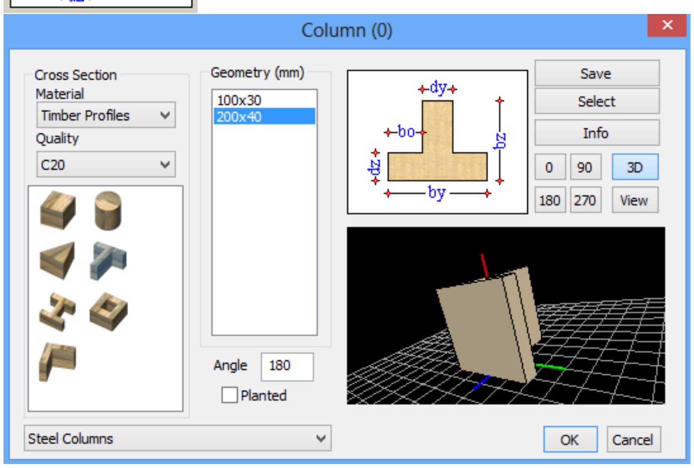
In the dialog box choose:

- **Cross-Section**
- **Material**
- **Type**
- **Geometry**

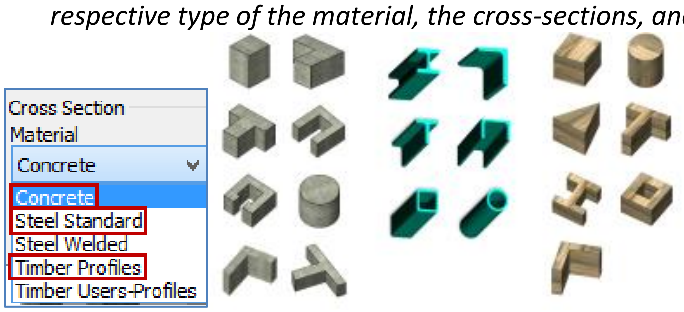
1.3 Timber columns

SCADA Pro contains timber sections inside columns’ dialog box.





⚠ *In the dialog box, all the sections are included. According to the selected material the respective type of the material, the cross-sections, and the geometry are displayed.*



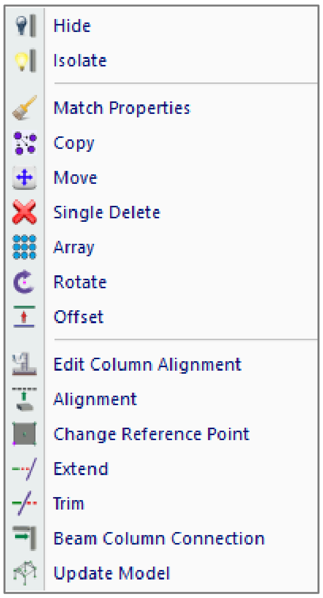
⚠ *You can model Timber sections by following the same procedure, described for the concrete and steel sections.*

⚠ *In SCADA Pro, you can model and analyze concrete, steel, timber and masonry structures.*

MOUSE RIGHT BUTTON

In each “tab” of the program and for each item you approach with your mouse, by selecting the right button a list of commands related to the “tab” and the item opens.

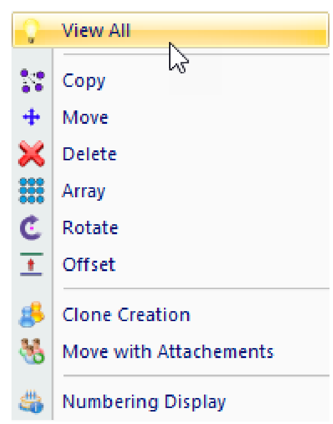
Move the mouse over the section of an inserted column and press the right botton:



The list includes commands in the corresponding “tab”, related to the element that is right-clicked.
However, there are some commands that are only here, like:
“**Hide**”: to hide an item.
“**Isolate**”: to isolate an item, hiding all the other ones.

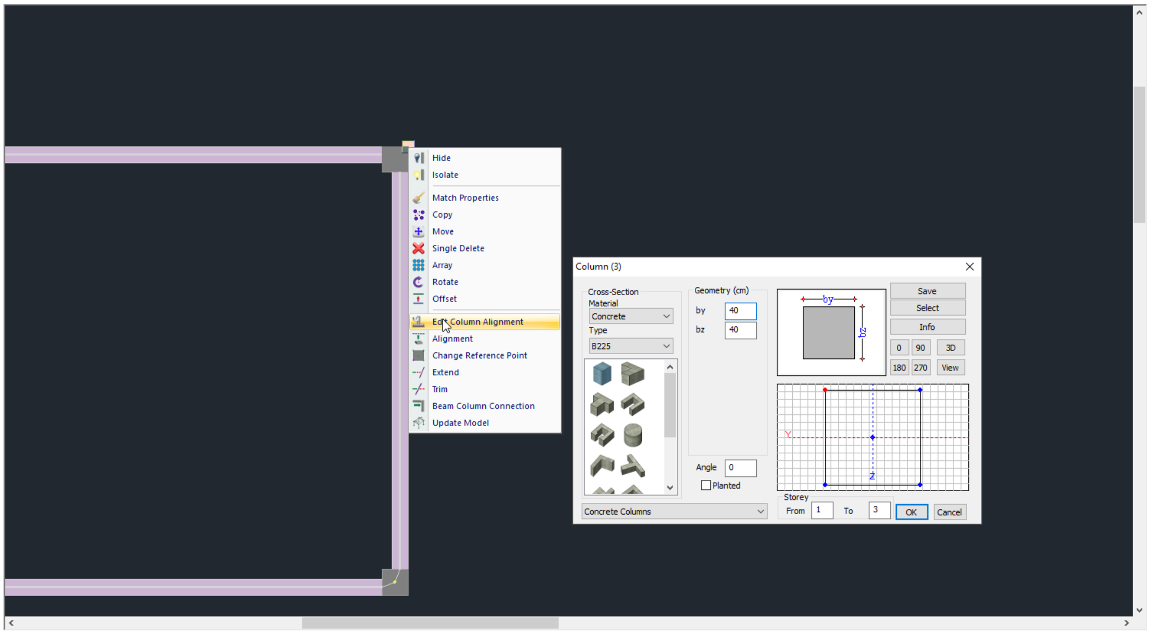
To deselect, press the right mouse button at any point on the surface of the grid.

A new list opens and you select "View All".

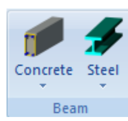


“**Edit column alignment**”: to modify the cross-section of a column in height.
Approach the end or side of the cross section of the column to the lowest level about which will be modifiedy, right-click and “Edit column alignment”.
The relative sections window opens, where you can modify the geometry and select the storeys in which the modification will be applied.

Choose the cross section to the lowest level, and in the window "blush" the correct reference point.



2. Beams

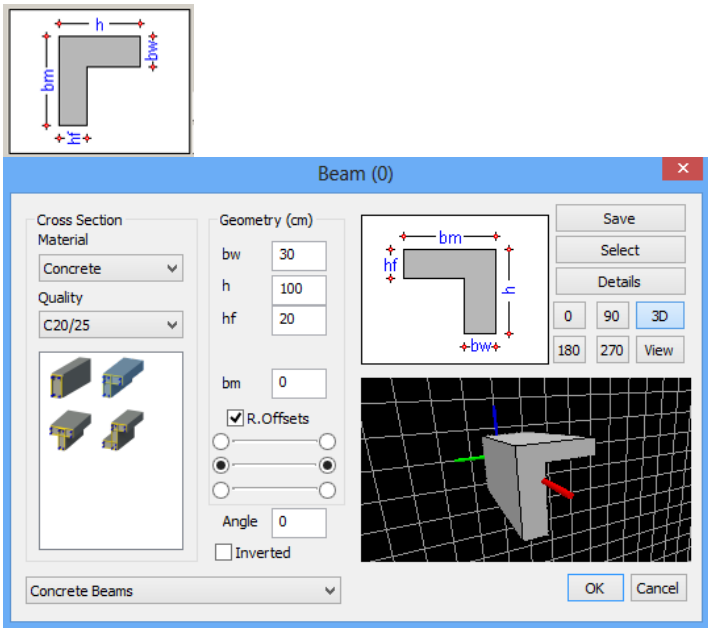


The command group “Beams” includes cross – sections of :

Concrete
Steel (Cold formed & Cold Rolled)
Timber

Each one contains relative section commands which define the type and the shape of the cross section.

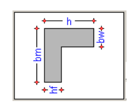
2.1 Concrete beams

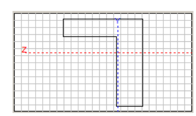


Define beams’ **parameters**. More specifically:

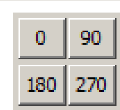
“**Cross Section**”: select the material (Concrete /Steel) and the type. Choose one standard section from all the appearing sections, depending on the type of the material.

“**Geometry**”: type the section’s dimensions according to the figure on the right.

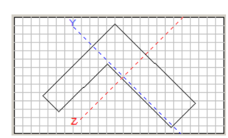




In the window, you can see the section formed according to the geometry and the selected dimensions. You can see also the local axes yy and zz.



The four buttons under the figure define the placement angle in degrees. The beam can be rotated by 90, 180 or 270 degrees.

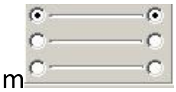


“**Angle**”: type the angles’ value, manually (The angle refers to the xx local axis of the beam) (angle Beta).
Example: Rotated beam45º in :

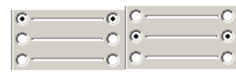
The button”**3D**” 

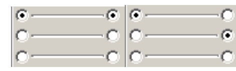
provides a three-dimensional representation of the column section. Use the degrees’ buttons to rotate the beam.

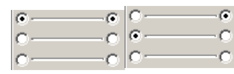
When the checkbox “R.Offsets” is active, the beam will be modeled with rigid offsets, otherwise, it will be modeled without them.

Select the insertion axis for the beam .

There are three ways to place the beam (according to the insertion axis), which can also be set with the keypad.

Use TAB to change the start and end alignment 

Use SHIFT to change start beam alignment 

Use CTRL to change end beam alignment 

Select the section of the beam and the insertion point (start point) with left click on the desktop. Before you select the end point of the beam press TAB, SHIFT, or CTRL, depending on how you want to place it.

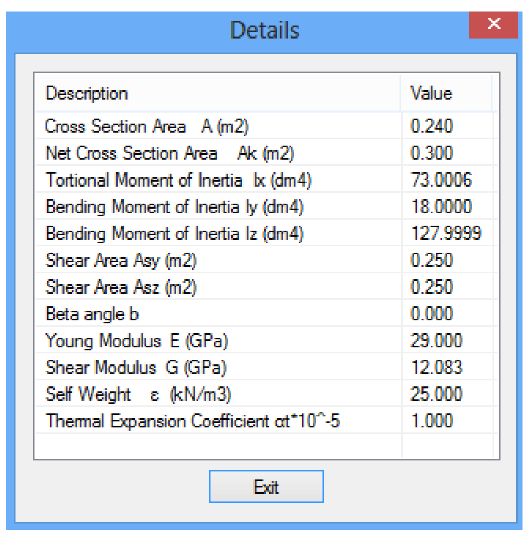
Activate the checkbox “**Inverted**” to insert an inverted beam.

From layers list 

select the layer where the beam will belong. The default layer is the layer “Concrete Beams”.

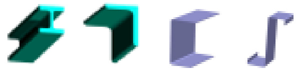
"**Save/Select**": User can create its own sections library through the command "Save". The user can open it, at any time, through the command "Select", without the need of defining the same section every time.

“**Details**”/: to see the description of all geometrical and inertial data of the section

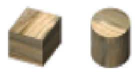


Description	Value
Cross Section Area A (m2)	0.240
Net Cross Section Area Ak (m2)	0.300
Tortional Moment of Inertia Ix (dm4)	73.0006
Bending Moment of Inertia Iy (dm4)	18.0000
Bending Moment of Inertia Iz (dm4)	127.9999
Shear Area Asy (m2)	0.250
Shear Area Asz (m2)	0.250
Beta angle b	0.000
Young Modulus E (GPa)	29.000
Shear Modulus G (GPa)	12.083
Self Weight ε (kN/m3)	25.000
Thermal Expansion Coefficient αt*10^-5	1.000

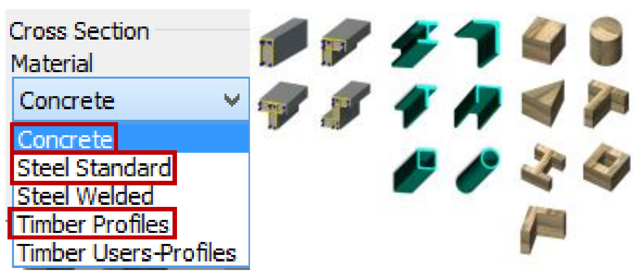
2.2 Steel beams


For steel sections (Hot & Cold Rolled) set the material type and angle with respect to the concrete section. The three-dimensional view helps to select the right angle versus the local axes, and the "Save" button to create sections’ library.

2.3 Timber beams


SCADA Pro contains timber sections inside beams’ dialog box.

⚠ *In the dialog box, all sections are included. Based on the material you select, the qualities, cross sections and geometry are adjusted.*

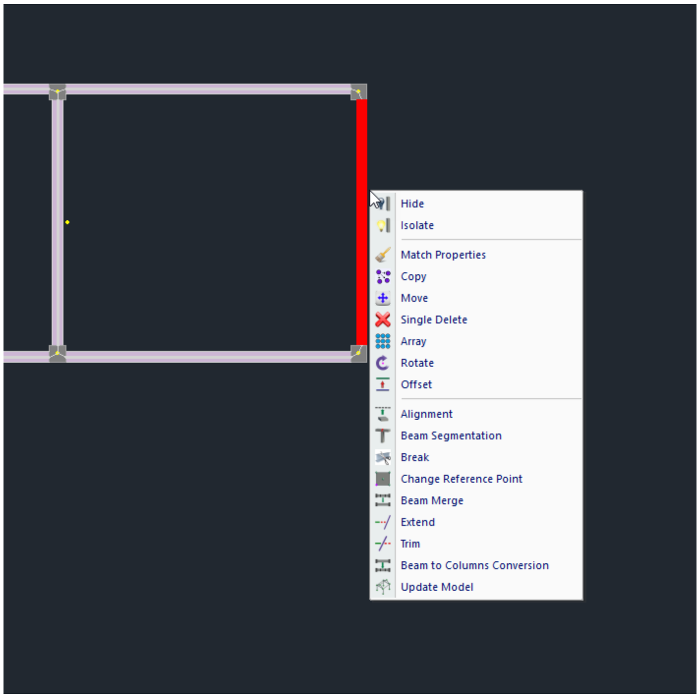


⚠ *You can model Timber sections following the same procedure described for the concrete and steel sections.*

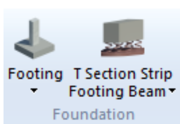
⚠ *In SCADA Pro, you can model and analyze concrete steel, timber and masonry structures.*

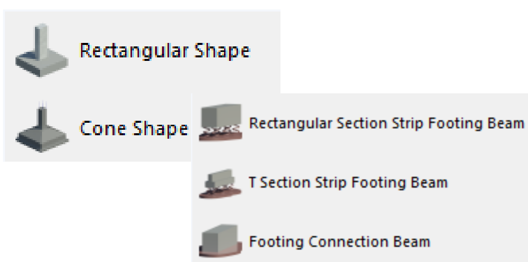
MOUSE RIGHT BUTTON


The list includes commands in the corresponding tab, related to the beam.



3. Foundation





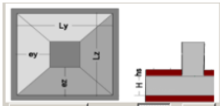
The command group "Foundation" includes:

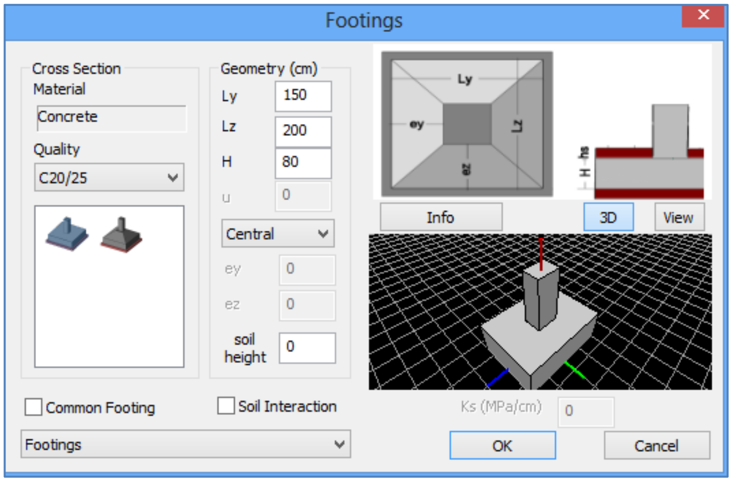
Footing and
T Section Strip Footing Beam

Each one contains relative section commands which define the type and the shape of the cross section.

Define foundation's parameters. More specifically:

3.1 Footings

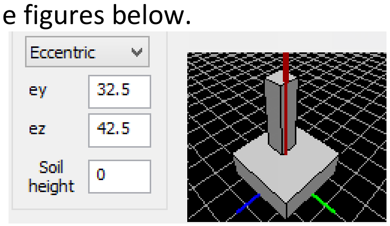




Define the footing **parameters**. More specifically:
"**Cross Section**": select the type of the material and the shape of the section.
"**Geometry**": type the section's dimensions according to the figure on the right side of the dialog box.

Select from the following drop-down list 

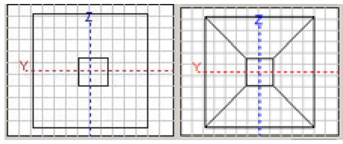
the footing's position in the corresponding column. For eccentric footing, enter the corresponding eccentricities according to the figures below.



The button "**3D**" 

: it provides a three-dimensional representation of the footing section. Use the degrees buttons to rotate the footing.
In "**Soil height**" type the value you wish. The soil height is calculated beginning from the base of the foundation up to the ground level.
For "**Cone Shape Footing**", the only difference is that you must define the plate thickness *u* of the foundation (equal to H/3).

⚠ *Prerequisite for the footing's placement is the existence of the columns.*
Activate the checkbox "**Common Footing**" to place the same footing under two or more columns.

In the following figure,  you can see the section formed according to the selected geometry and dimensions. You can also see the local axes yy and zz.

For elastic supports, check the "**Soil Interaction**" 

and type a value for the Ks constant.

From the layers' drop-down list 

select the layer, to which the column will belong. The automatic default layer is "Footings".

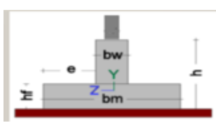
How to place a footing:

Define the footing geometry and select the corresponding column, indicating one of the tops, or one of its sides (to place the footing parallel to it).

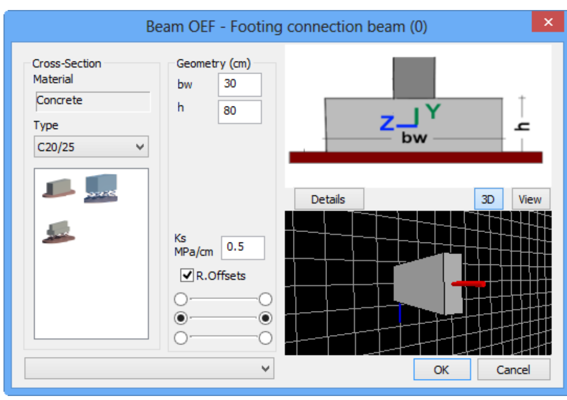
How to place a common footing:

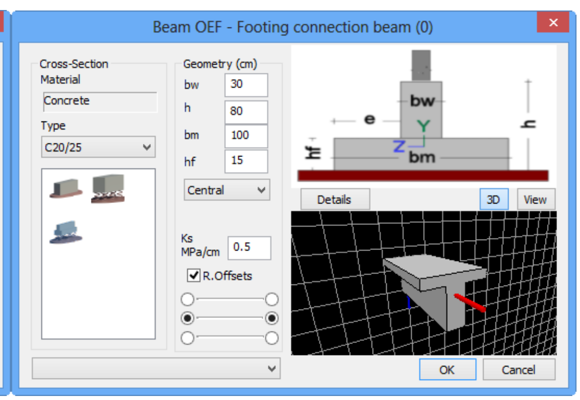
Define the geometry of the footing and then select the columns that will be placed successively.

3.2 Strip footing beam:

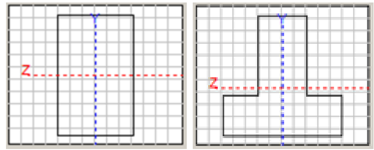


Select the section and the following dialog box is displayed:





Define strip footing beam's **parameters**. More specifically:
"**Cross Section**": select the type of the material and the shape of the section.
"**Geometry**": type the section's dimensions according to the figure on the right.

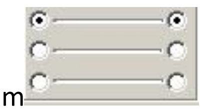
In the following figure,  you can see the section formed according to the selected geometry and dimensions. You can also see the local axes yy and zz.

Button “**3D**” 

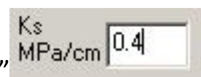
: it provides a three-dimensional representation of the beam section. Use the degrees buttons to rotate the beam.

When the checkbox “**R.Offsets**” is active, the beam will be modeled with rigid offsets, otherwise, it will be modeled without them.

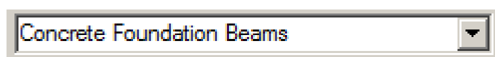
⚠ *To place the strip footing beam under the wall basement, you have to deactivate the checkbox “**R.Offsets**” and “**Autotrim**”.*

Select the insertion axis of the beam .

⚠ *To place the foundation beams follow the same procedure as for beams.*

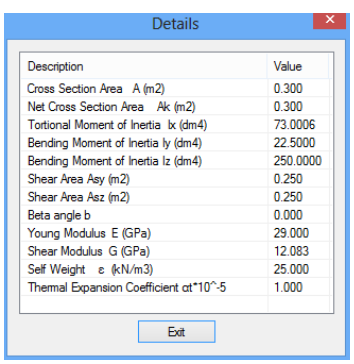
In the field “**Ks**” 

, type the value of the Ks soil constant.

From the layers’ drop-down list 

select the layer, to which the beam will belong. The layer “Concrete Foundation Beams” is a default layer.

“**Details**”/: to see the description of all geometrical and inertial data of the section


Details

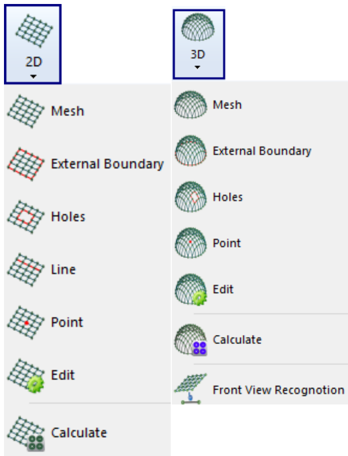
Description	Value
Cross Section Area A (m2)	0.300
Net Cross Section Area Ak (m2)	0.300
Tortional Moment of Inertia Ix (dm4)	73.0006
Bending Moment of Inertia Iy (dm4)	22.5000
Bending Moment of Inertia Iz (dm4)	250.0000
Shear Area Asy (m2)	0.250
Shear Area Asz (m2)	0.250
Beta angle b	0.000
Young Modulus E (GPa)	29.000
Shear Modulus G (GPa)	12.083
Self Weight ε (kN/m3)	25.000
Thermal Expansion Coefficient αt*10^-5	1.000

Exit

3.3 Footing connection beam

The insertion procedure is similar to the strip footings, without the participation of the soil elements.

4. Surface Elements



The command group “Surface Elements” includes the commands for modeling:

- **2D Surface Elements** and
- **3D Surface Elements**

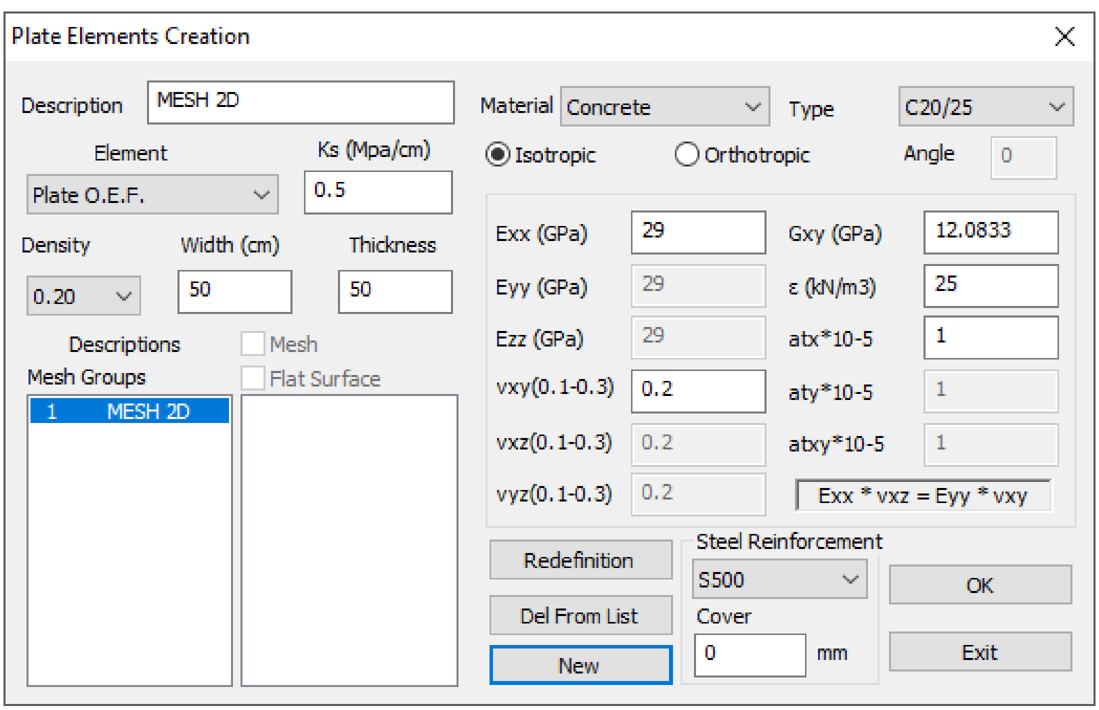
Each command contains sub-commands to define, describe, and generate the mesh.

4.1 Surface elements 2D

You can model foundation slabs and any other type of horizontal surfaces with 2D surface finite elements.

4.1.1 Mesh

Select “**Mesh**” and the following dialog box appears which displays the field-parameters:



Specify the **properties** of the mesh generation:
“**Description**”: type a name to describe the mesh group
“**Element**”: select the type of the plate. If you choose “Plate (O)n (E)lastic (F)oundation” you have to type a value for the soil constant “Ks” in the relative box. This choice is appropriate for raft foundation, while the option "Plate" in all other cases.

“**Density**”, “**Width**” and “**Thickness**”: displays the geometry of the mesh.

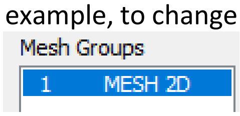
- "**Density**": the value expresses a smooth transition from an area with dense mesh elements in an area with sparse. High-density expresses smoother "flow" of mesh elements and of course more elements. Low density can be used when you want to use a few elements to estimate the stresses in a region (ex. preliminary design).
- “**Width**”: type the width of each mesh element
- “**Thickness**”: type the thickness of the plate.

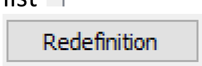
The “**Mesh**” and “**Flat Mesh**” selections four are deactivated and they will be active only in mesh with 3D finite elements.

Select “**Material**” and “**Type**” and set the behavior of the material “**Isotropic or Orthotropic**”.
The orthotropic material allows you to assign different material properties in each direction. An orthotropic material satisfies the following equality with respect to the modulus of elasticity $Exx * vyx = Eyy * vxy$.
“**Angle**”: for an orthotropic material, it will be activated in next versions.

“**New**”: complete the data and click “New”. In the list “Mesh” the mesh you have just created will appear, with a serial number and the name that you specified. Follow the same procedure to create another mesh with different geometrical and physical properties.

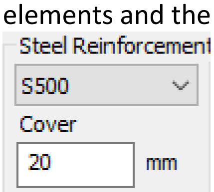
“**Redefinition**”: this command is used to modify the data of a generated mesh.

EXAMPLE:
For example, to change the Thickness of the “MESH 2D” from 50 to 60 cm, select it first from the list 

and then type the new value in the corresponding box. Then, click on 

and the thickness will be updated. Similarly, you can change any other geometrical or physical characteristic of the mesh group.

Steel reinforcement and Cover
It is the field where you select the quality of the **steel reinforcement** for the surface finite elements and the mm of the **cover**:



“**Delete**”: this command is used to delete one or more generated mesh groups. Select a mesh from the list and click “Delete”.

⚠ *The mesh will not disappear, but you will see the word “Delete” next to the name. This means that it has been deleted, but you can recover it if you click “Delete” again. Then, the word "Delete" disappears and the mesh becomes active again.*
To delete a mesh permanently, after using the “Delete” command, save the project by selecting the command “Save” Save.

4.1.2 External Boundary:

External Boundary this command is used to define the external boundary of the Mesh.

⚠ *If there is no mesh group created, the dialog box “Plate elements creation” opens to define a mesh group. If you have already created it, perform the following steps.*

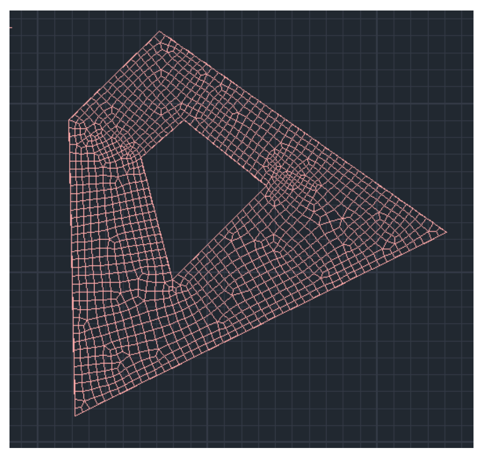
Select the command and left click to the points that will be the vertices or the external boundary. Use snap tools for help. To complete the command, use right click and close the border by using the first point as the last one.

⚠ *To delete an external boundary, you have to delete the corresponding mesh group.*

4.1.3 Holes:

Holes this command is used to define the mesh’s gaps/holes, in case there are gaps/holes in the mesh surface. Select the command and define the perimeter of the hole as previously.

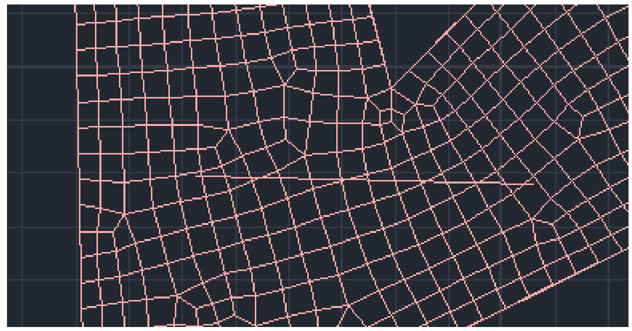
⚠ *The definition of the holes can also be done after the creation of the mesh surface by using the command “Calculate” that regenerates the mesh which now includes the gaps/holes.*



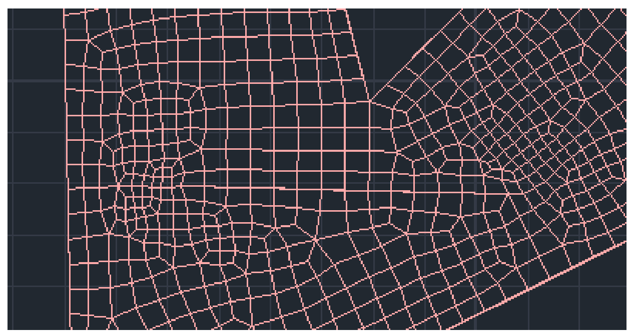
4.1.4 Line:

this command is used to define the region or regions of the surface with lines, where an adaptive mesh will be generated. Select the command and design one or more lines inside the defined boundaries.

The lines’ definition can also be done after the creation of the mesh surface by using the “Calculate” a posteriori since you have created the mesh surface by using the command “Calculate” that regenerates the mesh according to the line.

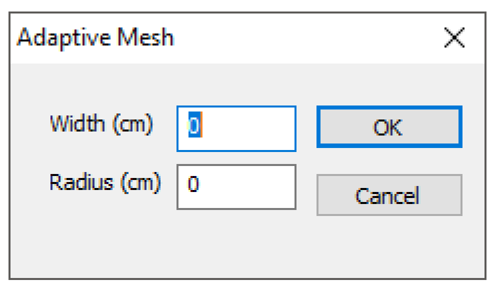


This leads to a new configuration according to the line densification.



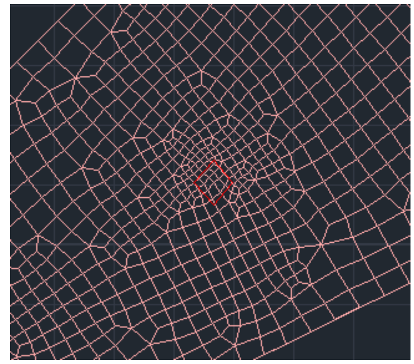
4.1.5 Point:

this command is used to define points inside the boundary, which will be the points of thickening of the meshing.



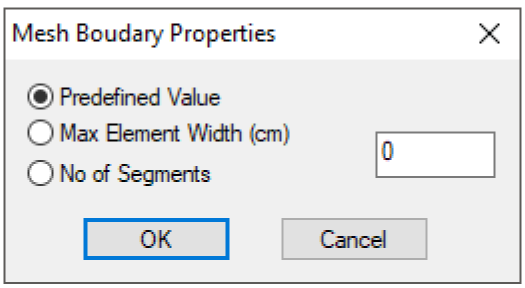
Select the command and define the area of thickening around the point. Click one or more points in the boundary.
Select "*Calculate*" and you get the thickening.

The point definition can also be done a posteriori, since you have created the mesh surface, using the “Calculate” command, that recalculates the meshing based on the point too.

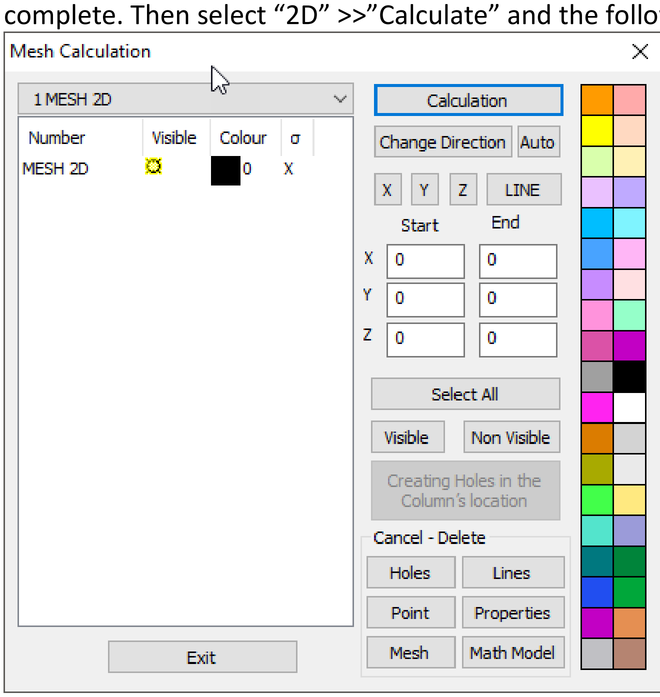


4.1.6 Edit:

this command is used to edit a mesh that you have already created.
Edit must be after mesh calculation and before the mathematical model creation.
Since you select the command, the following dialog box is displayed:



“**<u>Predefined Value</u>**”: This checkbox is activated to define the number of the elements for the total mesh. Type the number, click “OK” and left click on the sides of the boundaries. Right click to complete. Then select “2D” >>”Calculate” and the following dialog box opens:

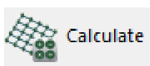


Select a mesh and click "Calculation". The number of the elements of the mesh will change and the mesh will be updated based on the predefined value.

"**<u>Max Element Width</u>**": This command is used to define the maximum width of a single mesh element. Type the value of the maximum width of a single mesh element. Click "OK" and left click on one or more sides of the boundary, where mesh elements will have the maximum width. Right click to complete. Then select "2D" >>"Calculate" and in the dialog box, select the mesh and click "Calculation". The mesh will be updated considering the maximum width of the single mesh element.

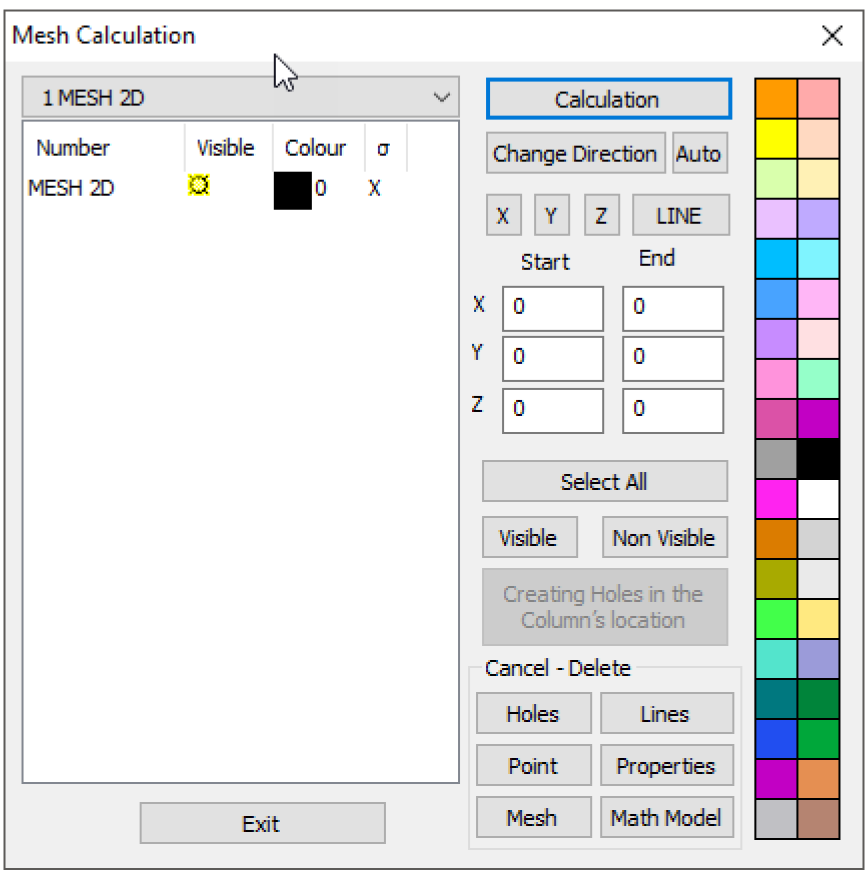
"**<u>No of Segments</u>**": This command is used to define the number of segments (not of the total mesh elements). Click "OK" and left click on one or more sides of the boundary, to apply the segmentation. Right click to complete. Then select "2D" >>"Calculate" and in the dialog box, select the mesh and click "Calculation". The mesh will be updated considering the number of the segments.

4.1.7 Calculate:



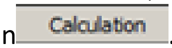
this command is used to generate a 2D mesh, based on the defined external boundary, lines and points.

When you select the command, the following dialog box is displayed:



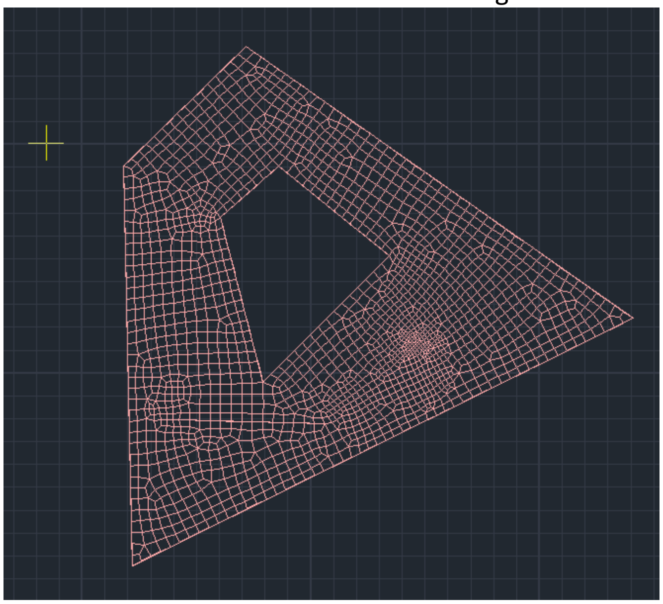
From the top list of the dialog box, choose the mesh group you want to generate. Note that each boundary must have its mesh group. This means that you cannot use "MESH 2D" in two different areas.

In order to generate a mesh, first, select the mesh group from the list, or click on 

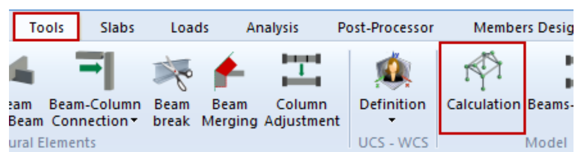
and then click on 

. The same process is applied to different mesh groups.

The simulation results are shown in the figure below:

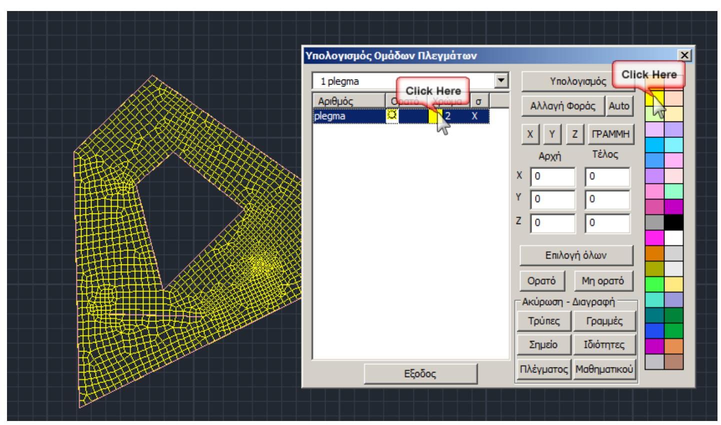


⚠ *This command generates the mesh but not its mathematical model. For this purpose use “**Tool**” >> ”**Calculation**”.*



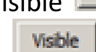
The same dialog box, in addition to the generation of the mesh, includes more useful tools:

To change the color of the plate, select the plate from the list and select the desired color from the palette of colors.



Use the command “Visible” 

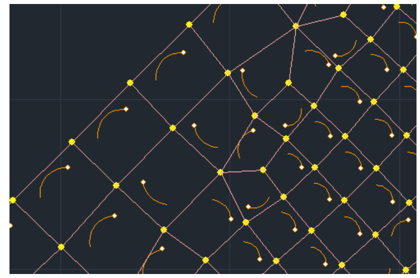
or “Non Visible” 

, to show or hide the selected mesh. The indicator under the column “Visible” 

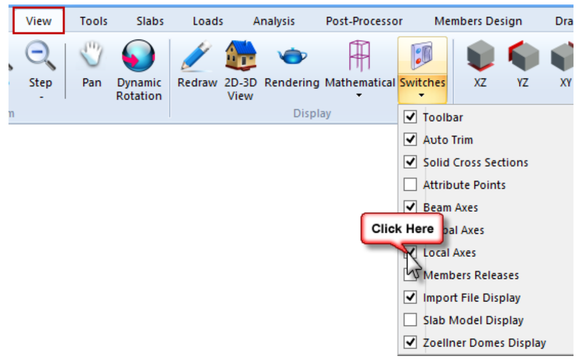
changes from visible ( ) to invisible ( ). You can apply this change only after the generation of the mathematical model.

Use the command “Change Direction” 

to change the direction of the local axes of the surface finite elements.

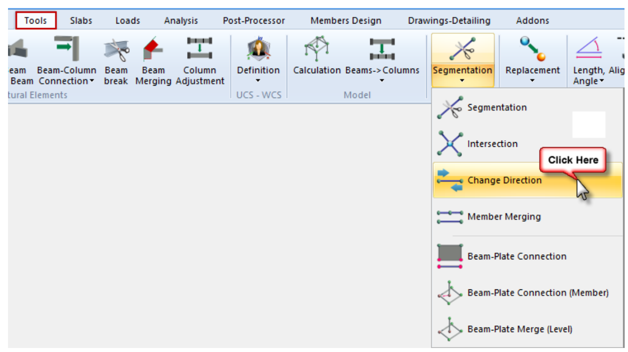


⚠ *Local axes, which are acivated through, the unit "**View**" >> "**Switches**" >> "**Local Axes**", are displayed on the elements after the creation of the mathematical model.*

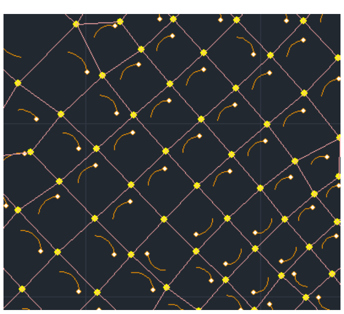


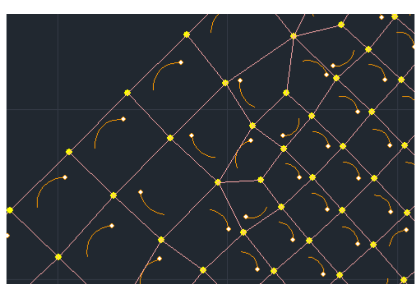
The command “Change direction” is used to change the direction of all surface elements of the selected plate.

⚠ *If you want to change the direction of some surface elements, use the command “Change direction” through the unit "**Tools**" >> "**Segmentation**" >> “**Change direction**”.*

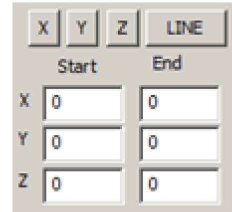


With the command “**Auto**” the program adjusts local axes of all the surface finite elements of the plate to have the same direction.





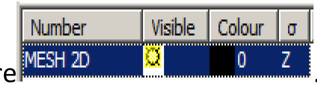
Use the left field to define the main direction of the steel reinforcement (direction X, Y, Z).



EXAMPLE:

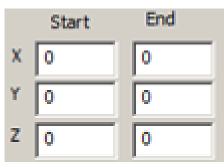
Select a mesh from the list and if you choose the button "X" you will see the sign "X" under the column ''σ'',

, but if you choose the button "Z" you will get the following picture

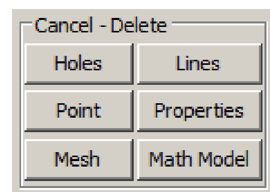


.

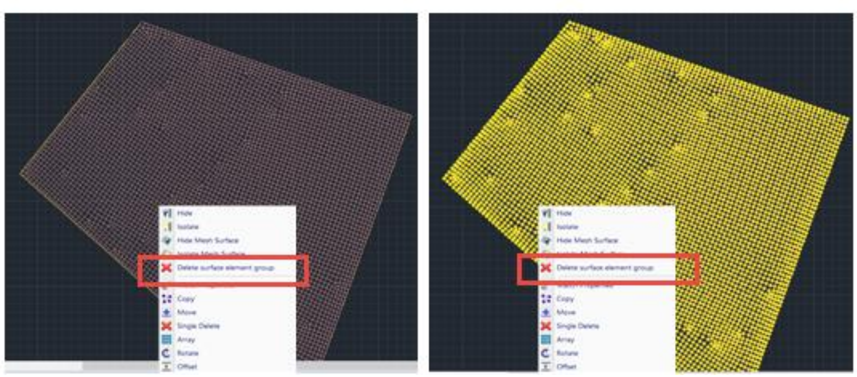
Use the command "LINE" combined with the values in the following fields, to define the coordinates of the line and then, the program will consider the line direction as the main direction of the steel reinforcement. Use this command in case that the mesh is not parallel to the main axes.

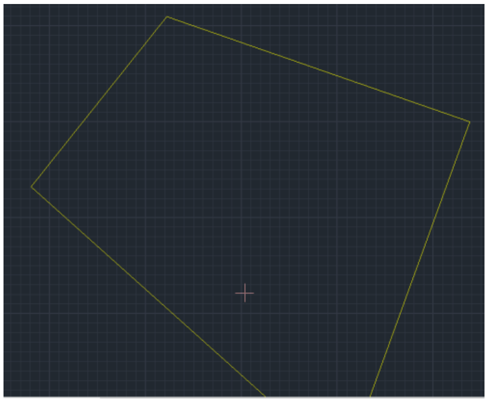


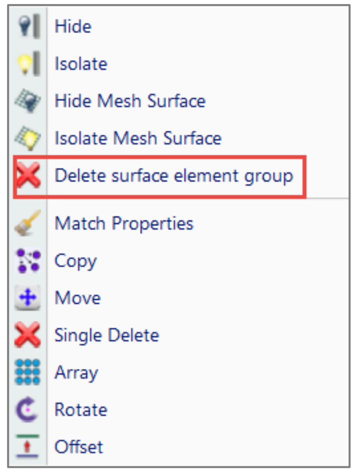
The commands "Cancel/Delete" are used to delete created holes, lines, points etc. Click to delete one of them and then click the button "Calculation" again. The program will calculate the mesh considering the new data (ex. If you delete ¨Lines¨, the new mesh will be generated without lines).



Right-click inside the grid and a list of commands relevant to the grid will open. The Grid Delete allows deleting the mathematical model (if any) as well as the mesh itself from the list.







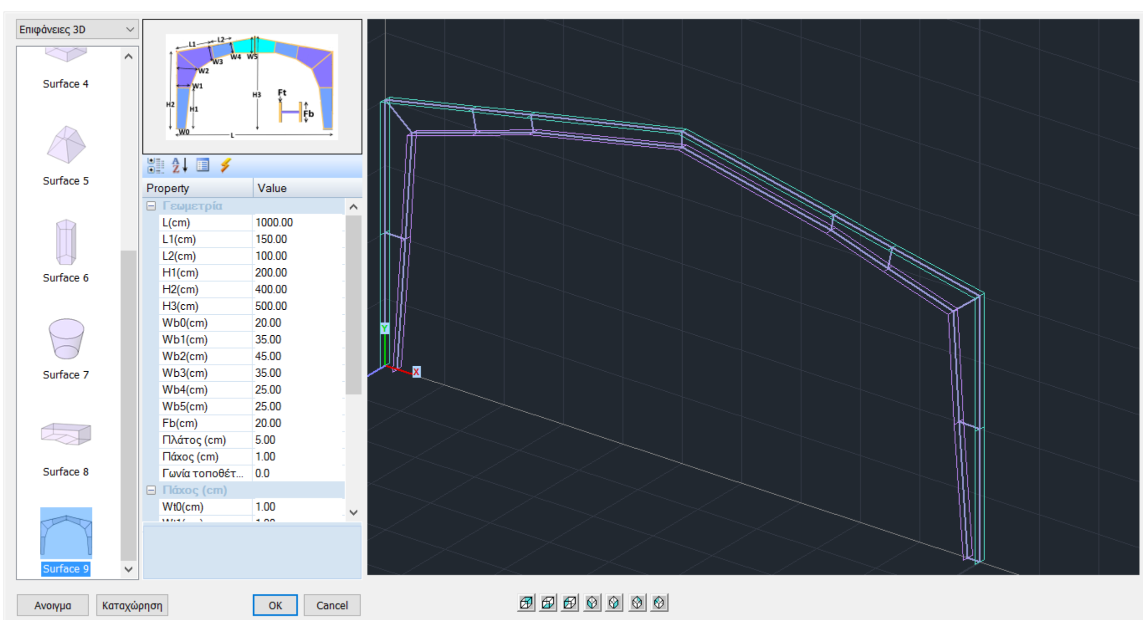
4.2 3D Surface Elements

4.2.1 Mesh

With three-dimensional surface elements, you can model surfaces of any shape (horizontal, vertical, inclined, hollow) and surfaces with common bounds.

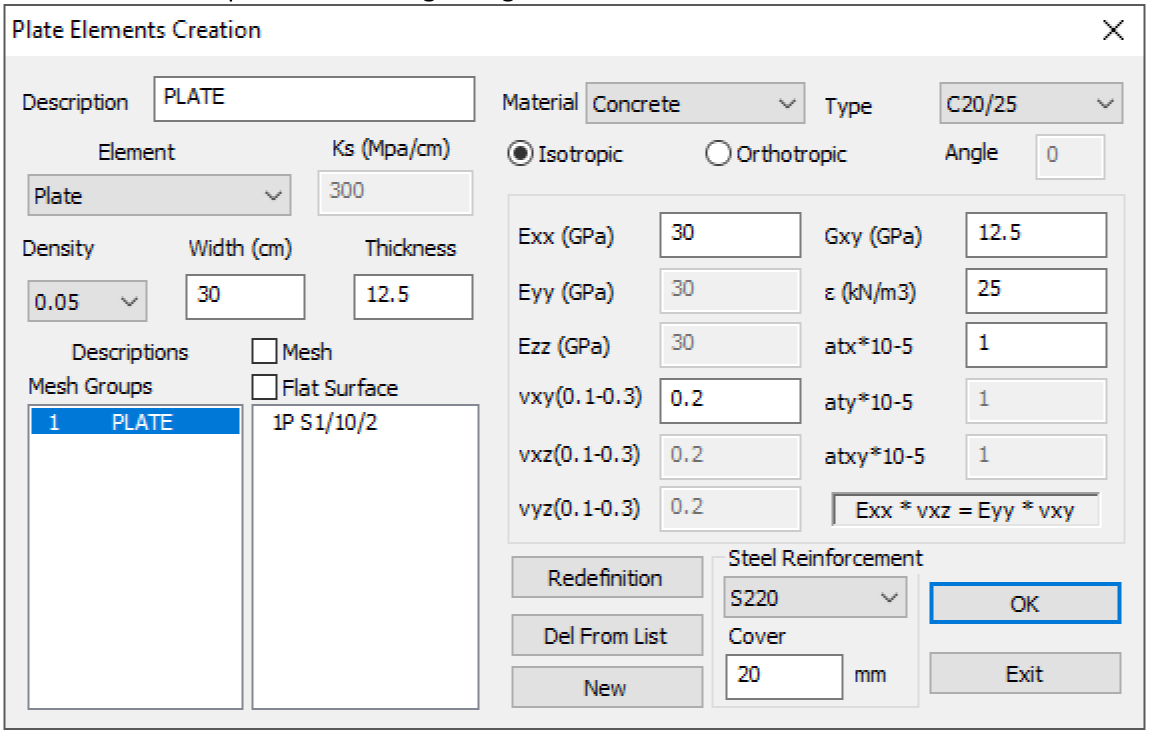
The three-dimensional modeling with surface finite elements is appropriate for projects of Masonry Structures.

There is also the possibility of automatically simulating a typical metallic frame of the variable cross-section with finite surface elements.



SCADA Pro provides multiple possibilities for using 3D surface elements, through "Templates" and the command "Front view identification", which is explained in detail later in this chapter.

Select "Mesh" to open the following dialog box:

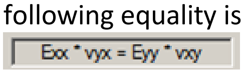


Specify the **properties** of the mesh:
"**Description**": type a short mesh description.
"**Element**": select the type of the surface finite element. If you choose "Plate (O)n (E)lastic (F)oundation" you have to type a value for the soil constant "Ks" in the corresponding box. This selection is appropriate for raft foundation, while the option "Plate" in all other cases.

"**Density**", "**Width**" and "**Thickness**": These fields correspond to the geometry of the mesh.

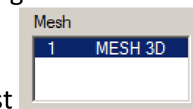
- "**Density**": it expresses a smooth transition from an area with dense mesh to an area with Sparse mesh. High density expresses smoother "flow" of surface elements and course number of elements. Low density can be used if you want to use just a few elements to estimate the stresses in a region (i.e. preliminary design).
- "**Width**": type the width of each element.
- "**Thickness**": type the thickness of the surface elements.

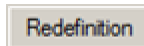
Select "Material" and "Type" and "Isotropic" or "Orthotropic". The orthotropic material allows you to assign different material properties in each direction. In an orthotropic material, the following equality is satisfied concerning Young's modulus and the Poisson's ratio.



"**Angle**": it is for an orthotropic material and will be activated in a next version of the program

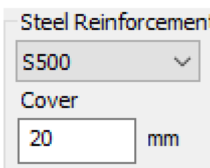
"**New**": define all the appropriate data and then click the button "New". In the list "Mesh", the mesh you have just created will appear with a serial number and the name you defined. Follow the same procedure to create another mesh with different geometric and physical properties.
"**Redefinition**": to save the data modifications of the created mesh.

EXAMPLE:
For example, to change the thickness of the "MESH 3D" from 50 to 60 cm, select from the following drop-down list 

the mesh and then type the new value in the corresponding box. Then, click on 

and the thickness will be updated. Similarly, you can change any other geometrical or physical characteristic of the mesh.

Steel reinforcement and Cover
It is the field where you select the quality of the **steel reinforcement** for the surface finite elements and the mm of the **cover**:



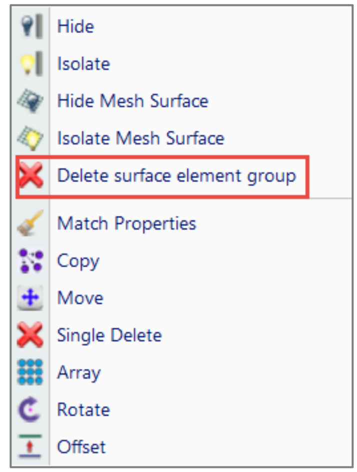
Now it's also possible to define different cover for each mesh of the Mesh Group.

"**Del From List**": it is used to delete one or more existing mesh groups. Select mesh from the list and then click the button "Del from List".

Mesh will not disappear, but you will see the word "Delete" next to the name, which means it has been deleted, having always the possibility to get it back, if you click "Delete" again. Then, the word "Delete" disappears and the mesh becomes active again.

To delete a mesh permanently, after the command "Del from List", save the project by selecting the command "Save".

⚠ *Right-click inside the grid opens a list of commands relevant to the grid. The Grid Delete allows deleting the mathematical model (if any) as well as the mesh itself from the list.*



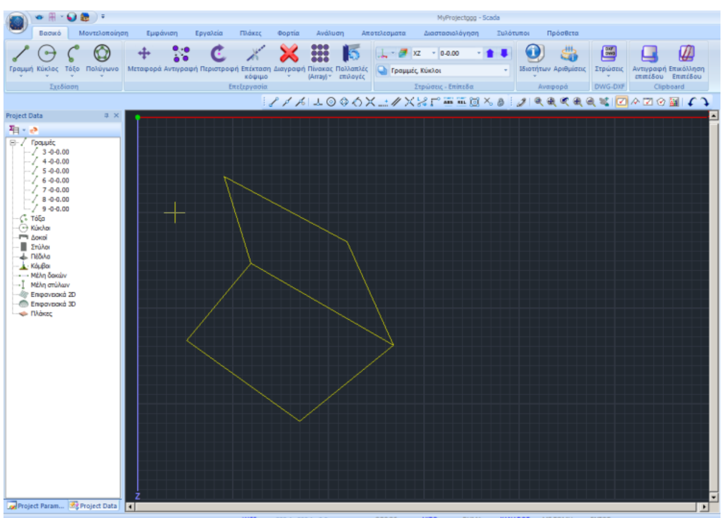
“**Mesh**” and “**Flat Surface**” selections are active and are only used to input 3D mesh elements. You can use these options if the defined mesh has more subgroups.

MESH GROUP CONTAINING MORE MESH SURFACES:

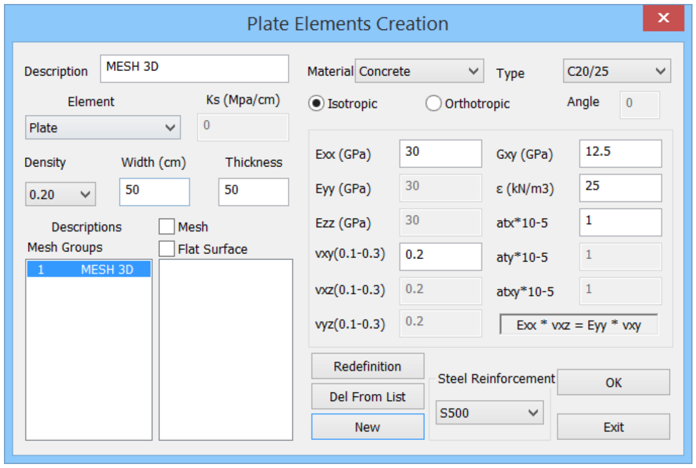
In case we want to model surfaces with common bounds, we create a Mesh Group with more Mesh surfaces so that the nodes on the bounds, which will be created by calculating the math. The model will be common to both surfaces.

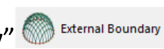
EXAMPLE:

Use line or polyline to design the two areas:

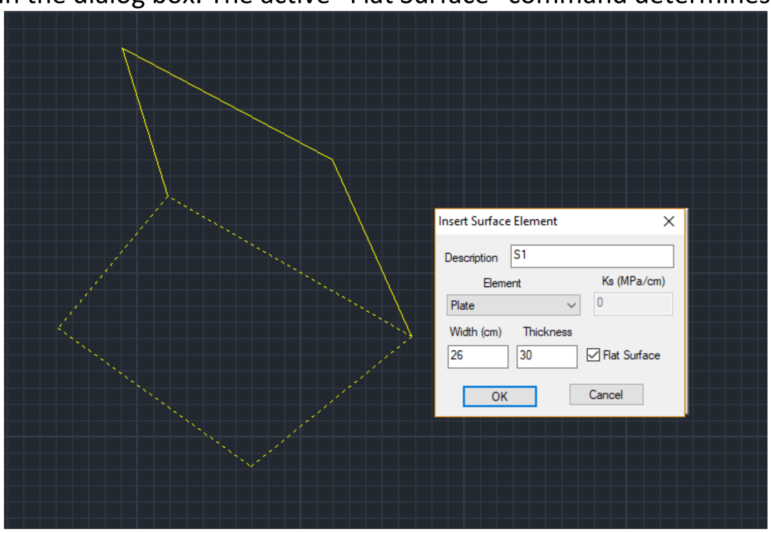


Use “**Modeling**” >> “**Surface elements**”>>”**3D**” >> “**Mesh**”, to define the mesh:

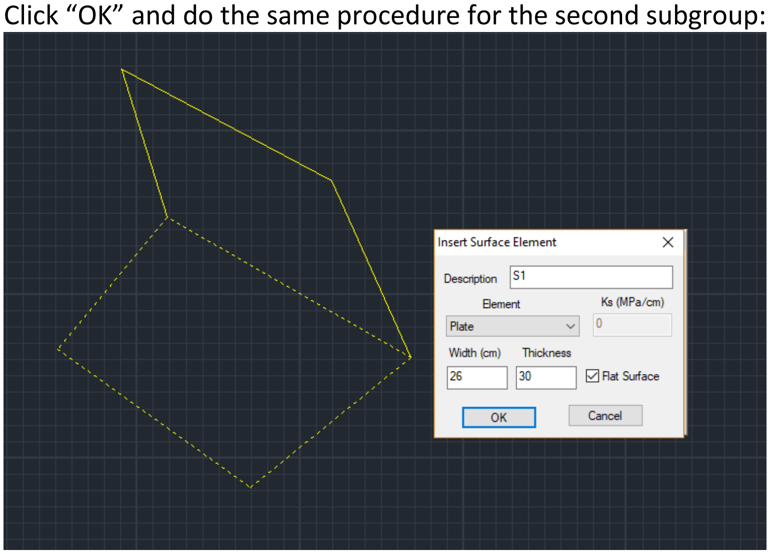


Then select "**3D**">> "**External Boundary**" 

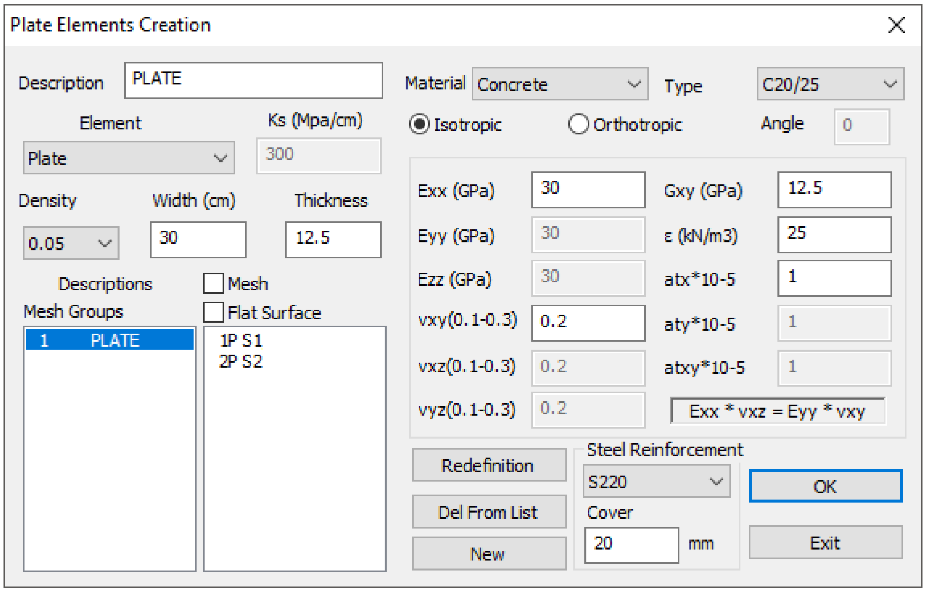
, left click to select the lines of the first boundary and right click to complete. The properties of the first mesh subgroup are displayed in the dialog box. The active "Flat Surface" command determines that the surface is in one level.



Click "OK" and do the same procedure for the second subgroup:



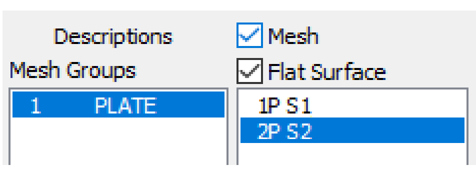
By choosing once again the command "**Modeling**" >> "**3D Surface**" >> "**Mesh**", notice that "**MESH 3D**" contains the two subgroups "S1" and "S2".



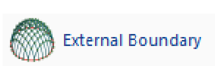
NOTES:

⚠ To delete a subgroup, select it, activate the checkbox "Mesh" and then click "Del From List".

⚠ To change the category of a subgroup from non-Flat Surface to Flat Surface, select it and activate "Flat Surface".



4.2.2 External Boundary



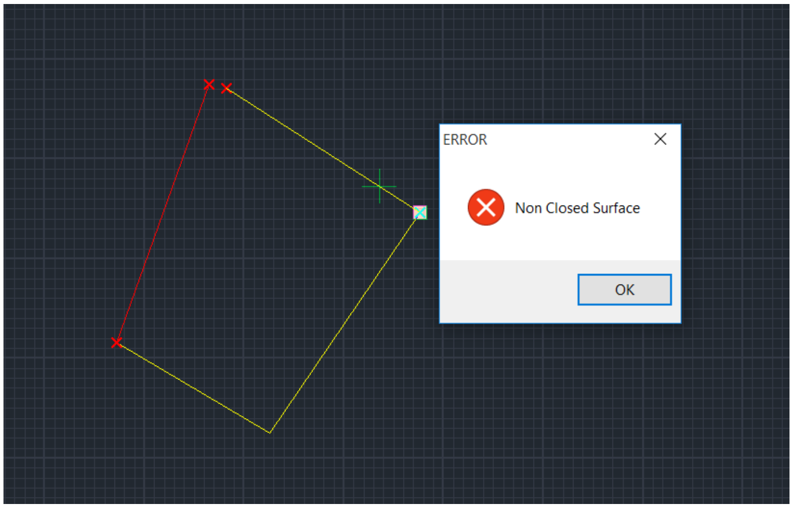
This command is used to define the external boundary of the surface. To design the external boundary of the surface, use line or polyline command. Then select the command and show with left click successively, one by one, the lines or polylines of the contour of the surface.

⚠ *If there is no mesh group generated, the dialog box "Plate Elements Creation" opens for you to create a mesh group. If you have already created it, follow the description below.*

To define the outer limit of the mesh surface, first draw it, using help lines or polyline and then select the command:

1. If the border is closed and there are no common lines with other contours and branches, it demands only to point (left-click) one of the lines and right-click to complete the command.
2. If the border is closed but there are common lines with other contours and/or branches, point (left-click) all contour lines of the surface, one by one successively.

The program provides automatic error detection during the definition of the surface closed contour and displays red cross points where the selected contour is not closed.

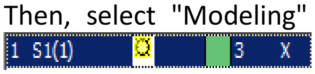


If the surface mesh consists of more than one subgroups, follow the procedure described in the previous example.

To delete lines or polylines that compose an external boundary, you have to delete the corresponding mesh group.

4.2.3 Holes

this command is used to determine the perimeter of the hole inside the surface of the 3D mesh.

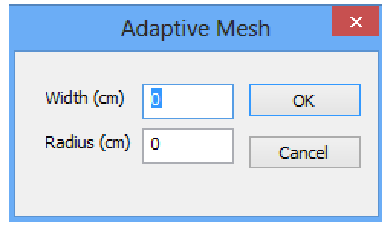
First, use "line" or "polyline" and draw the hole (closed perimeter).
Select the command "Modeling" >> "Mesh 3D" >> "Holes" and left click to select all the sides of the hole.
Then, select "Modeling" >> "3D Mesh" >> "Calculate". Then select the predefined mesh 

and press "Calculate" to generate the mesh which contains the hole.

The parenthesis () next to the symbol S shows the number of the holes, which belong to that mesh.

The definition of the holes can also be done after the mesh generation of the surface. Then use the command "Calculate" to recalculate the mesh which contains the hole(s).

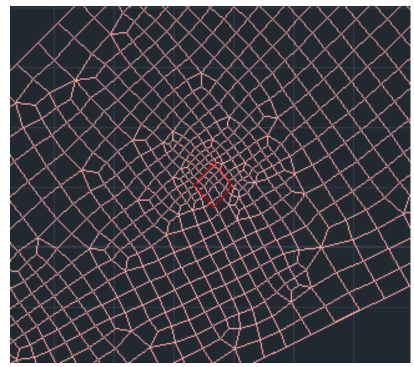
4.2.4 Point



This command is used to define points inside the boundary which will be the regions of the adaptive mesh.

Select the command and click one or more points inside the boundary you defined.

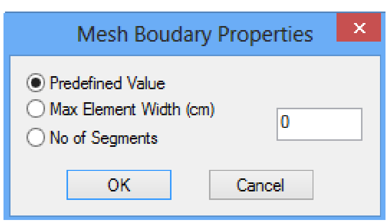
The point's definition can also be done a posteriori, since you have created the mesh surface, using the "Calculate" command that regenerates the mesh based on the point already defined.



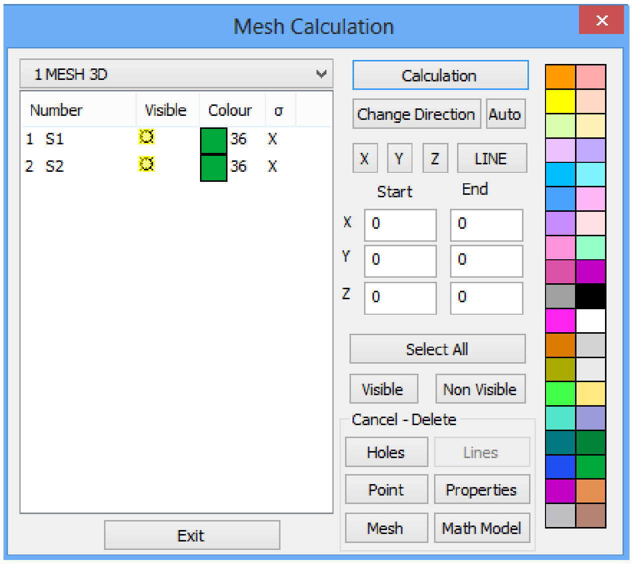
4.2.5 Edit

this command is used to edit a mesh that you have already created (it can be used after the generation of the mathematical model as well).

Select the command and the following dialog box is displayed:



"Predefined Value": This checkbox is used to define the number of the mesh elements. Type the number, click "OK" and then left click on the sides of the boundary. Right click to complete. Then select "3D" >>"Calculate" and the following dialog box appears:

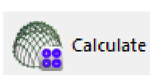


Select one or more mesh subgroups and click "Calculation". The number of the mesh elements will change and the mesh will be updated based on the predefined value.

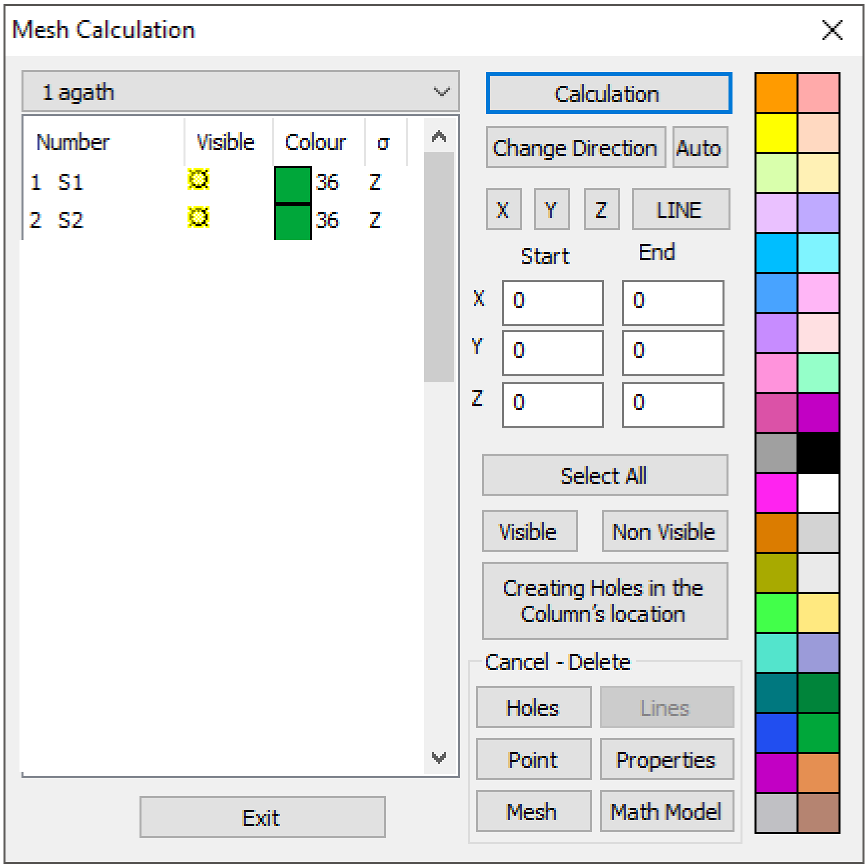
"Max element Width": Type the maximum width of a single surface finite element of the mesh. Click "OK" and then left click on one or more sides of the boundary, where the mesh elements will have the maximum width. Right click to complete. Then select " 3D" >>"Calculate" and in the dialog box select the mesh and click "Calculation". The mesh will be updated considering the maximum width of the single surface finite element.

"No of Segments": Type the number of the segments (not the total surface elements number). Click "OK" and left click on one or more sides of the boundary, to apply the discretization. Right click to complete. Then select "3D" >>"Calculate" and in the dialog box, select the mesh and click "Calculation". The mesh will be updated considering the number of the segments.

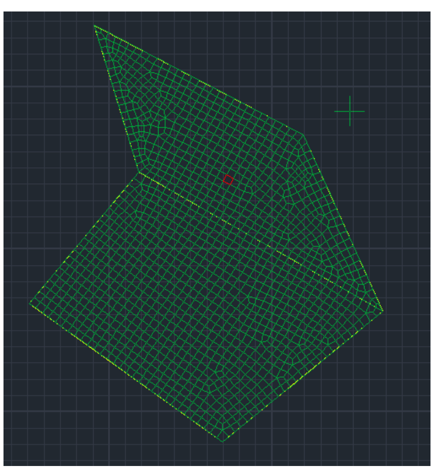
4.2.6 Calculate



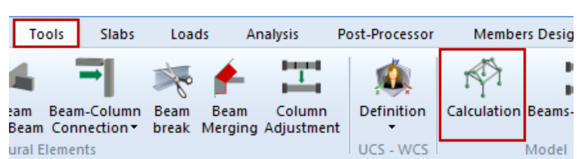
This command is used to calculate a 3D mesh by using an already defined surface while considering possible points and holes.
Select the command and the following dialog box is displayed:



From the drop-down list in the dialog box, choose the mesh you want to calculate. Note that each boundary must belong to its mesh.
In order to generate the mesh, first, select the mesh group or subgroup from the list and then click on "Calculate". The same procedure is applied for the generation of a different mesh group.
The simulation results are shown in the figure below:

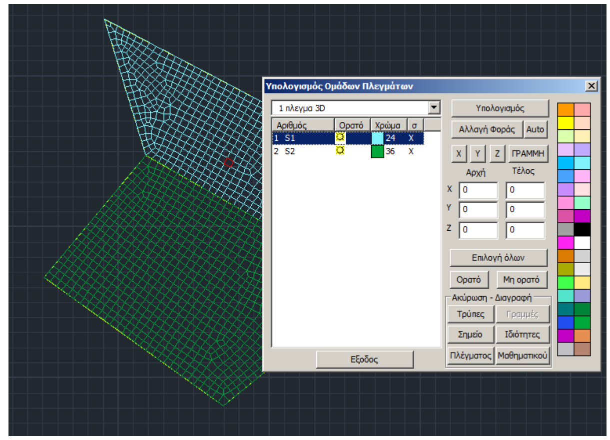


This command is used to create the mesh but not its mathematical model. For this purpose use "Tool" >> "Calculation".



The same dialog box includes not only the calculation process but some other useful tools:

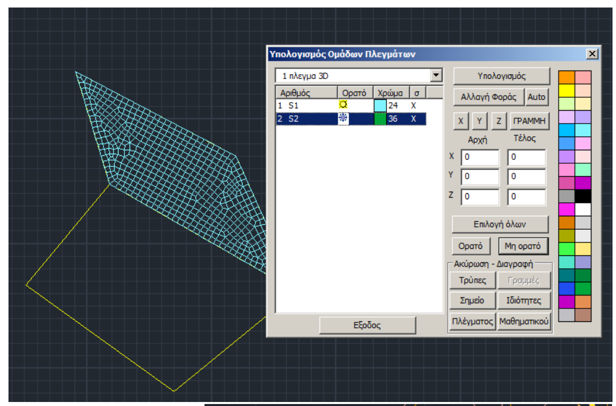
To change the color of the surface element, select the plate from the list and click on the desired color from the palette of colors.

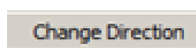


Use the button 

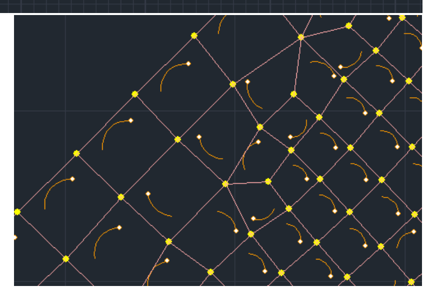
or 

, to show or hide the selected mesh. The indicator under the column/changes from visible () to invisible (). Since the mathematical model has been generated you can apply this change



⚠ Use the command 

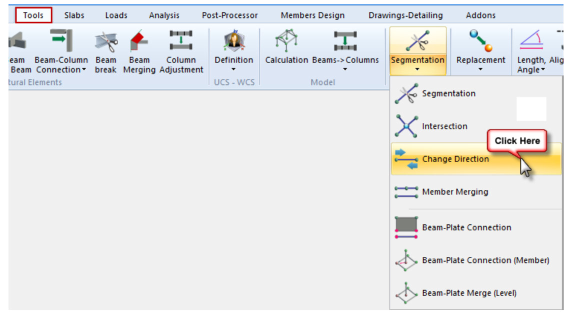
to change the direction of the local axes of the mesh elements.



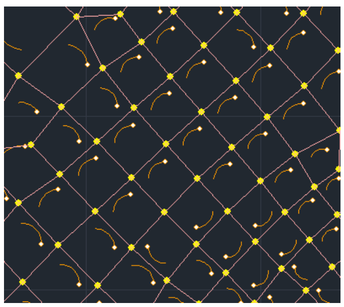
⚠ *It is useful to display Local axes after the mathematical model's generation, which can be activated through the unit "View" >> "Switches" >> "Local Axes".*

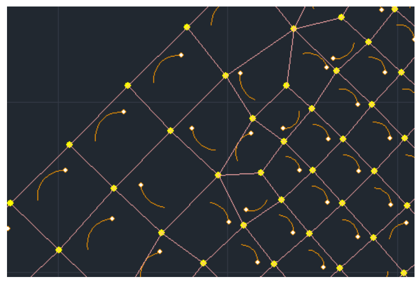
The command "Change direction" changes the direction of all surface elements of the selected plate.

⚠ *If you want to change the direction of specific surface elements, use the command "Change direction" through the unit "**Tools**" >> "**Segmentation**" >> "**Change direction**".*



With the command "Auto", the program adjusts local axes of all plate surface elements to have the same direction.



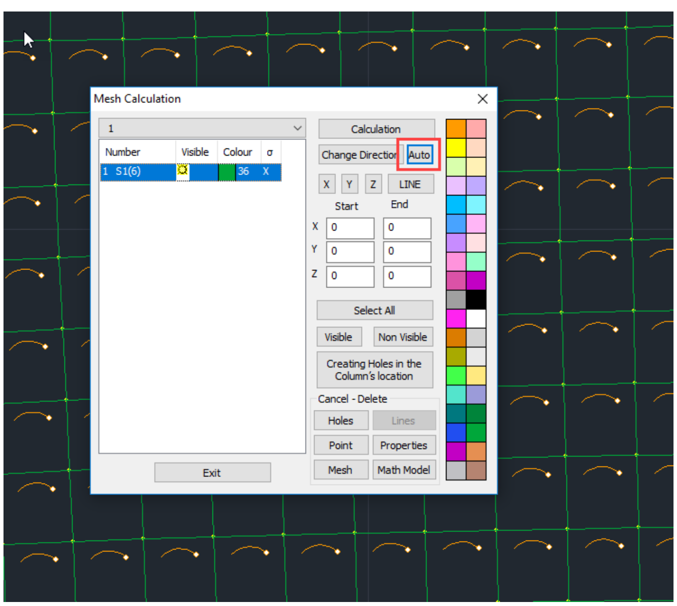


NOTE:

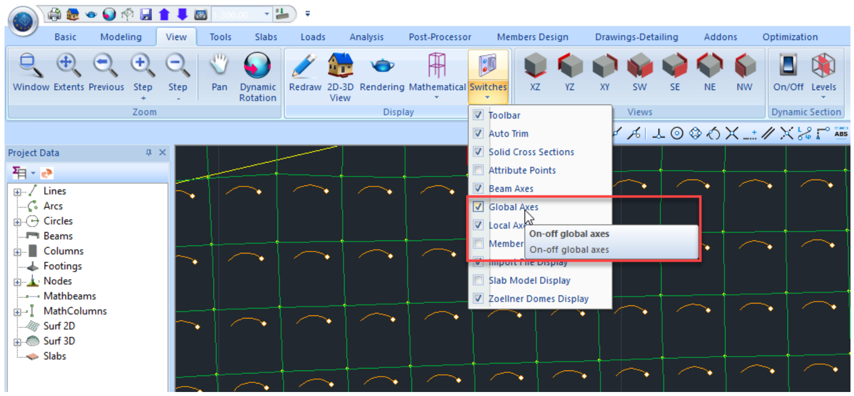
⚠ *After the creation of the Mathematical Model of the mesh, always return to the window "Mesh Calculation" and press the "Auto" button.*

ATTENTION:

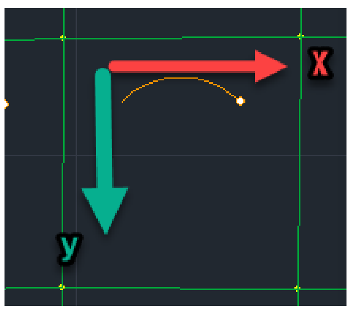
Every time you define a 3D Mesh, and after the Mathematical Model creation, ALWAYS go back to the Mesh "Calculation" command, for the automatic redefinition of the local axes of the surface elements, using the "**Auto**" command.



To display the local axes of the surface elements, activate the corresponding option through the "Switches".



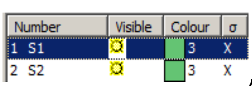
The arrows appearing defining local axes of the surface elements, according to the right hand rule. The direction of the arrow indicates the **x**-axis and the point on the edge indicates the orientation.



Use the fields depicted in the figure on the right to define the main direction of the steel reinforcement (direction X, Y, Z). 

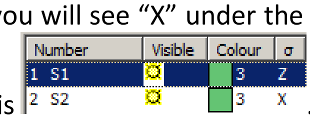
EXAMPLE:

Select a plate from the list and if you choose you will see "X" under the column ''σ'',



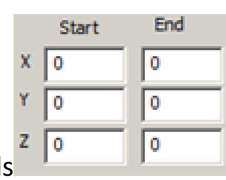
Number	Visible	Colour	σ
1 S1		3	X
2 S2		3	X

, but if you choose you will get this



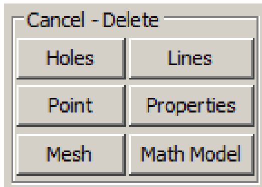
Number	Visible	Colour	σ
1 S1		3	Z
2 S2		3	X

.

Use the command "Line"  combined with the following fields 

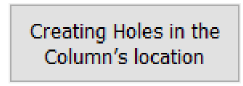
, to define the line's coordinates. The program will consider the line direction as the main direction of the steel reinforcement. Use this command when the meshing area is not parallel to the main axes.

The commands "Cancel/Delete" are used to delete created holes, lines, points etc. Click to delete

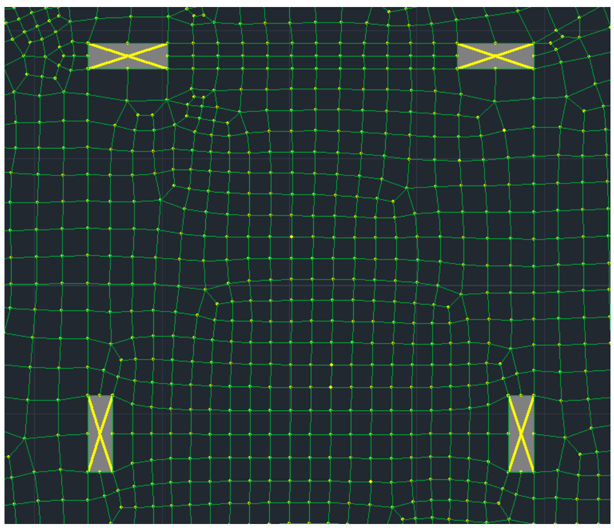


one of them and then click the button "Calculation" 

again. The program will generate the mesh considering the new data. For example, if you delete "Lines", the new mesh will be generated without lines.

The command "Creating Holes in the Column's location" 

is used in the «*Flat Slabs*» and allows the automatic creation of holes in the flat slab area where columns exist. (cf. Chapter 9 - Flat Plates).



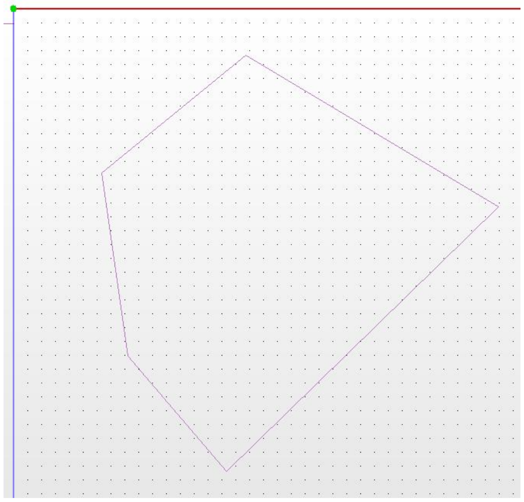
4.2.7 Front View Identification:

SCADA Pro gives you the ability to create a masonry structure on any external boundary, by using the tool "Templates", quickly and easily.

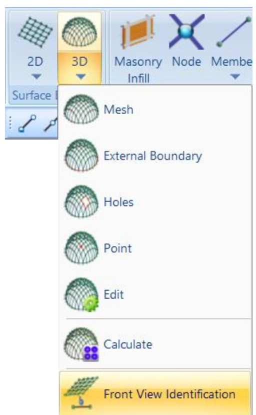
The process is the following:

(i) Enter a plan view in DXF or DWG file format by using the command group "Draft" in a closed surface on X, Z plan level.

Unit: "**Basic**", command path: "Draft">>"Line">>"Polyline" → create a surface → right click.

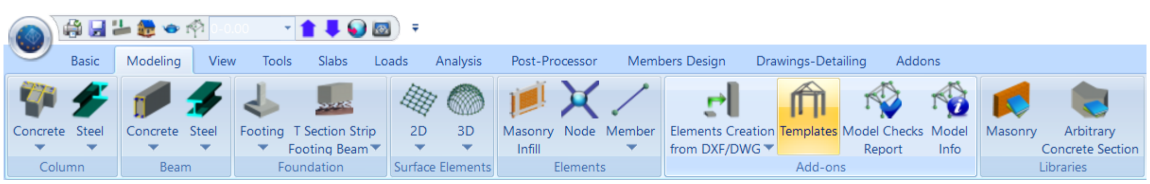


(ii) Unit: “**Modeling**”, command path: “Surface Elements”>>” 3D”>>“Front View Identification”.

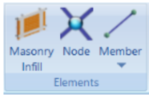


Then use the selection command “Window”  to select the total plan view. Right click and the masonry templates dialog box is displayed:

⚠ (see Templates - Masonry)



5. ELEMENTS



"Elements" command group contains commands to define and insert:

- Masonry Infill

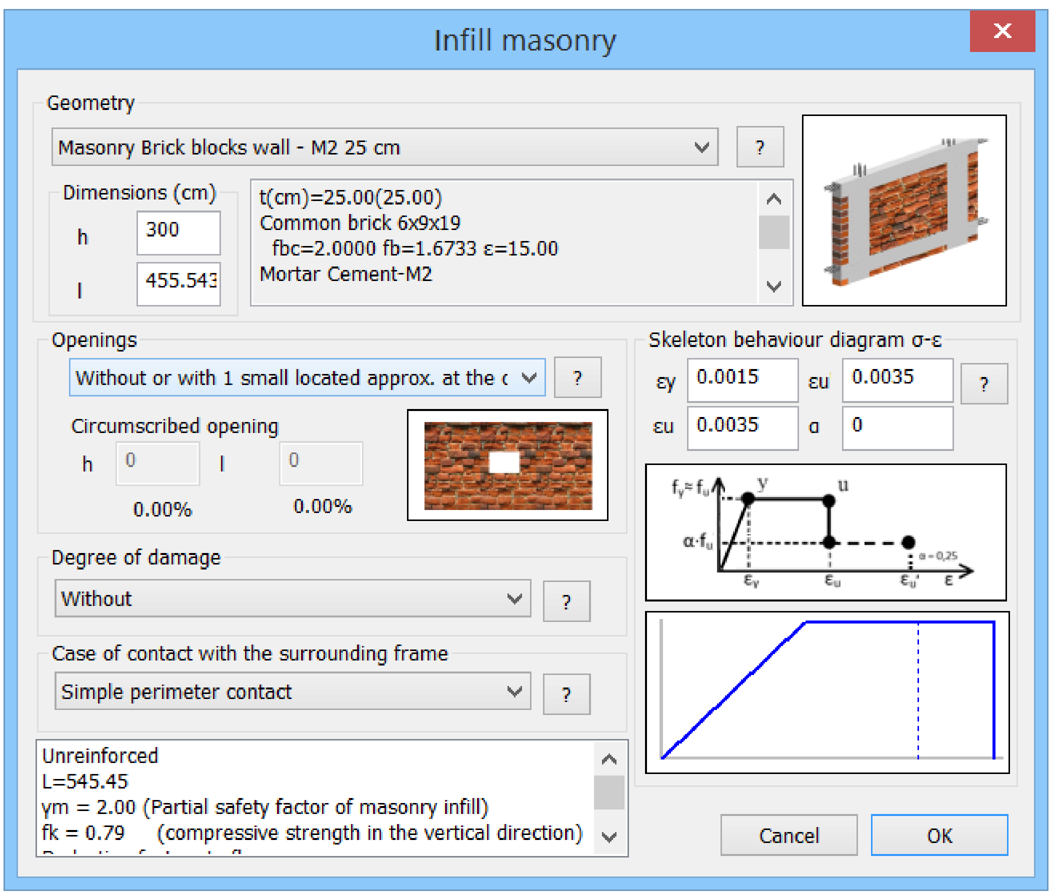
-Node and

-Member (Linear-Surface)

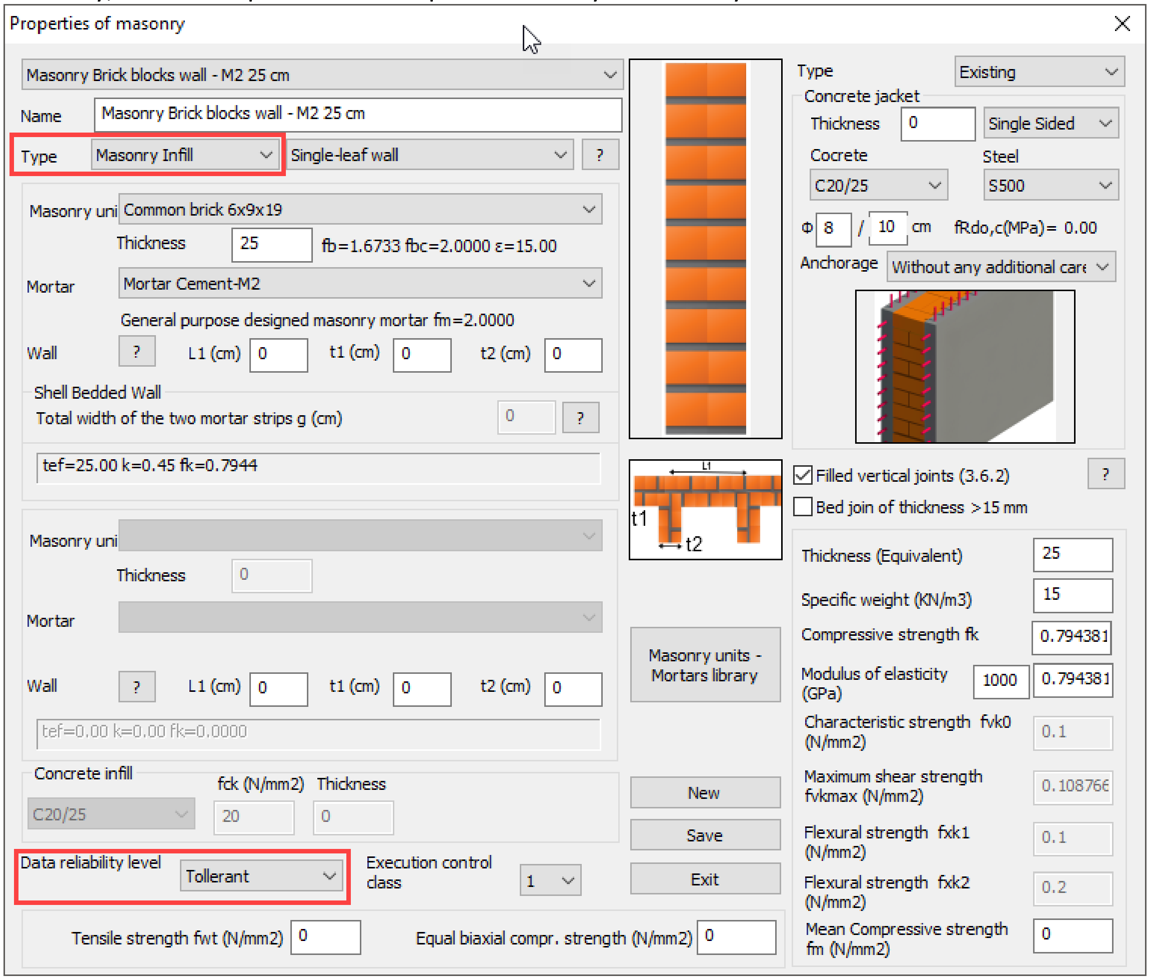
5.1 Masonry Infill

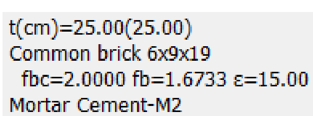
Using this command you can model the masonry infills. Modelling is accomplished with two diagonal bars with zero specific weight (wall loads have already been taken under concideration as linear loads on the beam members). Select the command and left click on the upper beam member of the panel that contains the masonry infill.

The dialog box with the relative parameters is diplayed:

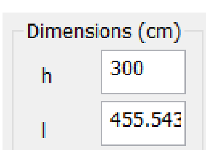


In "Geometry", select from the drop-down list the wall that has been created in the library of masonry, otherwise press  to open the library of mansory and define it.

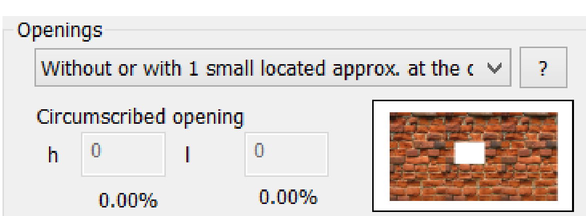




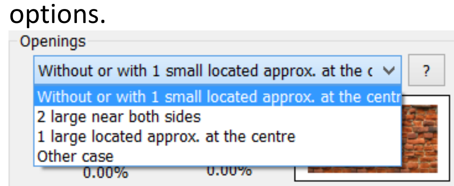
The wall properties such as the total thickness t (mantle and wall), the type of masonry with its strength and the type of the corresponding mortar strength are displayed.



In "Dimensions" the editable height (h) and width (l) properties of the panel which are automatically calculated by the program, are displayed.



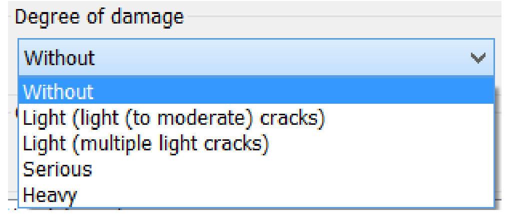
"Openings" are for defining the wall openings. Select from the list one of the options.



Choosing "Other case" you must type the corresponding dimensions.

Define the Openings to calculate the reduction factor of the compressive strength n1.

Define the Degree of damage to calculate the reduction factor of the compressive strength rR.



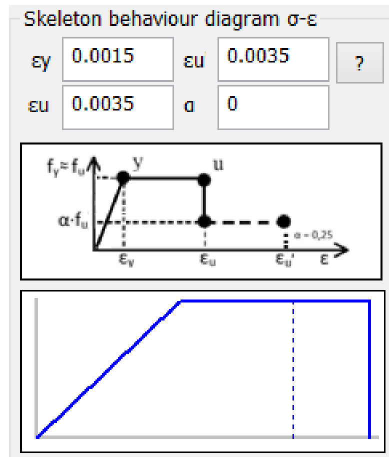
In the end, select the Case of contact with the surrounding frame, affecting the calculation of the reduction factor n3 concerning slenderness.

Skeleton behaviour diagram tension – deformation of the masonry infill.

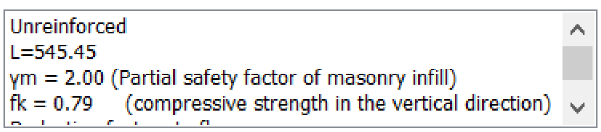
The graph is drawn automatically.

For unreinforced masonry, you have to type the corresponding values. For reinforced masonry, εy and εu are calculated automatically. In both cases, the values can be modified by the user.

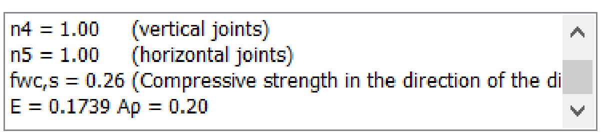
α factor is the proportion of the residual strength after fracture and conserns only the reinforced masonry, such as the reduced distortion total failure ε'u.



The choises and the results can be seen here:


Unreinforced
L=545.45
γm = 2.00 (Partial safety factor of masonry infill)
fk = 0.79   (compressive strength in the vertical direction)

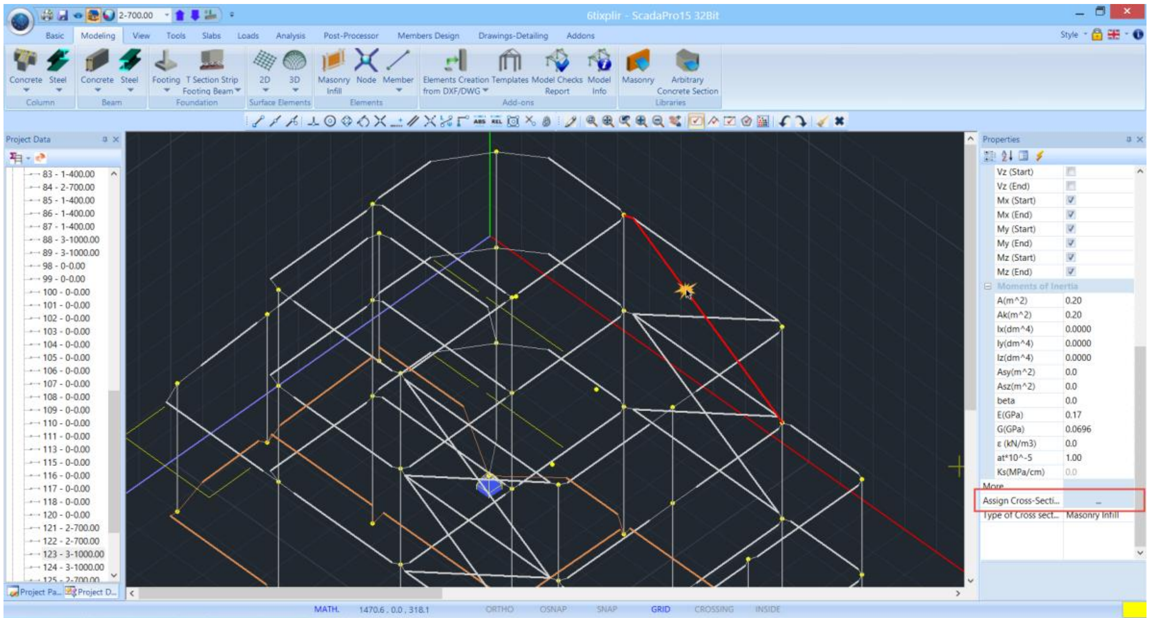
The compressive strength, the elastic modulus, the partial safety factor, etc.


n4 = 1.00   (vertical joints)
n5 = 1.00   (horizontal joints)
fwc,s = 0.26 (Compressive strength in the direction of the di
E = 0.1739 Ap = 0.20

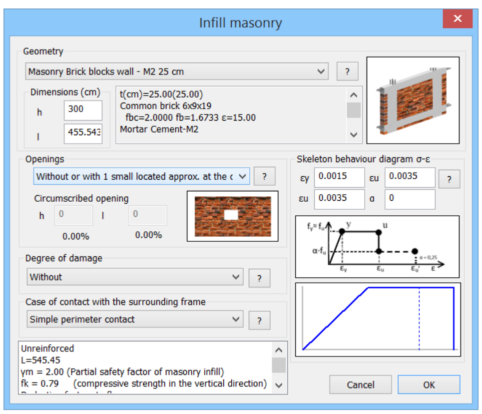
Press OK to create automatically the two diagonal members in the panel.

The masonry infills can be inserted either in the plan view of each storey or 3D View.

Reporting, editing, and modification can be performed through the "Assign Cross-Section" command of the properties.



Left click on a diagonal member and in Properties select “Assign Cross-Section” to open the respective dialog box and make the changes.



ATTENTION:

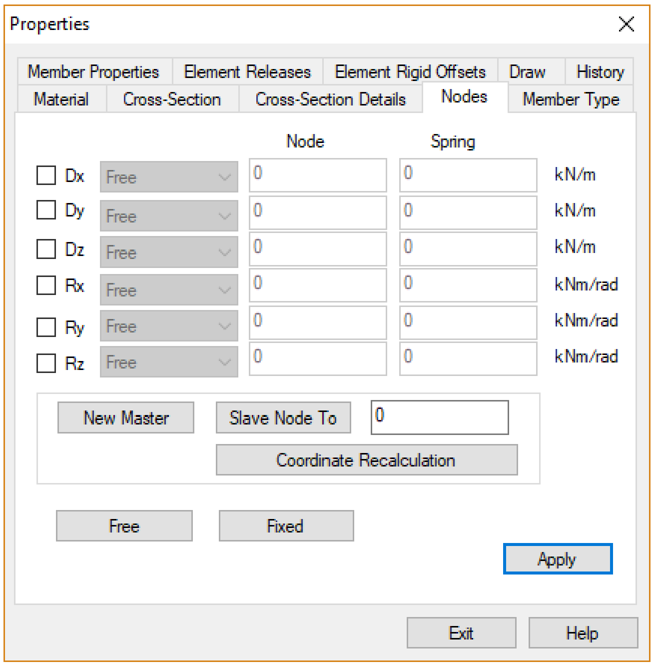
Changes affected only the selected member. To change other members, you must follow the same procedure for each one.

ATTENTION:

If you change data contained in the library of the masonry infill, changes are not automatically updated. To update the properties of the members of the masonry infills according to the new data, corresponding to the updated library, you must open the “Plate Elements Creation” form, select it and click Redefinition .

5.2 Node

This command is used to define and insert nodes on the mathematical model.
Select the command and in the following dialog box define the node's parameters:



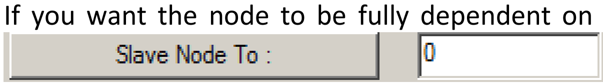
Type the serial number "**A/A**" and the "**Coordinates**", otherwise, the program will fill them up automatically. In this case click "OK" and show "i" and "j" node by left clicking on 2D or 3D display.

"Degrees of Freedom"

Define the degrees of freedom of the node. There are four choices concerning the status of the corresponding displacement or rotation of the node:

"Free", "Fixed", "Slave", "Spring".

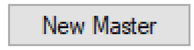
- "**Free**": This type of constraint allows the node's displacement and rotation in the corresponding direction.
- "**Fixed**": This type of constraint doesn't allow either the displacements or rotations.
- "**Slave**": It means that the displacements or rotations of the node depend on a master node, which is indicated in the column "Node" and it is activated automatically when you select "Slave".

You can also depend on the displacements and rotations on another node (Master Node).
If you want the node to be fully dependent on one node, click the button "Slave Node To"


and type the number of the master node.

- "**Spring**": This selection activates the "Springs" to type the soil constantly.

"New Master"

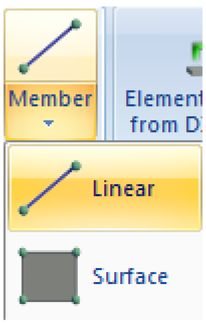
When you select the command "New Master" 

the selected node or nodes are becoming depended on a newly created master node.

"Layers"

Select from the drop-down list the layer, to which the node will belong. As default is the "Mathematical Model".

5.3 Member



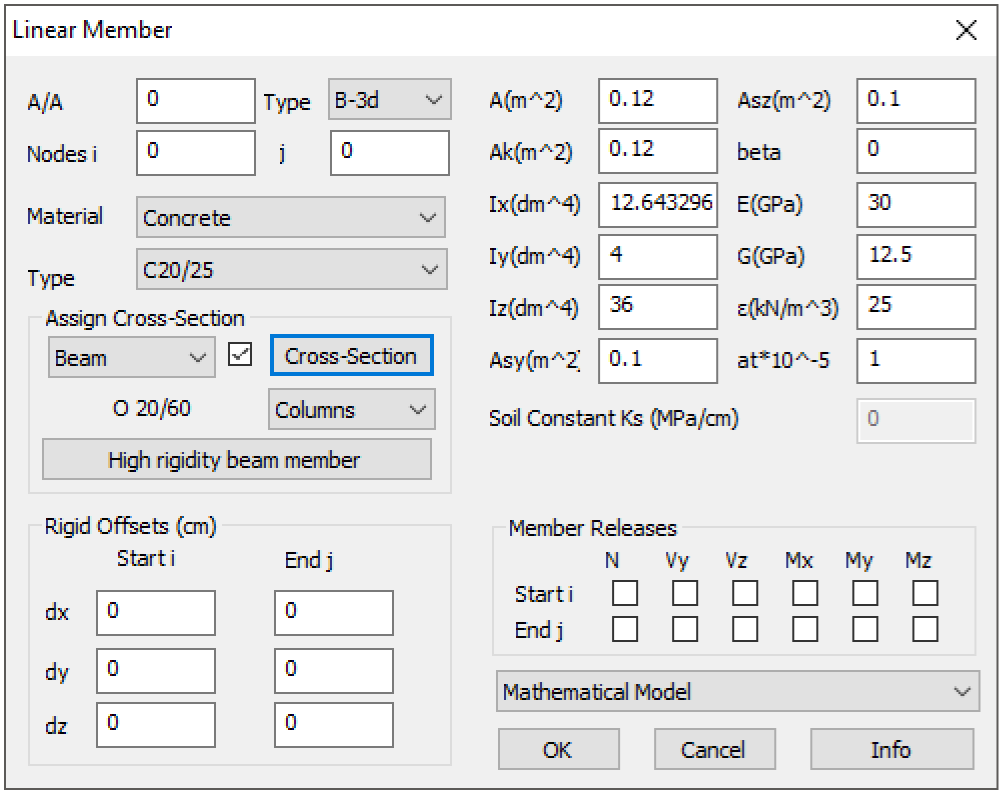
5.3.1 Linear

This command is used to define and insert linear mathematical members.

Except for the ability to direct type the physical characteristics of the member, the program gives the opportunity of calculating automatically these data by entering the corresponding cross section.

SCADA Pro contains three types of linear members: **B-3d**, **Truss** and **B-3def** beam on elastic foundation.

Select the command and in the dialog box define the parameters of the linear members:

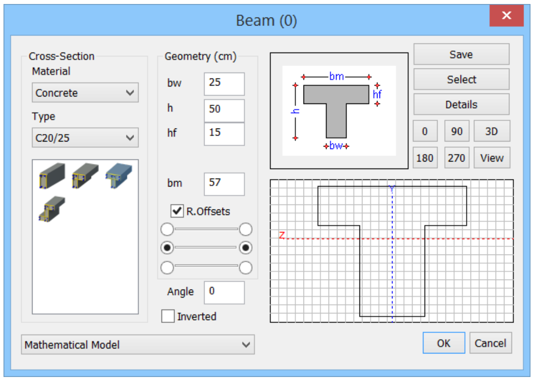


Define the “Material” and the “Type”. The assignment of a cross section to the linear member is available. Type the serial number “A/A” and the number of the first node “i” and the last node “j”, the Rigid Offsets coordinates and the geometrical and inertial characteristics. Otherwise, let the program perform automatic calculations. In this case, click “OK” and show node “i” and node “j” by left click on 2D or 3D display.

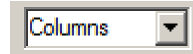
More specifically:

• “Type”,
B-3d: The most common member type. Subject to tension, compression, bending, shear depending on the member releases.
Truss: The member type that is subject only to axial forces.
B-3def (Beam 3d on elastic foundation): The member type used for modeling the foundation beams. In this case, there are not axial forces. The displacement in x and z direction and the rotation in the y direction, of the first and last node, are fixed.

• “Assign Cross Section”,
Assign a cross section and the program will calculate automatically the inertial characteristics of the section.

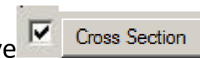


Select from the first drop-down list 

beam or column for concrete members or from the second drop-down list 

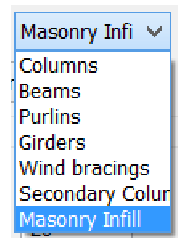
for steel members. Click on the button "Cross Section" 

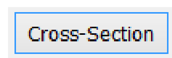
, and in the dialog box type the geometric characteristics of the section: Click "OK" and the parameters in the dialog box "Linear Members" will be automatically filled in.

If the "Cross Section" checkbox is active 

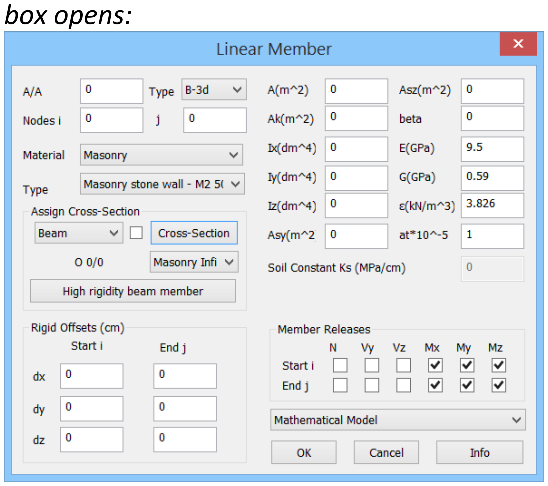
, it means that the member has a "physical" representative that will be designed and calculated. Otherwise, the members will participate only in the mathematical model with their internal forces.

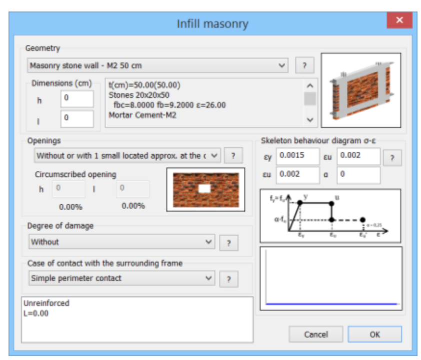
- ⚠ *Below the command of "Cross section", there is a drop-down list that deals exclusively with **steel** cross-sections and the selection of the corresponding group is involved for the modeling of a steel cross-section.*
- ⚠ *In case that you want to insert the diagonal members manually, you can choose "*Masonry Infill*".*



Choosing Material: Masonry and "Masonry Infill". Selecting 

the respective dialog box opens:

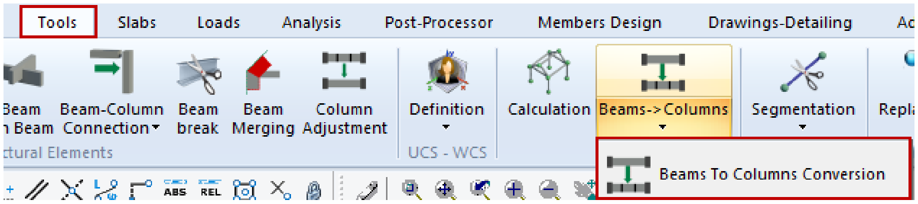




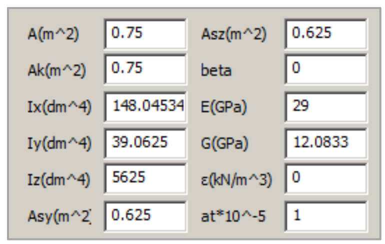
The data import process is identical to the one mentioned previously in the automatic process, except for the fact that you must enter by yourself the length l and the height h of the panel. Positioning the diagonal bars is a manual node to node process.

High rigidity beam member

This is a useful tool for the automatic import of data, which is applied mainly for the simulation of basement walls, by using the command "Beams to Columns Conversion».



By clicking the button “**High rigidity beam member**”, parameters’ fields are completed automaticaly with the characteristics of a high rigidity cross-section; zero specific weight and without an assignment of a physical cross-section.



“Rigid Offsets”

Type the lengths of the rigid parts of the elements, located at the start and end point of the member in cm.

"Geometric and inertial parameters"

Here the user can enter custom geometric and inertial parameters of the linear member, otherwise, the program will calculate them automatically by selecting the command "Cross section".

A: the area of the section, (in m^2)
Ak: the area of the web of the cross-section, (e.g. reinforced concrete T cross-section in m^2)
Ix, Iy, Iz: Main moments of inertia in x, y, z axes respectively (in dm^4)
Asy, Asz: shear surfaces of the section in y and z axes, respectively (in m^2)
beta: beta angle (in degrees)
E, G: Young’s modulus of elasticity and shear modulus, respectively (in GPa)
ε: specific weight of the material (in KN/m^3)
at: coefficient of thermal expansion
Ks: soil constant (in MPa/cm). It is active only for B-3def members.

“Member Releases”

By default, all releases are inactive, which means that all the internal forces are active. Activate the relative checkbox to omit the corresponding internal force at the starting or the ending node.

" Layers List "

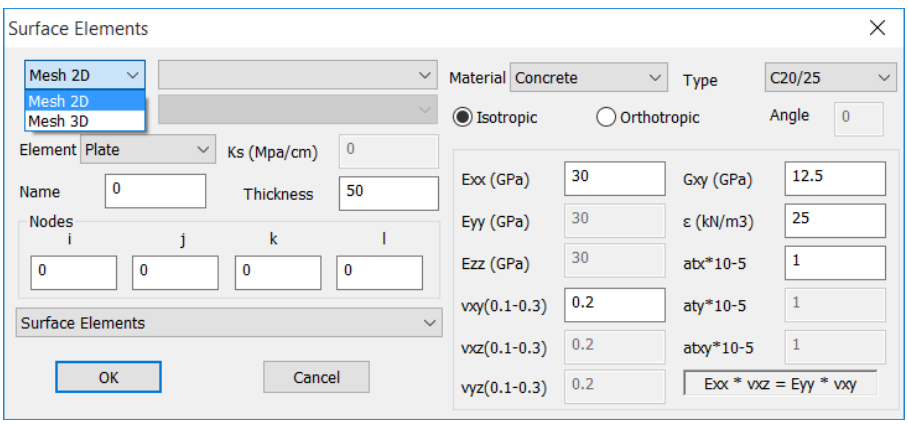
Select the layer, to which the member will belong. The default layer is the "Mathematical Model".

5.3.2 Surface

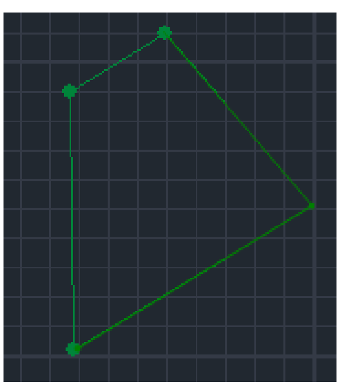
This command is used for the manual creation of individual 2D and 3D surface elements.

*This command assumes that you have already defined the characteristics of the mesh through the command "**Mesh**" in the command group "**Surface Elements**", while it enables individual modifications.*

Select the command and the following dialog box is displayed:



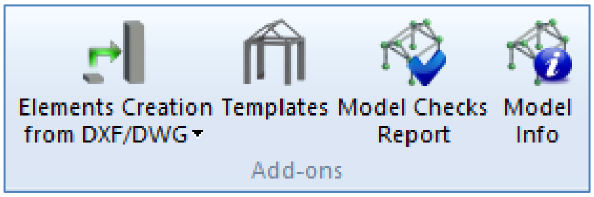
First, select the mesh type and the predefined surface. Parameters' fields are filled in automatically and can be modified manually.



In the field "**Nodes**" you may type the number of the nodes of the single surface element, or type nothing and select the four nodes graphically, by left clicking. The element will be formed on the input interface.

6. Add-Ons

The command group **"Add-Ons"** contains commands particularly useful for modeling process:

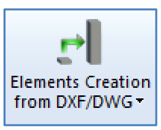


- Elements Creation from DXF/DWG
- Templates
- Model Checks Report
- Model Info

6.1 Elements Creation from DXF/DWG

The import of the floors' plan view to SCADA interface offers multiple features. The user can import to each level the corresponding plan and take advantage of the snap points of the plan for the elements' import.

Select "File">"Import" and open the DXF/DWG file into the project.

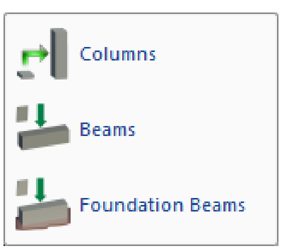


With the command "Elements Creation from DXF/DWG", SCADA Pro offers an additional unique feature that simplifies and accelerates the modeling of your project remarkably.

This is the automatic generation of data from DXF / DWG.

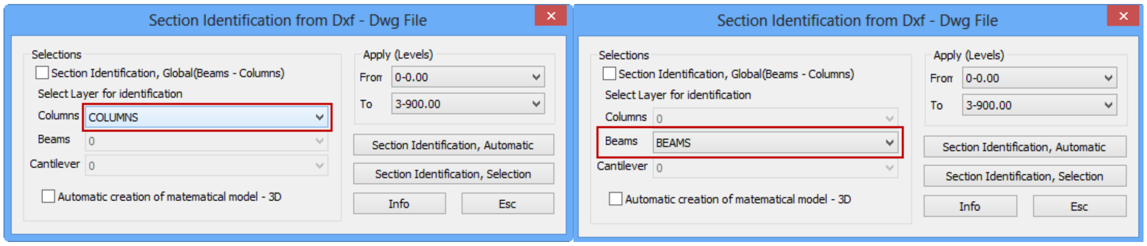
Prerequisites for using this command:

1. Existence of levels
2. The imported plan views of the floors' (DWG / DXF files) must belong to the corresponding levels

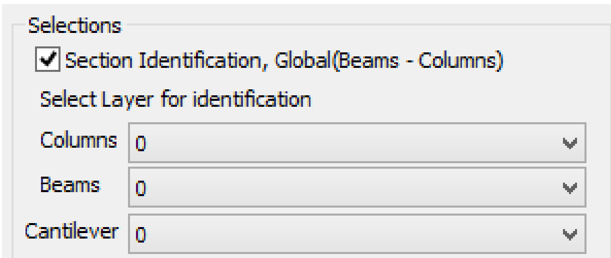


The command "Element Creation from DXF/DWG" contains a list of commands presented in the figure on the left.

Each selection opens the same dialog box and activates the relative element (beams or columns):



Furthermore:

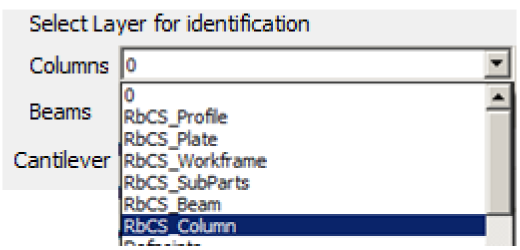


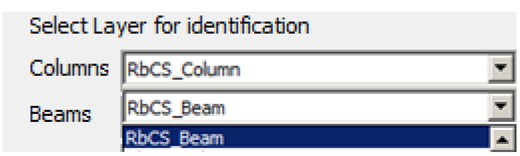
By activating the checkbox “**Section Identification, Global**”, all the elements (Columns, Beams, and Cantilevers) are activated for simultaneous identification.

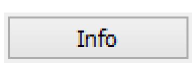
The drop-down list with the arrow below the label "Select layer for identification" (columns, beams, and slabs) contains all DWG layers.

An important prerequisite for the proper function of the automatic identification is that columns, beams, and slabs, must belong to a distinctive layer.

EXAMPLE:

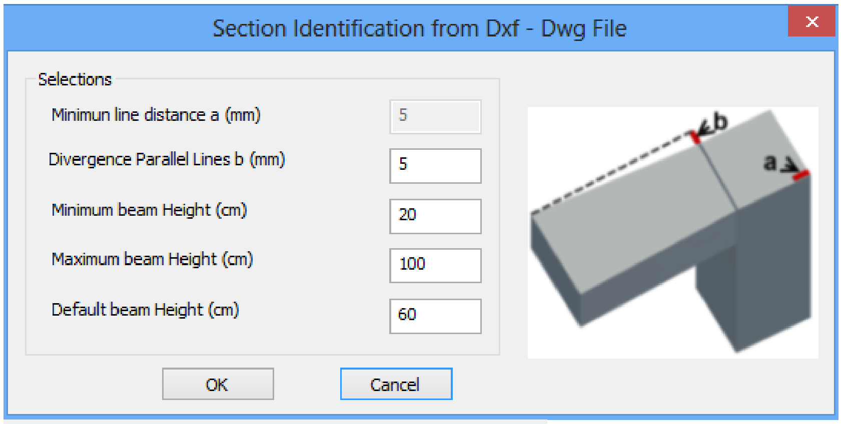




Press 

to define some extra geometric parameters:

- The first two parameters correct the possible design errors, for example, gaps, parallelism deviation etc. (see figure below)
- The last three parameters determine the parallel lines that define the beams and the height of the beam.

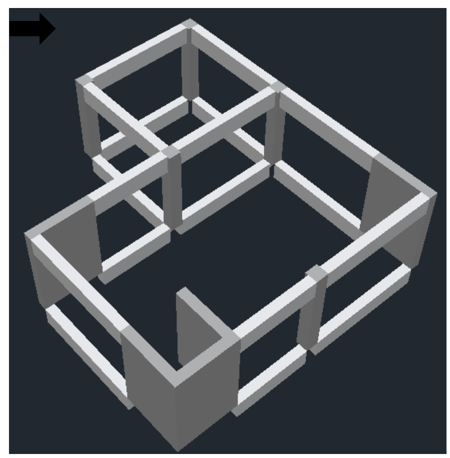


☑ Automatic creation of matematical model - 3D

By activating the automatic creation of the mathematical model, the program not only identifies and imports the physical sections (physical model) but also calculates the inertial properties and creates directly the mathematical model too.

⚠ *A Prerequisite for the automatic identification of the slabs and the cantilevers is the preceding columns and beams identification as well as the mathematical model creation.*

Choose the command [Section Identification Automatic]



You receive directly the photorealistic visualization of the model.

Choose [Section Identification, Selection]

Section Identification, selectively > **Columns**

select the columns one by one by clicking with the left mouse button at a point on each column.

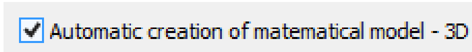
Section Identification, selectively > **Beams – Foundation Beams**

Select beams, as it was mentioned previously, one by one.

⚠ *To display the beam dialog box and set the height of the beam, set the pointer of your mouse inside the contour of the beam and press SHIFT on the keyboard. Enter the geometric characteristics and continue by clicking on the beam.*

⚠ *The beams' automatic import by using the command "Section Identification, Automatic", creates rectangular cross section beams with 60 cm height. You can define the height of the elements from the beginning by using the SHIFT button or after their placement through the properties toolbox that opens on the right of the screen whenever you select an element.*

⚠ *Make sure that in the drawing file DXF / DWG, the outlines of the columns and the beams are closed and defined by a polyline or individual lines, one for each side.*

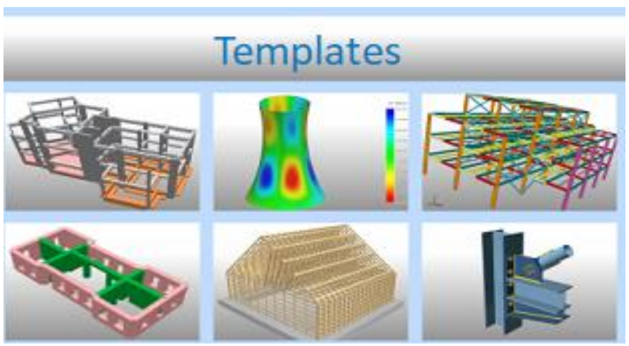
Without the members of the beams and columns, the import of the slabs will not work. That’s why the slab identification works, only with the following check 

being activated.

6.2 Templates

There are two modes to access the “Templates” command:

1st Mode:
Left click on the icon of the start screen window:



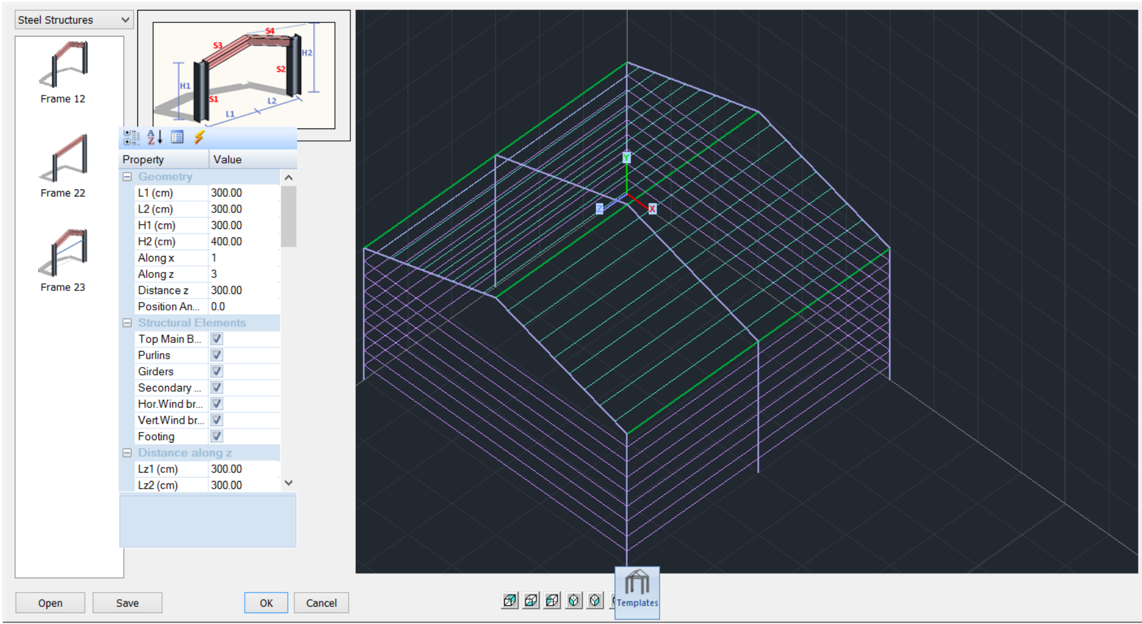
Concrete - 2D and 3D finite elements - Steel
Masonry (3D finite elements) – Timber

- Give a name to your project file in the dialog box that opens.

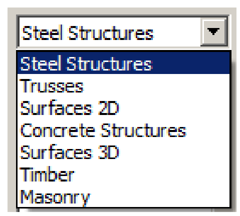
2nd Mode:

- Select the “Add-Ons> Templates”
- Choose the insertion point to the interface, close to the origin.

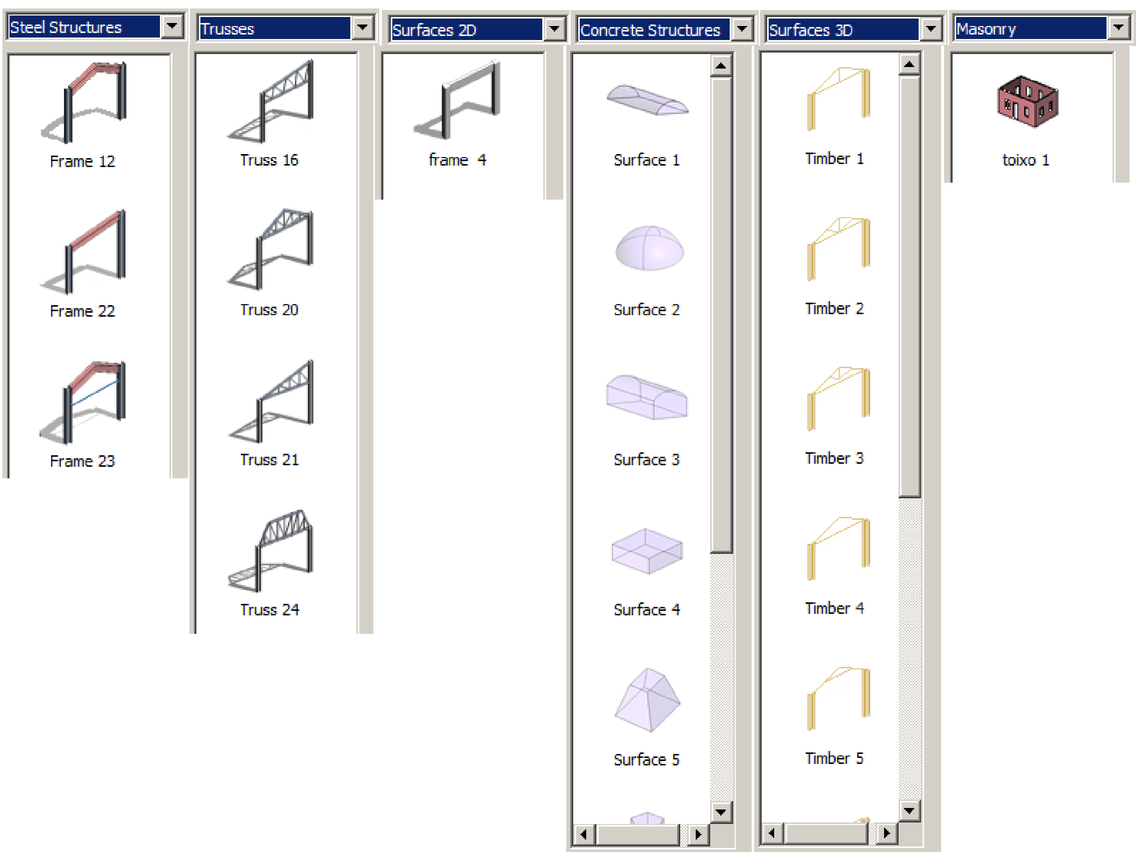
The templates’ dialog box opens automatically.



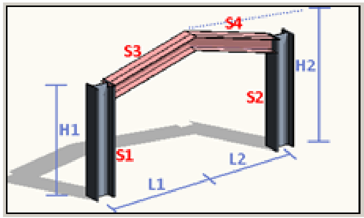
Select from the drop-down list the



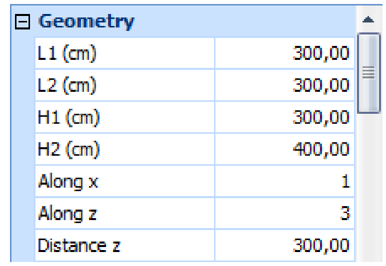
type of structure and the corresponding form.



6.2.1 Steel structures

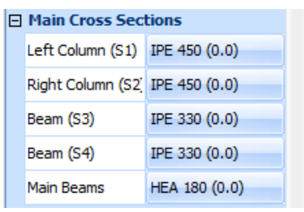


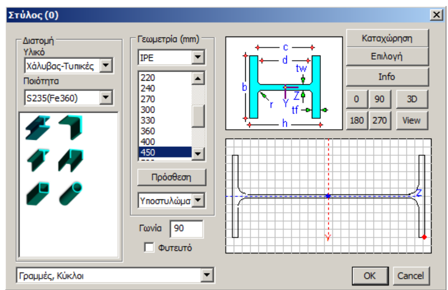
Set the parameters in the “Geometry” field according to the drawing and the repetitions in x and z direction.



The structural elements that will be part of the structure must have the corresponding checkbox active.

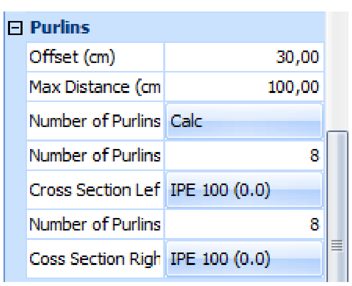
Select the corresponding cross section for each structural element in the field "Main Cross Sections".



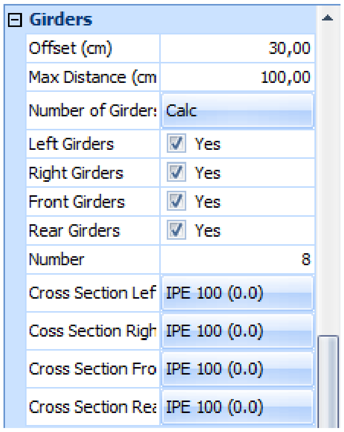


Click on the default section and in the dialog box select the desired one.

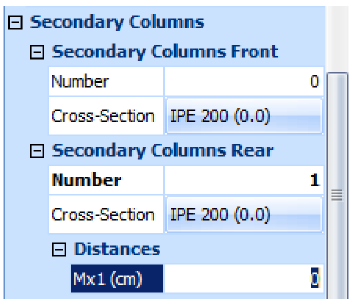
Everytime you modify the default section, remember to define the correct layer. The correct assignment of the layers is important so that you can take advantage of the program’s commands operating for each layer and thus saving a lot of time.



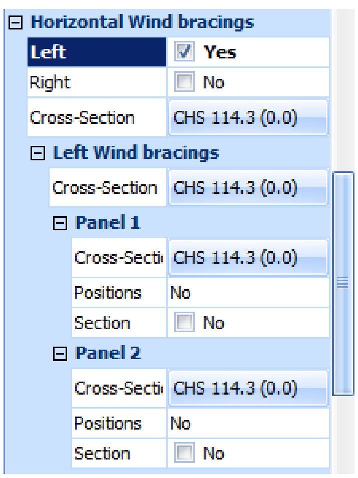
To calculate the number of purlins, type in the following fields:
“Offset”: the distance between the first purlin and the main beam
“Max distance”: the maximum distance between purlins. Then click the button "Calculate". The program automatically calculates the number of purlins per side. Alternatively, type the number of purlins on the left and right side, directly.



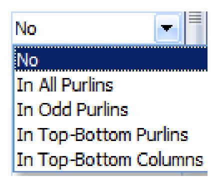
Calculate the number of girders as previously. Disable the direction without girders.
You can choose different cross sections for the different girders’ position.

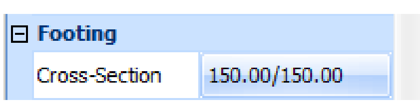


Type the number of the secondary columns (front and rear), define the cross section and type the relative distances.
For a number different than 0, the "Distances" field opens in which you can set the distance in cm.

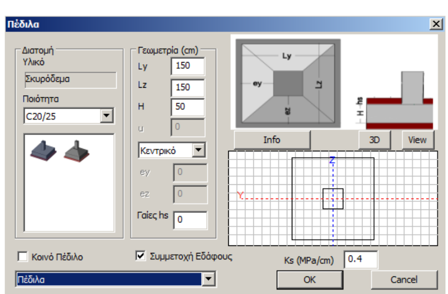


Activate the horizontal wind bracings of the left or/and the right side. The list expands further and for each panel, define the position and the intersection of the bracings.
The same procedure is followed for vertical wind bracings.





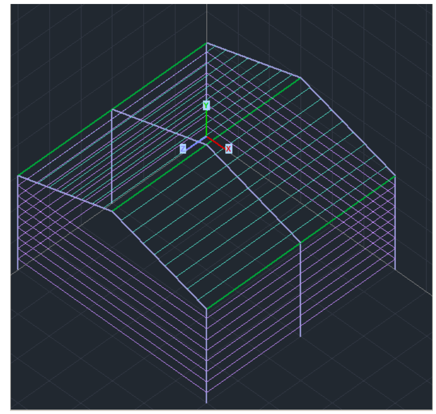
For footing, click on the default section to define the geometry, the coefficient for soil interaction and the corresponding layer.



Select the elements for display.

Display Structural elements	
All	☑ Yes
Secondary Colum	☑ Yes
Left Purlins	☑ Yes
Right Purlins	☑ Yes
Left Griders	☑ Yes
Right Girders	☑ Yes
Front Girders	☑ Yes
Rear Girders	☑ Yes
Left Hor.Wind bra	☑ Yes
Right Hor.Wind br	☑ Yes
Left Vert.Wind br	☑ Yes
Right Vert.Wind b	☑ Yes
Front Vert.Wind b	☑ Yes
Rear Vert.Wind b	☑ Yes

On the right, you can see the structure which is created. With the bar below you can change the view. You can also use left click to move and rotate the image.



Load Attribution	
Purlins	☑ Yes
Girders	☑ Yes

"Load Attribution" regards the wind and snow loads according to Eurocode 1 ("Loads>>Wind-Snow Loads"). When the fields "Purlins" and "Girders" are active, the program automatically allocates the wind and snow loads on them.

The command "Wind-Snow Loads" is presented analytically in the Chapter "Loads".
The command "Save" is used to save the template. You can create a folder and save all your templates in it and make your template library to use them later or perform changes and use them in other projects.

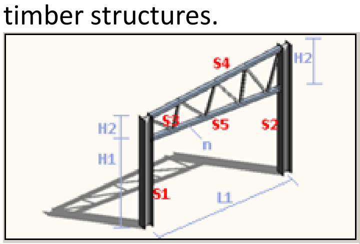
⚠ *It is now possible to preview the files you create and save in templates.*

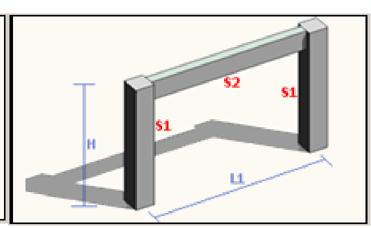
The button "OK" is used to open the template model in SCADA Pro interface in the virtual view. Switch off the virtual view to receive the 3D physical and mathematical model. Now you can work on the model using the appropriate tools (Chapter 2) and make all the changes you need to create the real model of your project.

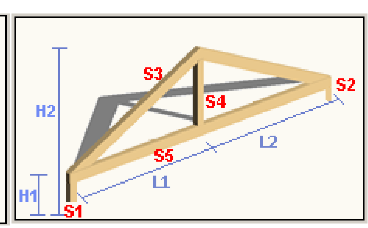
⚠ *You can also use more templates for the same project, with the same or not material. Just click on the input point, define the template, click "Save", "Ok" and repeat for the second template.*

6.2.2 Trusses, concrete, timber.

The same procedure, as previously described for the steel structures, is applied to concrete and timber structures.

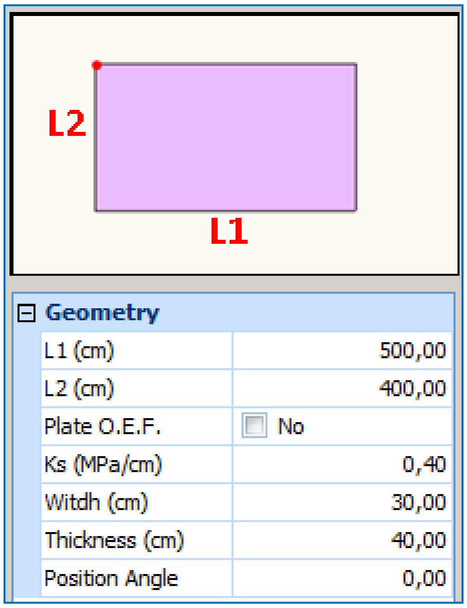






Define the geometry according to the drawing, the cross-sections, and the repetitions and press the button “OK” to open the template model in SCADA Pro interface.

6.2.3 Surface 2D



Choose one of the proposed 2D surfaces and define the geometric characteristics according to the drawing.

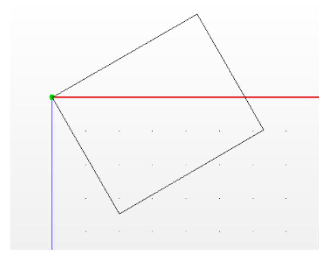
In case of Plate O.E.F. activate the corresponding checkbox and type a value for the soil constant Ks (MPa/cm).

The values in the fields “Width” and “Thickness” refer to the mesh that will be generated for the simulation of the surface. (Note: the default value for density is 0.15)*

“Position Angle”: In this field, type the angle (in degrees), versus the X, Z global axes, to define the direction of the surface in the desktop.

EXAMPLE:

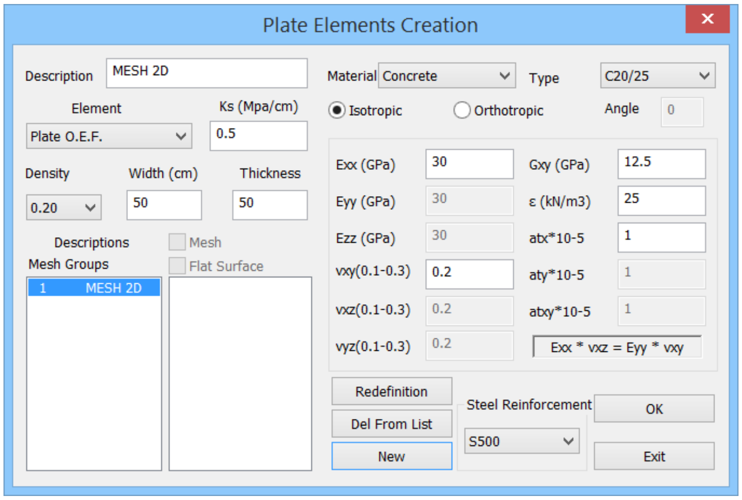
Type the angle of 30° to receive a rotation like the one on the right figure:



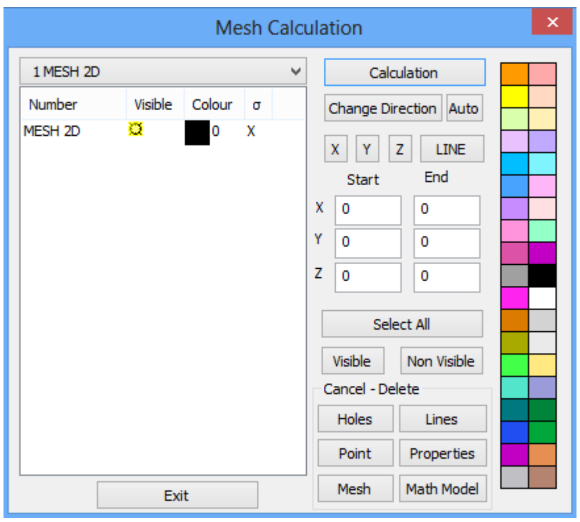
Click the button “OK” to import the defined surface in the interface.

On “Modeling” select “Mesh 2D”>>“Mesh”. In the dialog box, in the list “Mesh Groups”, the defined Plate is displayed. Select it in case you want to make changes (i.e. change density) and then click on “Redefinition”.

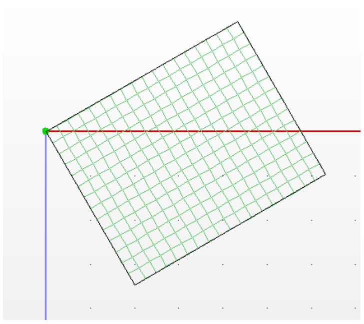
The command has been described thoroughly before.



If you don't need to make any change, go directly to "Modeling" >> "Mesh 2D">>"Calculate", and the following dialog box is displayed:

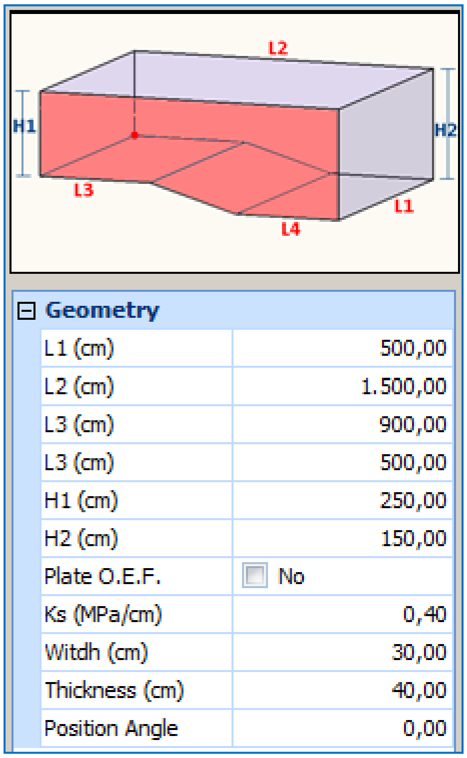


Select the Plate and then click the button "Calculation" for the mesh to be generated and to get the physical model of the surfaces.



Proceed to create the mathematical model by using the command "Tools" >> "Calculation", described in detail in Chapter 4, Unit "Tools".

6.2.4 Surfaces 3D



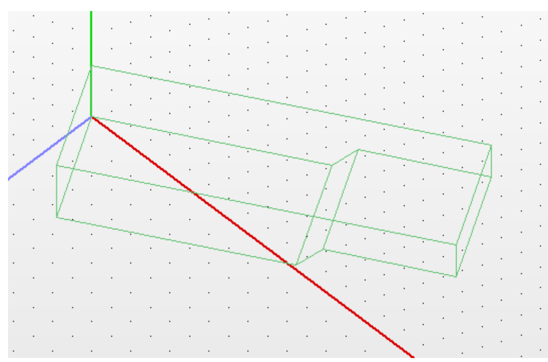
Geometry	
L1 (cm)	500,00
L2 (cm)	1.500,00
L3 (cm)	900,00
L3 (cm)	500,00
H1 (cm)	250,00
H2 (cm)	150,00
Plate O.E.F.	No
Ks (MPa/cm)	0,40
Witdh (cm)	30,00
Thickness (cm)	40,00
Position Angle	0,00

Choose one of the proposed 3D surfaces and define the geometric characteristics, based on the drawing. In case of Plate O.E.F., activate the corresponding checkbox **Plate O.E.F.** ☑ **Yes** and type a value for the soil constant Ks (MPa/cm).

The values in the fields “Width” and “Thickness” refer to the mesh that will be generated for the simulation of the surface.
(Note: the default value for density is 0,15)*

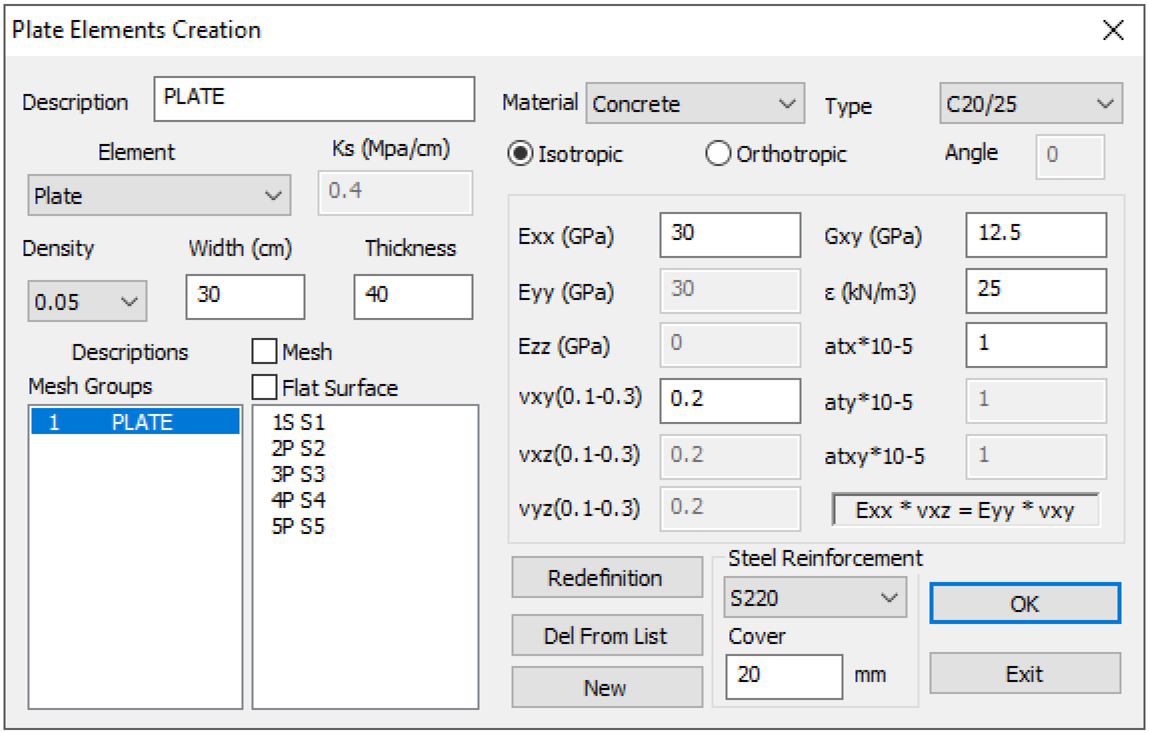
“Position Angle”: In this field type the angle (in degrees), versus the X, Z global axes, to define the direction of the surface in the interface.

EXAMPLE:
Type the angle 30° to get a rotated surface like the one presented in the figure below:

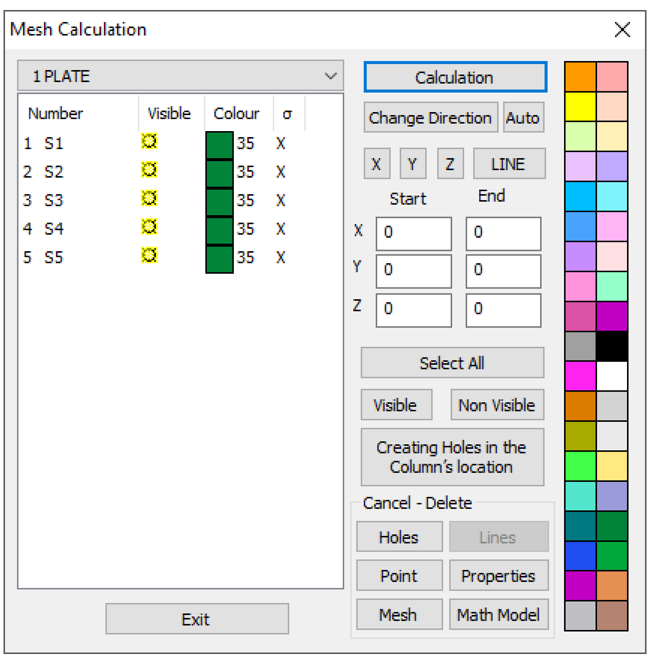


Click “OK” to import the defined surfaces in the interface.

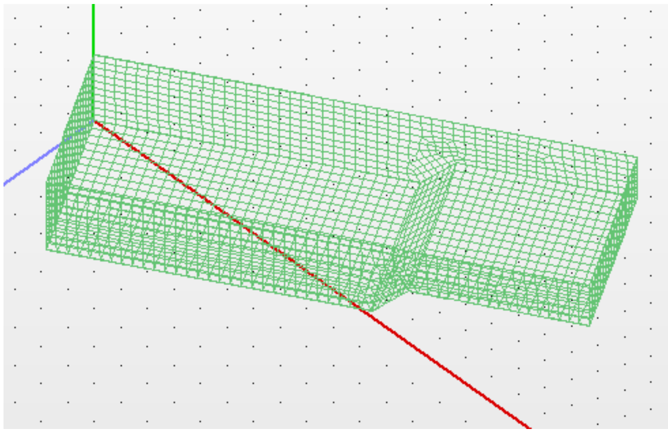
On the tab “Modeling”, select the group command “Surface Elements”>>”2D”>>“Mesh”. In the dialog box, an in the list “Mesh Groups”, the predefined mesh and the subgroups are displayed. In case you want to make changes, select the mesh and the subgroup, and then click “Redefinition”. (See more details on “Modeling”>>”Surface Elements”)



If you don’t need to make changes, go directly to “Modeling”>>“Surface elements”>> “3D”>>“Calculation”, and the following dialog box is displayed:

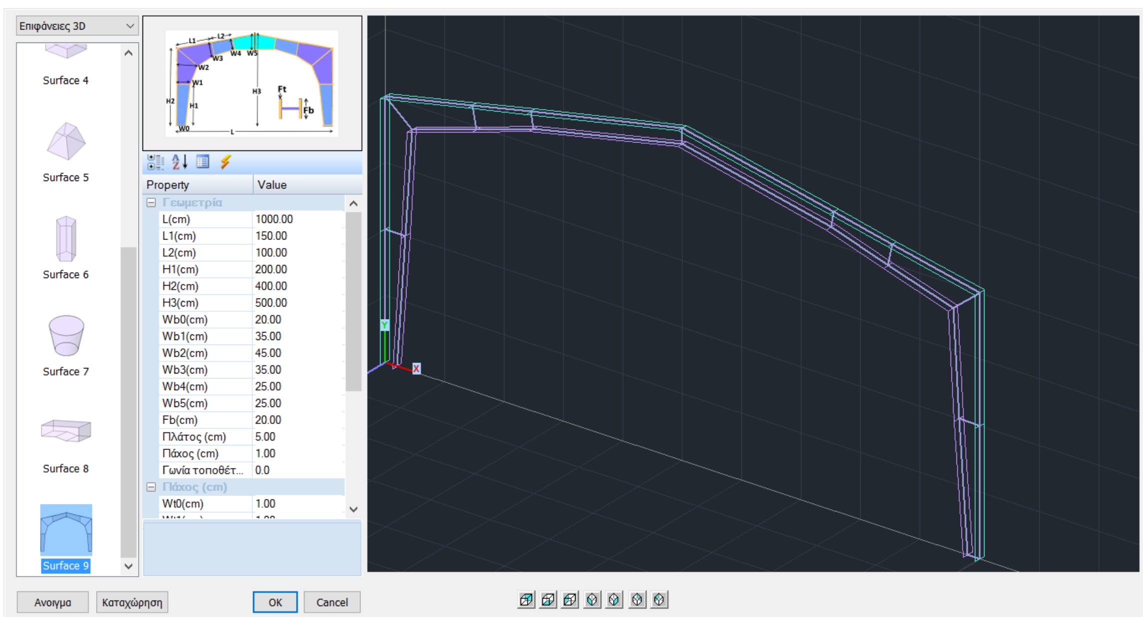


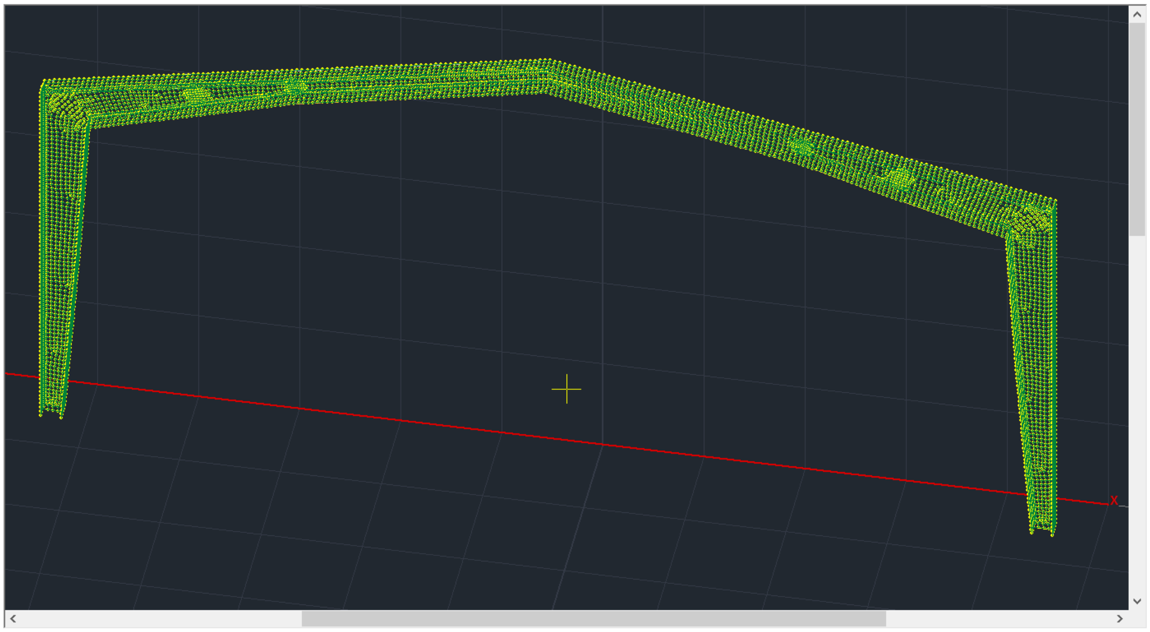
Select “Calculation” for the mesh generation and receive the following physical model of the surfaces.



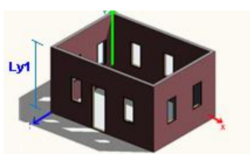
Proceed to create the mathematical model by using the command
“Tools”>>“Model”>>“Calculation”, described in detail in Chapter 4, Unit “Tools”.

*It is also possible to simulate automatically **typical metal variable-section** frame with finite surface elements, defining the geometry and the respective thicknesses.*

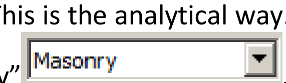




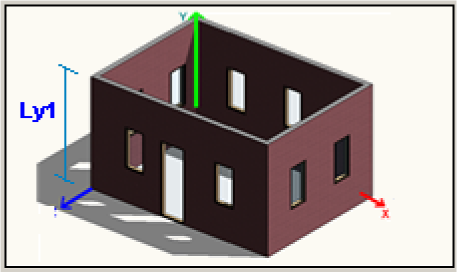
6.2.5 Masonry

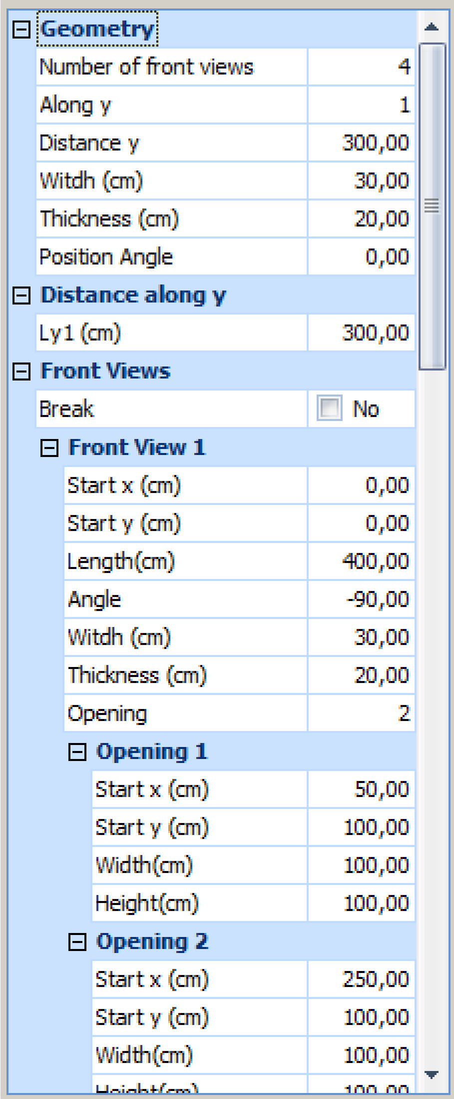


For masonry structures, the template tool can be used in two modes:

1st MODE: This is the analytical way. Select the insertion point and choose from the drop-down list "Masonry" 

.



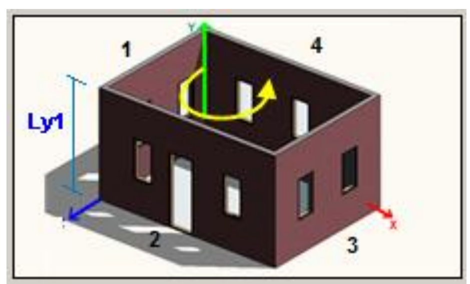


Define the geometry; the number of views, the repetitions on y direction (number of floors) and the distance between them (floor height). Type the values of the width, the thickness of the walls and the angle position according to X, Z global axes to define the direction of the surface in the interface.

If there are more than one floors, you can change the floor height in the field "Distance along Y".

The activation of the checkbox "Division", regarding the front views, is optional. With this command, each front view is slivered in more than one surfaces, with limits in the middle of the opening, so, each view is simulated from continuous surfaces without holes. Otherwise, in the simulation process, each view contains one surface with its existing holes.

For each view define: (i) the coordinates of the start point and the angle for the rotation of the structure according to X, Z global axes (see the drawing) counterclockwise, (ii) the length and the thickness of the wall and (iii) the number of the openings.
Similarly, define the geometry and the position of each opening.



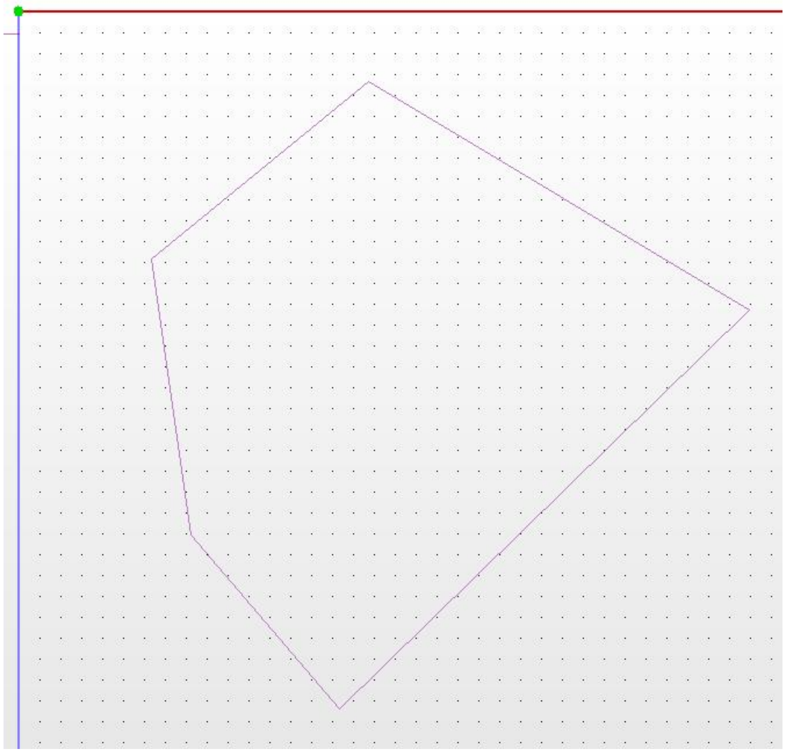
Click the button "OK" to import the defined structure in the interface.
Proceed to calculate the mesh, as described above.

2nd MODE: SCADA Pro gives you the opportunity to create a masonry structure on any external boundary, by using the tool "Templates", quickly and easily.

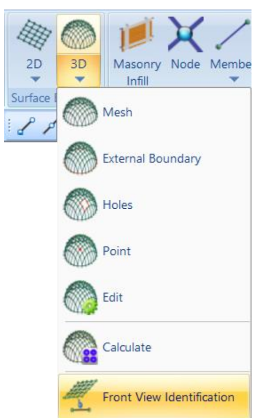
The process is the following:

(i) Enter a plan view in DXF or DWG file format by using the command group "Draft" in a closed surface on X, Z plan level.

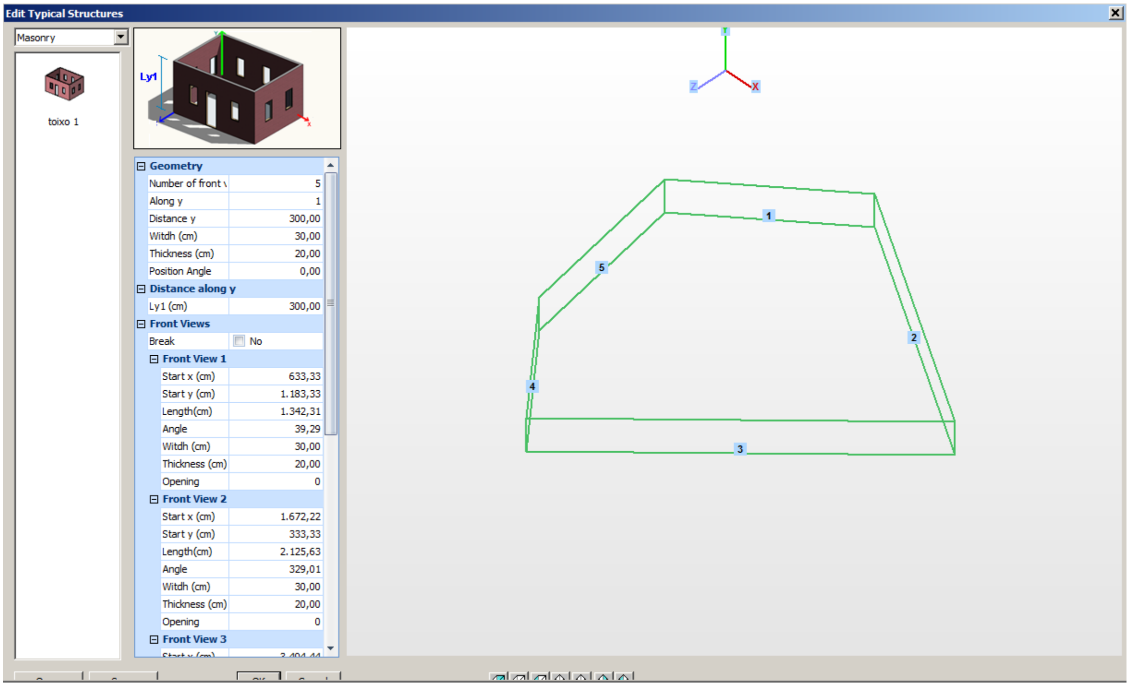
Unit: "**Basic**", command path: "Draft">>"Line">>"Polyline" → create a surface → right click.



(ii) Unit: "**Modeling**", command path: "Surface Elements">>" 3D">>"Front View Identification".



Then use the selection command "Window"  to select the total plan view. Right click and the masonry templates dialog box is displayed:



The program identifies automatically the geometry of the floor plan view. By default, the height is defined and the views are created according to the global axes.

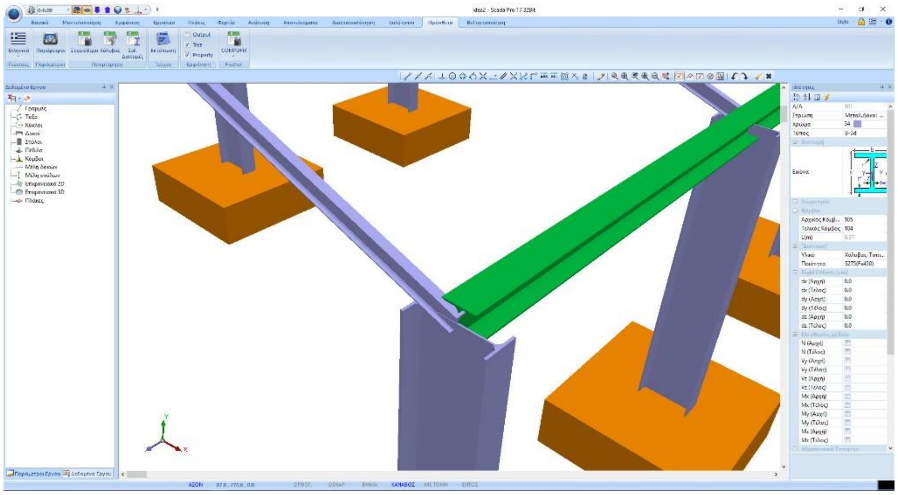
(iii) The user has to define the number of the floors and the corresponding heights, as well as the openings on each view by following the 1st MODE procedure.

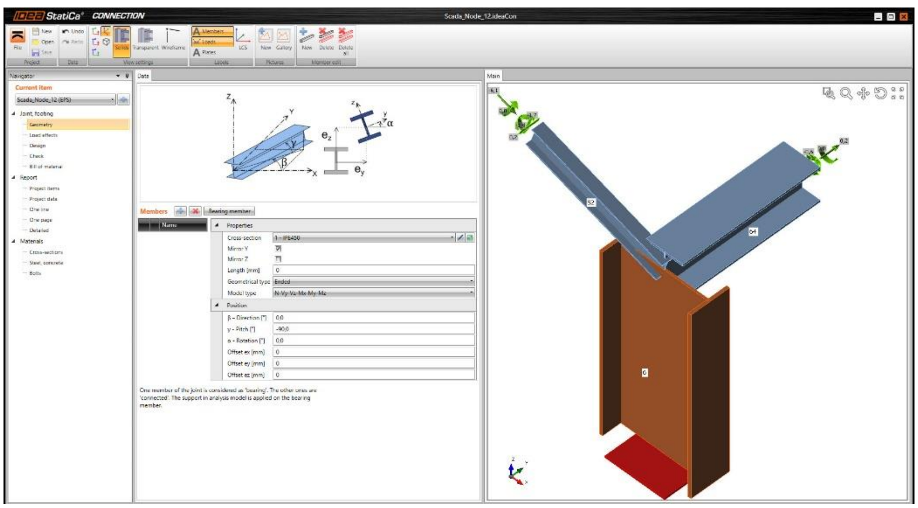
Since you have completed the process for each side and each opening, insert the project on the inteface by selecting the button “OK”.
Proceed to calculate the mesh, as described previously.

6.2.6 Steel Connections

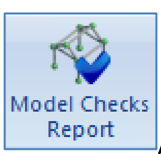
Regarding the steel connections, Templates tool can be used to export any steel connection from SCADA Pro to IDEA StatiCa's top IDEA Static design.

Export is achieved thanks to the BIM technology of the two applications and allows full and dynamic communication between them. Extract the full node topology, cross sections, material qualities as well as intensities per combination as they were derived from the analysis.





6.3 Model checks report



After the generation of both physical and mathematical model of the project, the program checks the model for possible errors and warnings with the selection of the command “Model Checks Report".

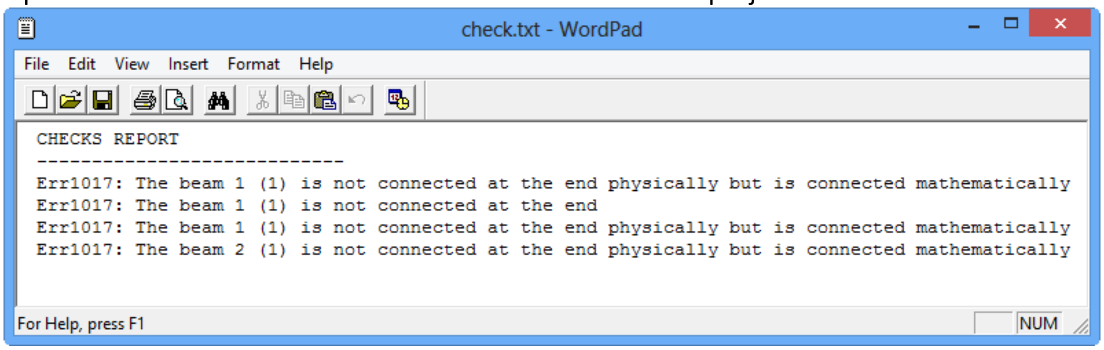
A TXT file appears in the screen that contains messages with possible errors related to the physical or mathematical model ("Err", number, message).

Take into consideration the messages’ content and make the appropriate changes, using the corresponding commands explained in detail in Chapter 4, Unit “Tools”.

⚠ *"Err" is not always an error indication, it could be just a warning. The user has to correct the errors and take into account the warnings.*

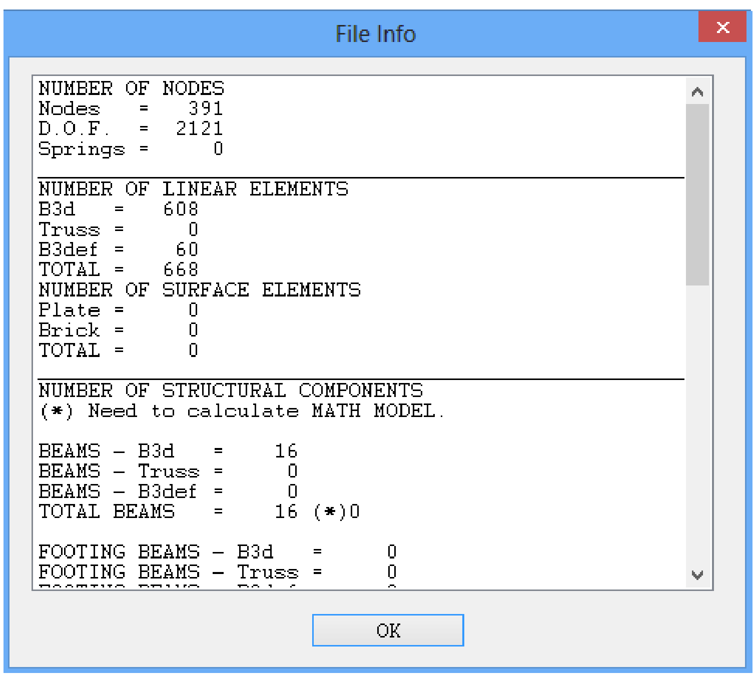
EXAMPLE:

In the example below, the Err1017 means "Beam on Beam" and it is a warning that doesn’t require correction. Close the window and continue with the project.

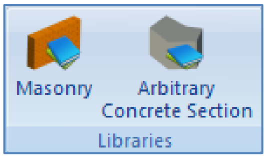


6.4 Model info

This command is used to receive a full display of the information related to the active project: number of nodes, members, components and the volume, the weight, etc.



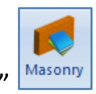
7. Libraries



The command group "Libraries" contains libraries of:
-Masonry and
-Arbitrary Concrete Sections.

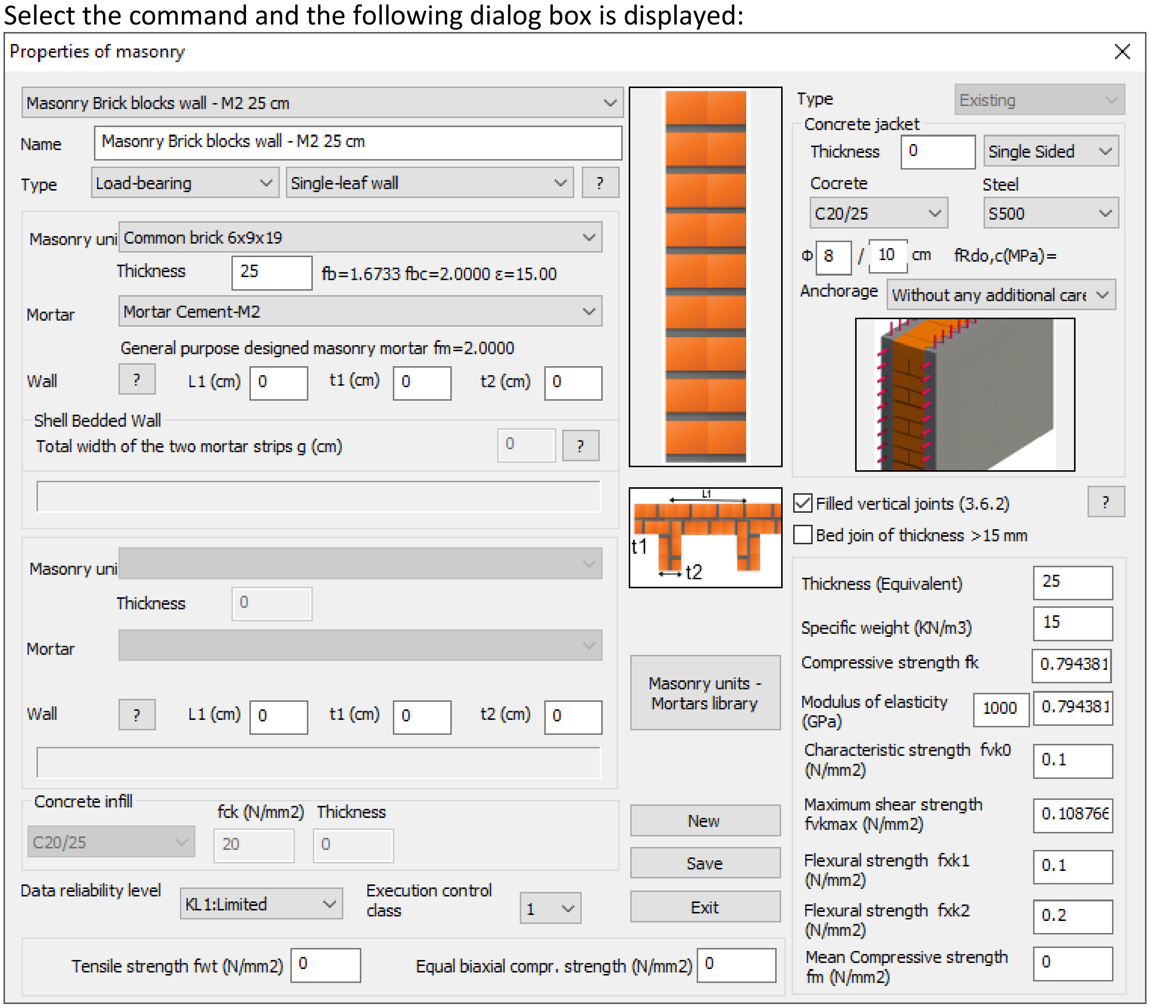
Libraries can be enriched by the user and be used in any other project.

7.1 Masonry

This command is used for the modeling of the masonry structures in conjunction with the commands "**Templates**" and "**Front View Identification**". The command "**Masonry**" 

is used for the definition of the properties of the masonry that you can also save in the library.

Select the command and the following dialog box is displayed:



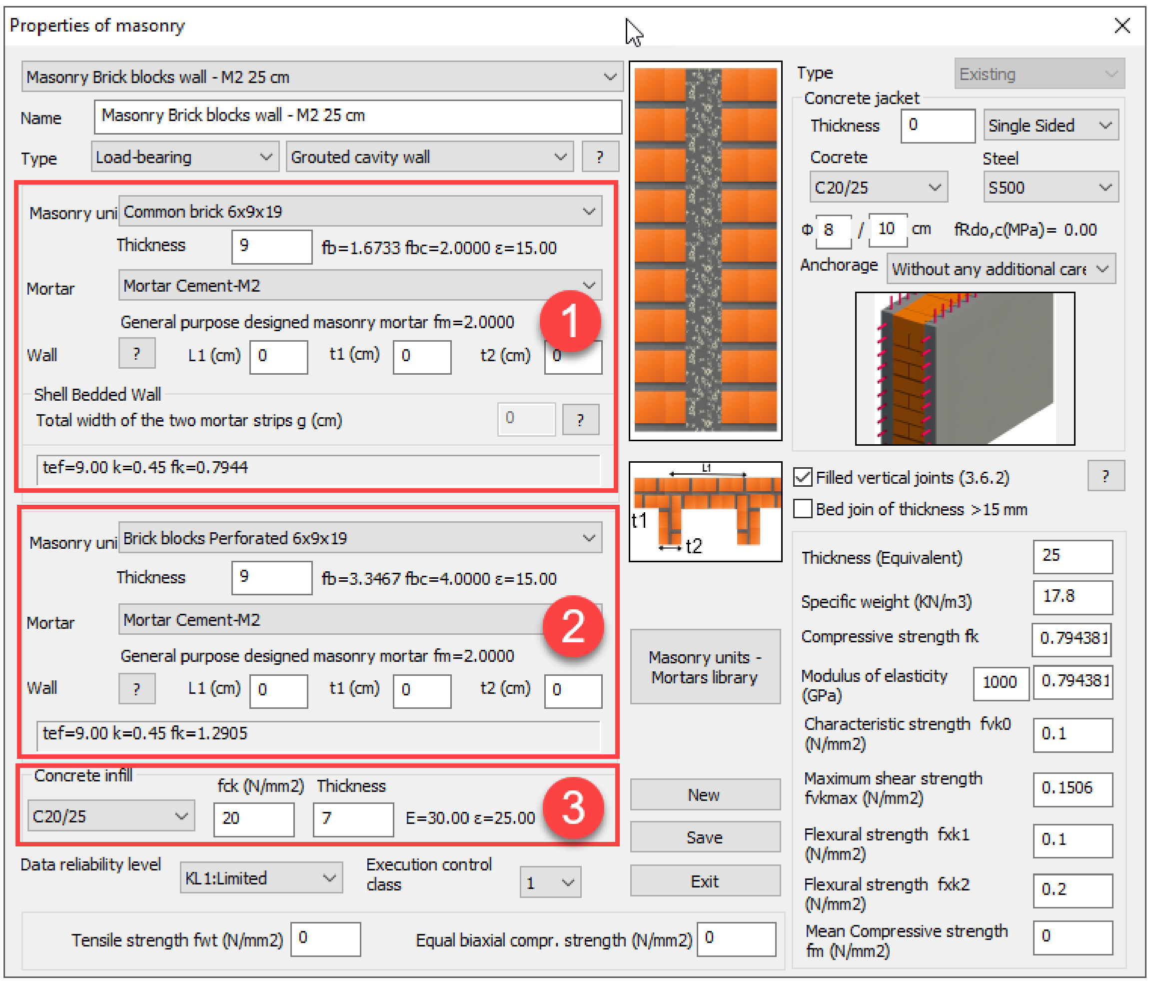
Choose a predefined wall, or create a new one. Type a name for the wall, select the "Type" from the drop-down list and define the related properties for the "Masonry Unit", "Mortar", "Piers", "Concrete Infill" and "Concrete Jacket".

⚠ *Depending on the selected TYPE of masonry, in the dialog box, some fields are enabled or disabled.*

Name: Wall 1

Type: Grouted Cavity Wall

⚠ All fields of the window are active since this type requires the definition of two single walls and a concrete infill.

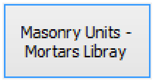


In *Wall1* & *Wall2* define

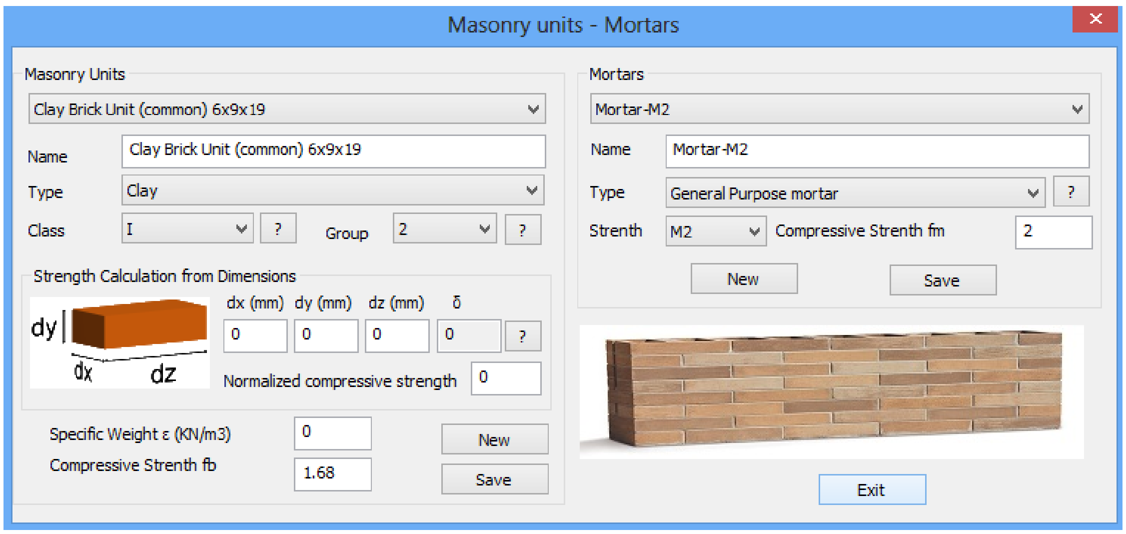
units: the type and thickness

Mortars: the type and the corresponding factors are updated automatically.

fb=3.3467 fbc=4.0000 ε=15.00

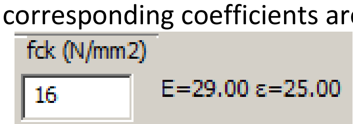


In the command "***Masonry Units – Mortars Library***" you will find standard typologies of clay bricks, mortar, and masonry. You can enter other bricks and mortar, by simply typing the name and specifying the class and group, for the compressive strength (which is updated automatically). Then select the button "New".
You can also change the class and group of an existing masonry or mortar and update it by clicking "Submit".
In the field "Masonry Units", select from the drop-down lists the type of bricks and mortar, and create a new type of masonry by clicking "New". The weight and strength are calculated automatically.



The selection of the command "*Piers*" affects the stiffness and the effective thickness of the wall.

In the field "*Concrete Infill*", choose the type of the concrete and set the thickness of the layer. The corresponding coefficients are updated, automatically.

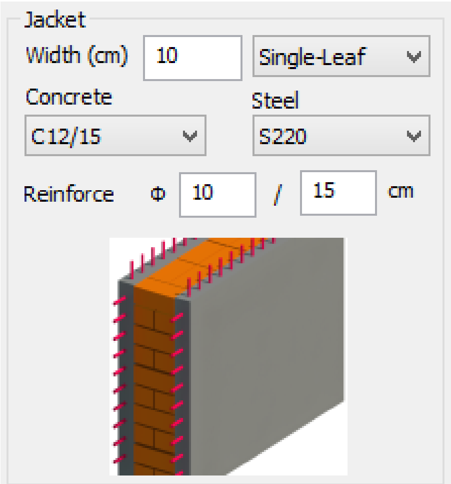


.

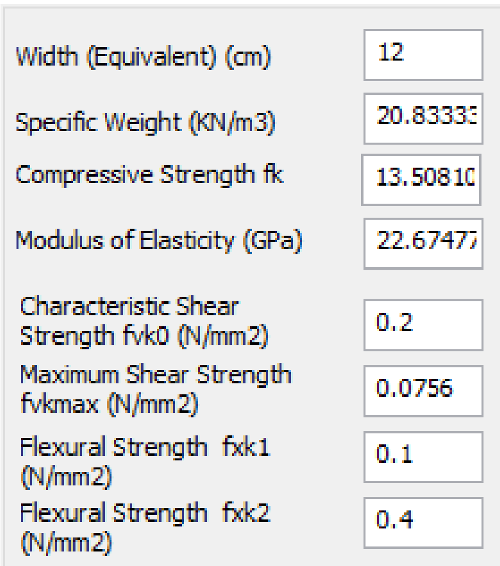
The selection of the wall type "*Shell Bedded Wall*" affects the characteristic compressive strength of the masonry.

⚠ *The calculation of the characteristic shear strength of masonry based on the formula (3.5) implies that the joints meet the design requirements. In this case, enable the corresponding checkbox* 

in order the formula (3.5) to be used for the calculation.



In case you need to use the concrete **jacket** in masonry, define the geometric characteristics, the type of materials and the steel reinforcement.



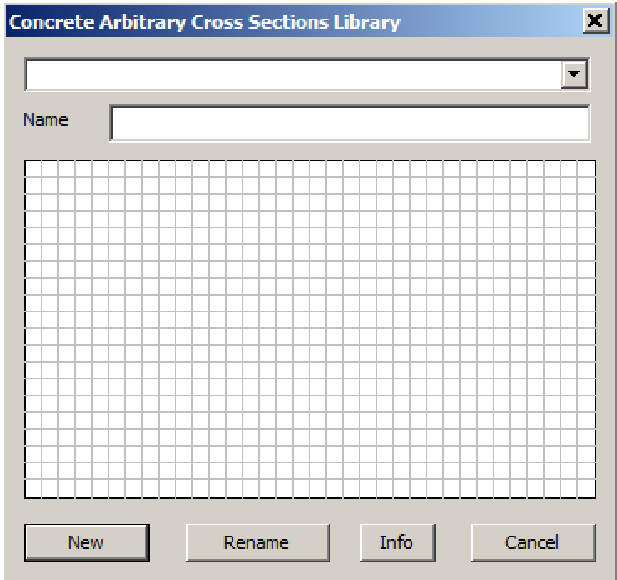
If the user knows the values for the **equivalent wall**, these can be defined manually.

The total masonry results are calculated by the program based on the input data and they are transferred to the summary table.

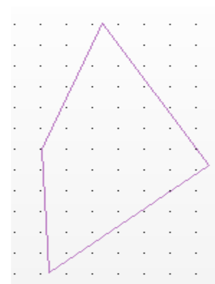
7.2 Arbitrary concrete section

You can create any arbitrary column cross section simply by setting the contour. The center of gravity and all inertial data are automatically calculated by the method of boundary elements. The section is saved automatically in your library.

The first time you select the command "**Arbitrary Concrete Section**", a blank dialog box appears:



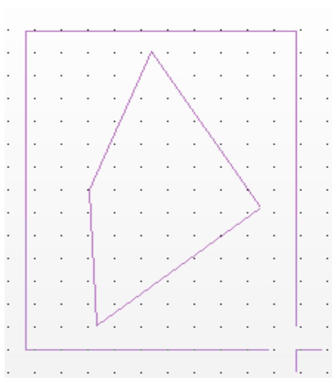
To insert your sections in the library and pick them whenever you want, proceed as follows:



- From the command group, "Basic" select a command to draw the closed contour of the arbitrary cross section (see Chapter 2, paragraph 1). Alternatively, you can enter a DWG or DXF file with the shape of the arbitrary cross section.

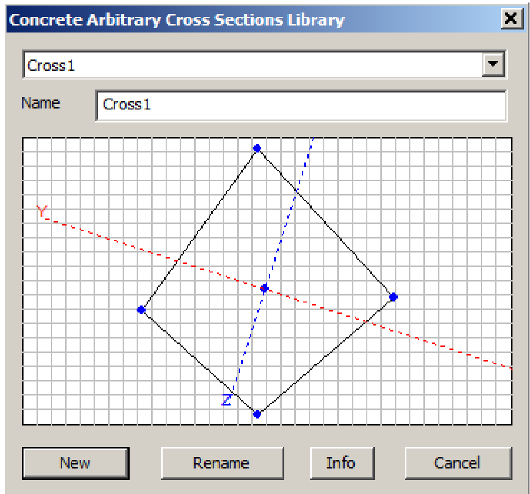
- Select the command "Arbitrary Concrete Section" and type a name in the dialog box (at least three characters) and then press "New".

- Activate the selection command "Window" . Left click and pull the window to include the total shape. Left-click again and the figure turns to the dash. Right click to complete.



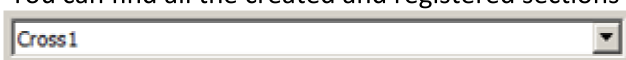
.

Select again the command "Arbitrary Concrete Section" and in the dialog box, the cross section appears, with the input points and the local axes. To change the name, type the new one in the field and click "Rename".



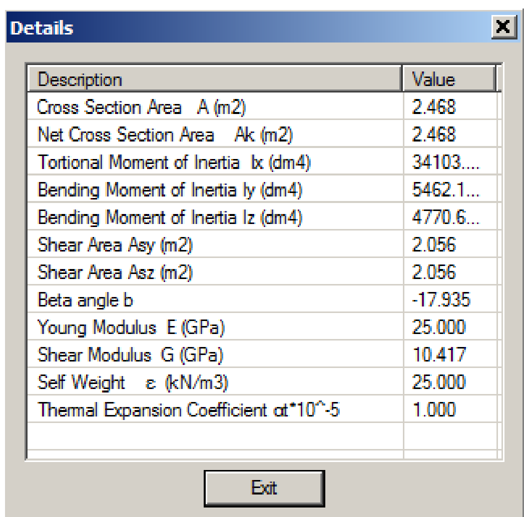
NOTE:

You can find all the created and registered sections in the list:



Click the button “Info” 

to read all the geometric and inertial characteristics of the section.



Description	Value
Cross Section Area A (m2)	2.468
Net Cross Section Area Ak (m2)	2.468
Tortional Moment of Inertia Ix (dm4)	34103....
Bending Moment of Inertia Iy (dm4)	5462.1...
Bending Moment of Inertia Iz (dm4)	4770.6...
Shear Area Asy (m2)	2.056
Shear Area Asz (m2)	2.056
Beta angle b	-17.935
Young Modulus E (GPa)	25.000
Shear Modulus G (GPa)	10.417
Self Weight ε (kN/m3)	25.000
Thermal Expansion Coefficient αt*10^-5	1.000

To enter an arbitrary cross-section column in the model, open the command “Columns” and find it in the section list.