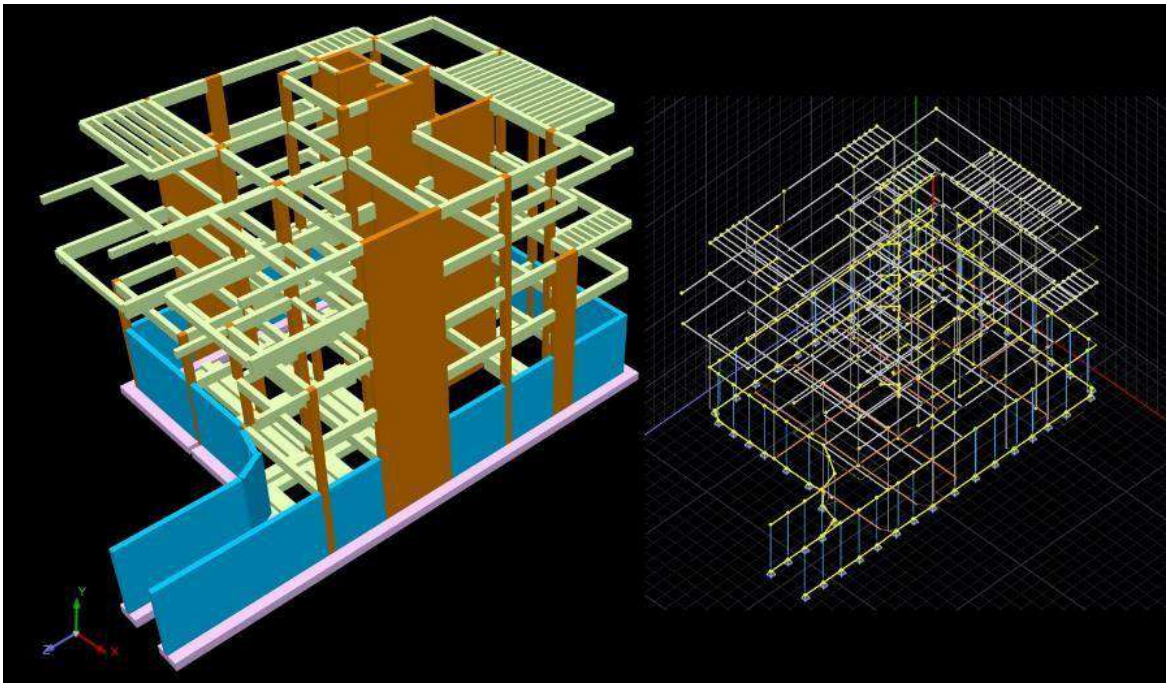
SCADA Pro 25™

Structural Analysis & Design

User Manual

2. MODELLING

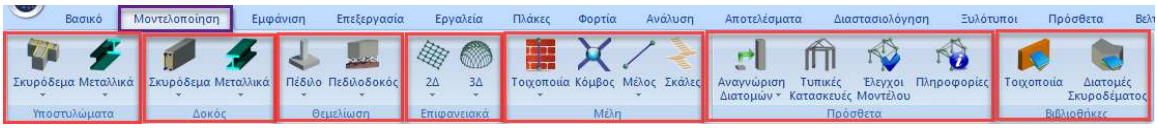


ACE HELLAS

INTEGRATED SOLUTIONS

CONTENTS

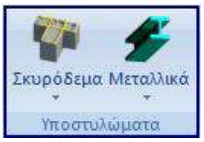
Chapter 2:
Modelling

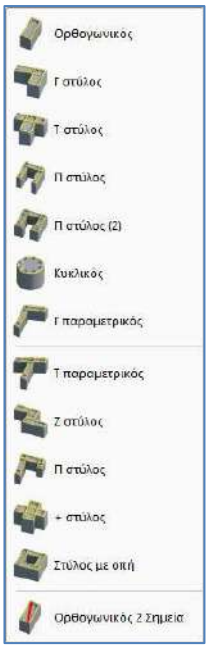


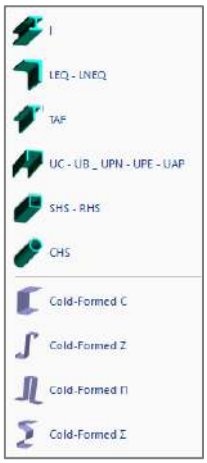
The 2nd Module is called "MODIFICATION" and includes the following 7 groups of commands:

√ Pillars
√ Beams
√ Foundation
√ Surface
√ Members
√ Parameters
√ Libraries

1. Pillars







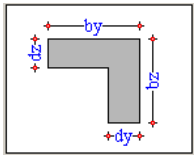
The "Columns" command group contains the commands to model the physical cross-sections of the columns.
Within the library of cross-sections you can select pole cross-sections from:

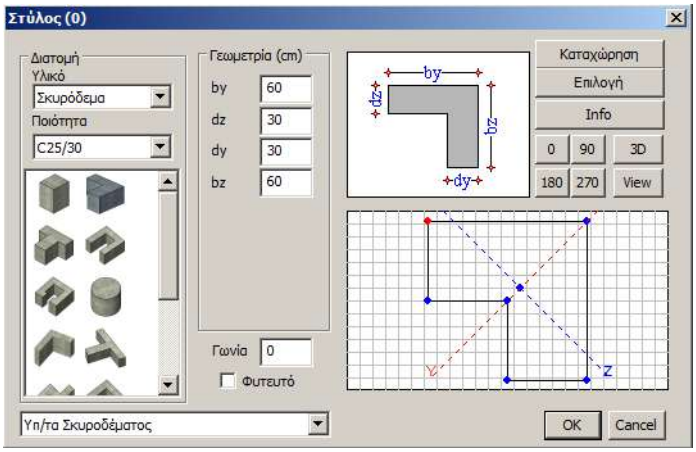
- **Concrete**
- **Metal (Hot & Cold Rolled)** and
- **Wood**

Each command includes the corresponding sub-commands that define the type and shape of the cross-section.

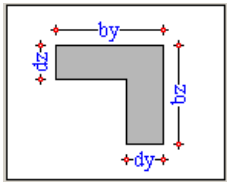
Set the **parameters of** the column. More specifically:

1.1 Concrete columns

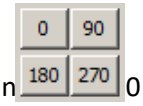




- "**Cross-section**": select the type of material (Concrete/Steel/Steel/Steel) and the quality. Then, from the standard cross-sections displayed according to the type of material, select the type of cross-section
- "**Geometry**": enter the dimensions of the cross-section based on the graph showing the shape of the column.

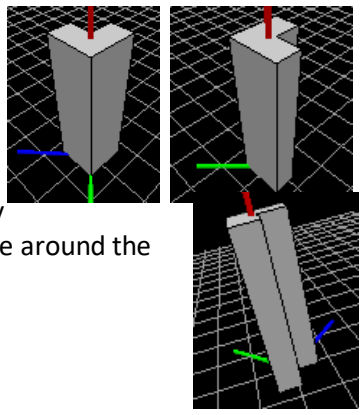


The four buttons below the shape of the support define the angle of the support in degrees. Depending on the selection the support is rotated by 90º, 180º, 270º.

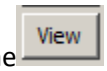


- "**Angle**": enter the value of the angle of rotation of the column for angles other than 0 or 270°.
- The button 

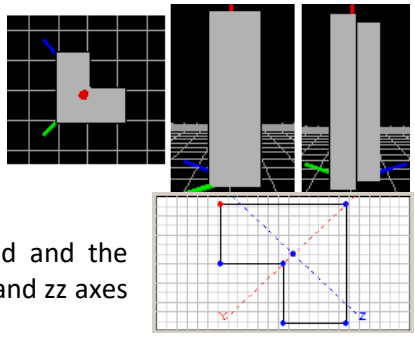
: offers a 3D representation of the pole with the specific cross-section and angle, which you can modify by selecting a different angle by pressing the buttons with the angles.



Pressing and holding down the left key and left key by moving the mouse, the plane rotates and you can see the pole around the perimeter.

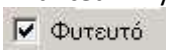
- By pressing the 

button in succession: you can see the views of the pole with the specific cross-section

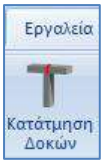


The cross-section of the column is formed in the window which is shaped based on the geometry you have selected and the dimensions you have entered. Here you also see the local yy and zz axes of the cross-section and you can select the insertion point (blue points). The selected insertion point is red.

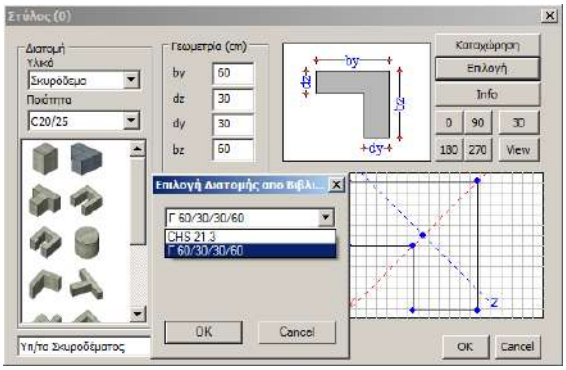
PLANTED PILLAR

- "**Planted**": if you wish to install a planted substructure check the option 

and insert the cross-section at the first (lower) level, on the beam where "planted", while in the next level up (and in all subsequent levels) you insert it on the corresponding beams, but <u>without</u> checking the "Planted" option.
To connect to the beam on which it "clicks", select (at each level): "**Tools**>> **Beam Partitioning**" and left click on the beam (at each level).

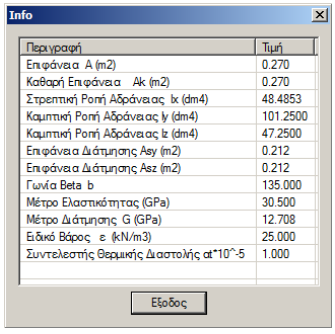


- "**Register - Select**": the user can create his own cross-section library, through the "Register" command, and call it at any time through the "Select" command, without having to define the same cross-sections each time.



- "**Info**" (Details) 

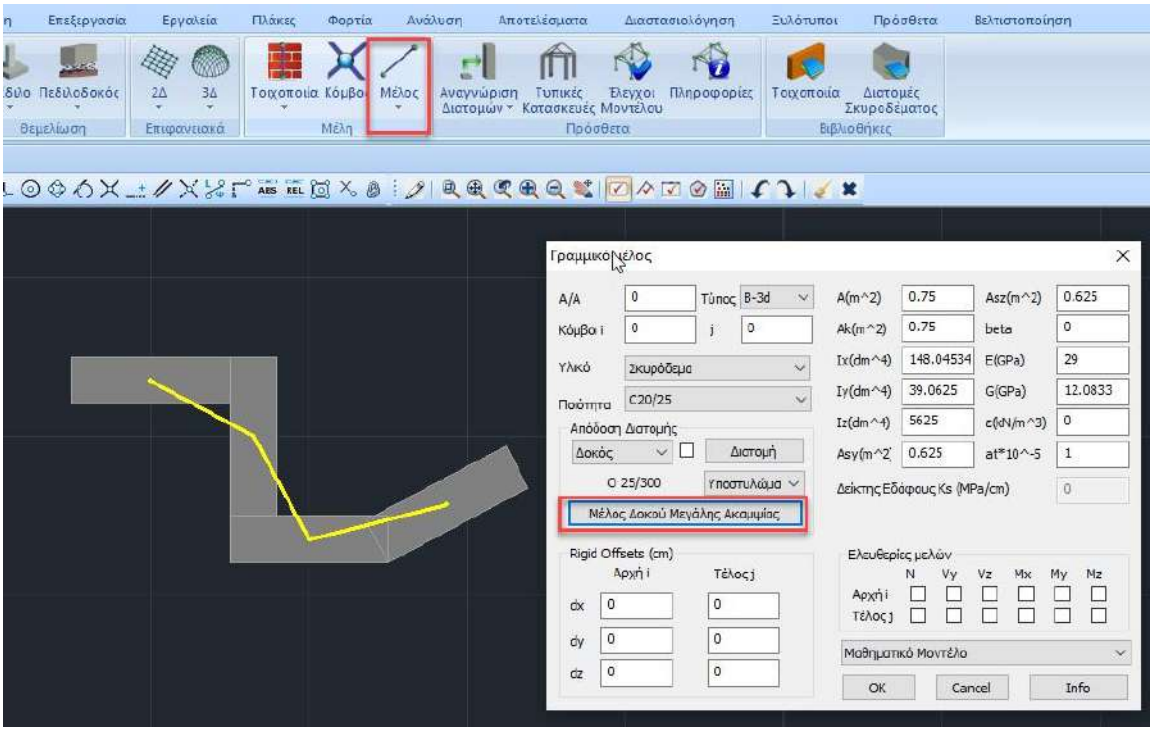
: select to see all the geometric and inertial data of the specific cross-section.



The list of concrete cross-sections includes <u>standard</u>, <u>parametric</u> and <u>random</u> cross-sections. To define a Random Cross-section use the Cross-sections command Concrete in the Library field of the Modeling Module, which is described in detail in relevant chapter.

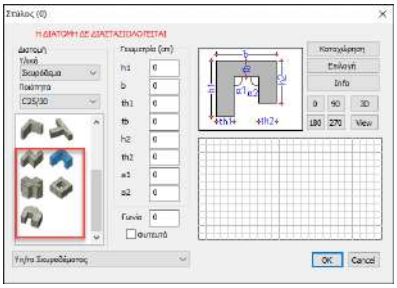
OBSERVATION:

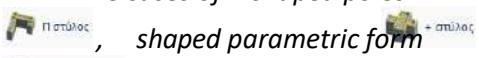
In addition, you can compose pole sections and create composite sections, which you can then connect with high stiffness bars.



ATTENTION:

The parametric cross-sections C & T ***are automatically dimensioned*** *by the program.*



The cases of Z-shaped poles  *, of cross-shaped parametric form*  *and pillar with a hole*  ***TE are dimensioned by the program*** *and in the upper part of the window is highlighted with the* Η ΔΙΑΤΟΜΗ ΔΕ ΔΙΑΣΤΑΣΙΟΛΟΓΕΙΤΑΙ *indication*

KEYBOARD COMMANDS:

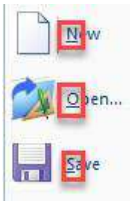
When entering physical cross-sections of columns you can use some keys on the computer keyboard as an aid. In particular :

TAB: Rotation every 30°

CTRL:Rotate in 0°,90°,180°,270°

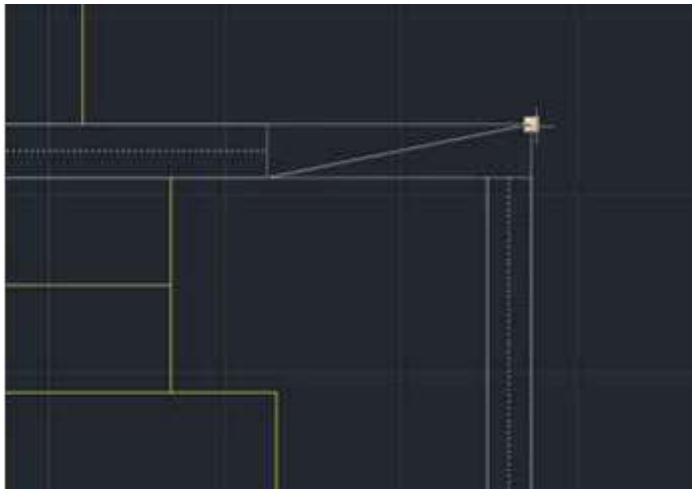
SHIFT: Rotation every 1°

`~ : Selecting the next pole top for placement The three basic commands, New, Open and Save can now also be activated with the keyboard shortcuts CTRL+N, CTRL+0, CTRL+S respectively.



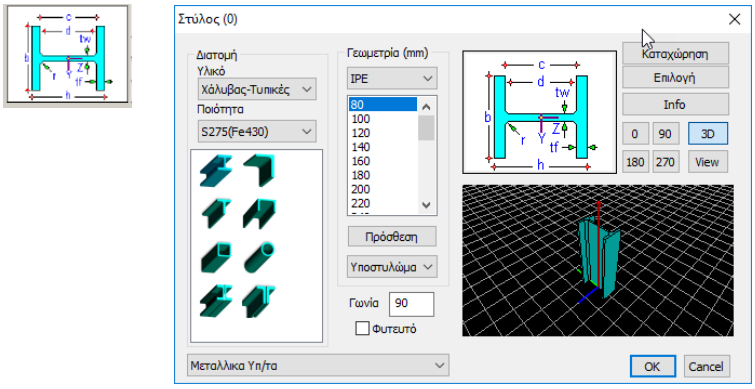
New method of inserting rectangular columns with 2 points.

In the new version of the program a new command has been integrated which allows you to insert a rectangular column or wall with the definition of 2 antidiametric vertices, a very useful command in cases where you have a design background with the positions of the columns



1.2 Metal supports

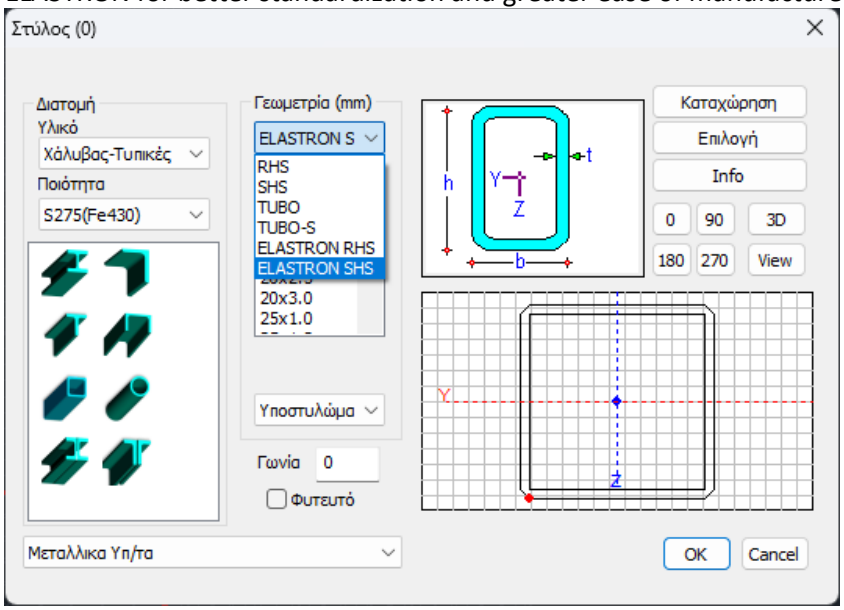
Similarly, for metallic sections, you define the material, the type of section and the angle. "Geometry" includes the multitude of commercial cross-sections, both Hot Cold Rolled.



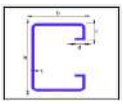
The 3D view helps to select the correct angle with respect to the local axes of the cross-section, and "Register" helps to create a library of cross-sections.

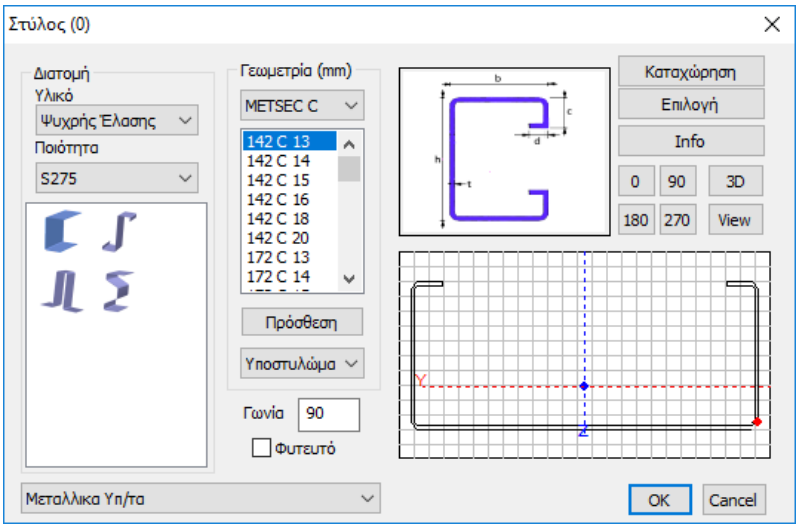
Cross sections of the company ELASTRON

The new version SCADA Pro incorporates 949 new RHS and SHS cavity cross-sections from ELASTRON for better standardization and greater ease of manufacture.



Cold rolled elements are inserted in a similar way to hot rolled elements. For both beams and columns, the cross-sections have been placed in a new category called 'Cold rolled'.





In the window that appears we select:

- **Type of cross-section**
- **Company**
- **Cross section**
- **Material**
- **Quality**

KEYBOARD COMMANDS:

When deriving physical cross-sections of columns you can use some keys on the computer keyboard as an aid. In particular :

TAB: Rotation every 30°

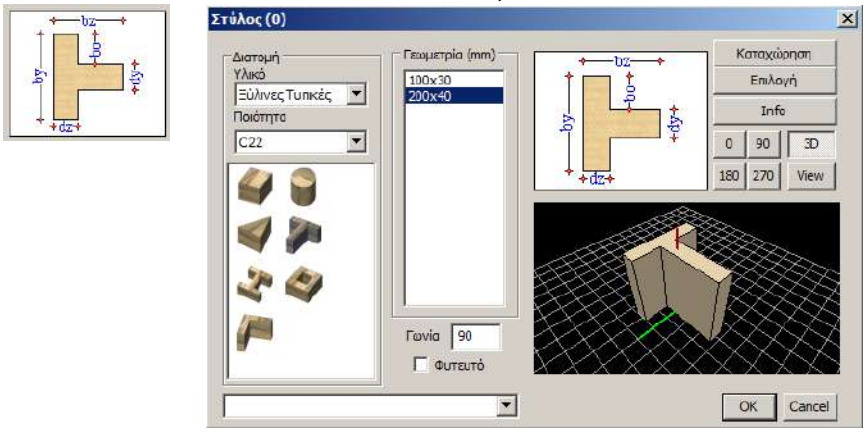
CTRL:Rotate in 0°,90°,180°,270°

SHIFT: Rotation every 1°

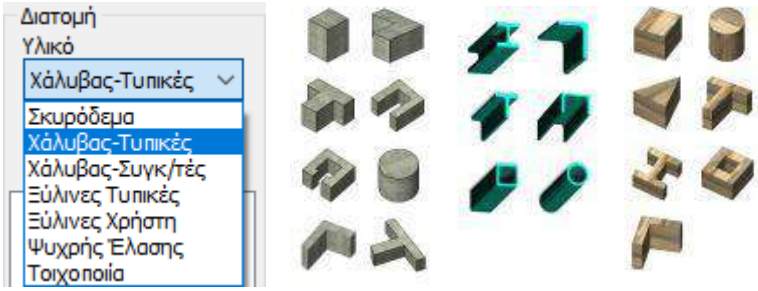
`~ : Selection of the next pole top for installation

1.3 Wooden pillars

SCADA Pro also features Wooden Sections, located inside the windows of the poles.



Regardless of the cross-section you select, the dialog box includes all other cross-sections. Based on the material you select, the grades, cross-sections and geometry are adjusted.



You can also model wooden sections, following exactly the same procedure as described for concrete and metal sections.
With SCADA Pro you can model and analyze concrete, metal, wood and load-bearing masonry structures.

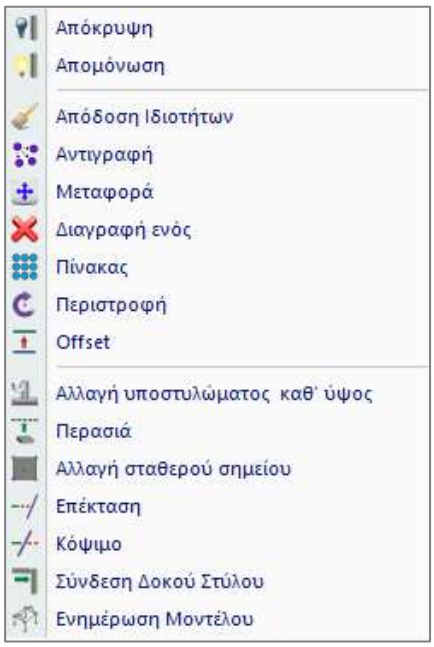
KEYBOARD COMMANDS:
When deriving physical cross-sections of columns you can use some keys on the computer keyboard as an aid. In particular :

TAB: Rotation every 30°
CTRL:Rotate in 0°,90°,180°,270°
SHIFT: Rotation every 1°
`~ : Selection of the next pole top for installation

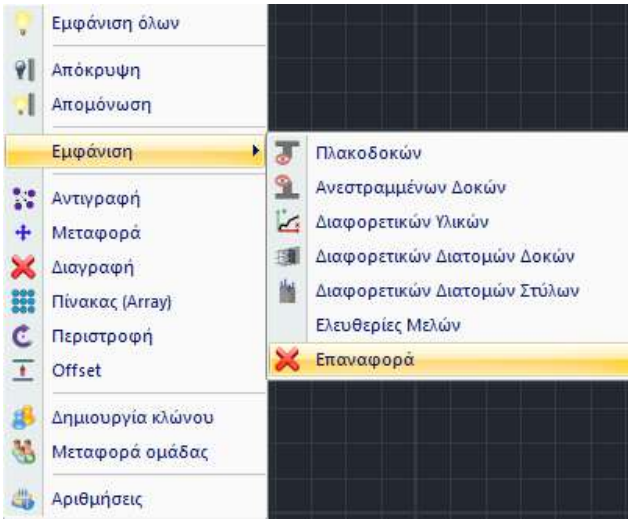
RIGHT MOUSE BUTTON

In each section of the program and for each item you approach with your mouse, right-clicking opens a list of commands related to the section and item.

Having entered one or more poles, at one or more levels, moving your mouse over a pole cross-section and pressing the right key opens the relevant command list:



The list includes commands found in the corresponding section that apply to the item that is right-clicked on.
But there are some commands that are only found here, such as the commands:
"**Hide**": to hide an item.
"**Isolation**": to isolate one element, hiding all the others.

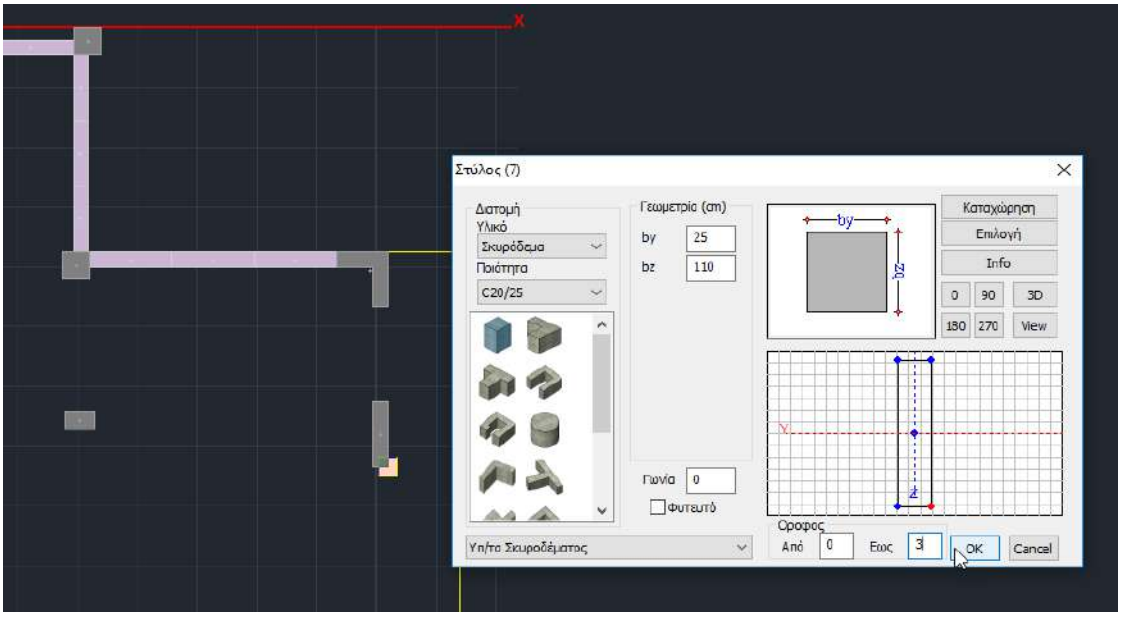


To deselect, press the right mouse button anywhere on the canvas surface.
The list opens a new list and you select "Show All".

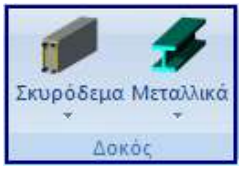
In addition, with the "Show" command you can display the different cross-sections of the poles in different colours and in the same colour the same cross-sections between them, for a better overview of the elements of the structure.

"**Change height**": to modify the cross-section of a post in height. Approach the end or the side of the cross-section of the post at the lowest level to which the modification is to be made, right-click and "Change height of sub-post".
The window of the cross-sections of the columns opens, where you can modify the cross-section of the element, and at the same time select the floors where this modification will be applied.

- *Be sure to select for the change, the cross-section at the lowest level, and within the window to "blush" the correct fixed point.*



2. Beams



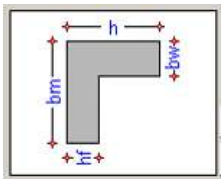
H team command team "Beams" includes the commands for to model beams from:

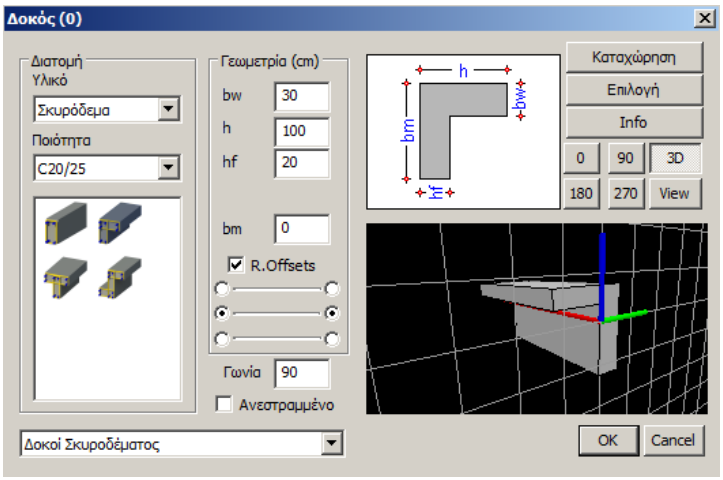
- **Concrete**
- **Metal (Hot & Cold Rolled)** and
- **Wood**

Each command includes the corresponding sub-commands that define the type and shape of the cross-section.

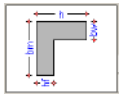
Define **the parameters of** the beam. More specifically:

2.1 Concrete beams

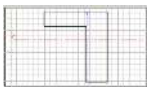




- "**Cross-section**": select the type of material (Concrete / Steel / Wood) and the quality. Then, from the standard cross-sections displayed according to the type of material, select the type of cross-section
- "**Geometry**": enter the dimensions of the cross-section based on the graph showing the shape of the beam.

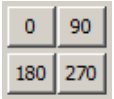


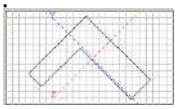
In the window, the cross-section of the beam is formed based on the geometry you have selected and the dimensions you have entered. Here you can also see the local yy and zz axes of the cross-section



The four buttons set the angle of the beam in degrees, changing its orientation by 90°, 180° and 270°.

- "**Angle**": enter the value of the angle of rotation of the beam for angles other 0, 90, 180 or 270°. The angle of rotation refers to the local axis xx of the beam angle Beta). For example, if the angle of rotation is 45º, the beam is placed based on the figure:

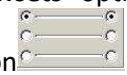




- The button 

: offers a 3D representation of the beam, with the specific cross-section and angle, which you can modify by selecting a different angle by pressing the buttons with the angles. By pressing and holding the left key and moving the mouse, the plane rotates and you can see the beam around the perimeter ,.

- The "**R.Offsets**" option enables or disables the existence of rigid sections in the beams.

The option  indicates the way of placing the beam, i.e. which passage will be chosen for its design.

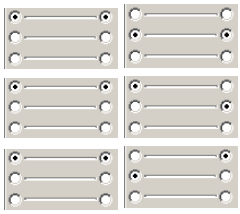
KEYBOARD COMMANDS:

The positioning of the beam can also be adjusted via the keyboard.

TAB to change the beginning and end of the beam

SHIFT to change the beginning of the beam's passage

CTRL to change the end of beam pass



Select the cross-section of the beam and the insertion point (start) by left-clicking on the desktop. Before selecting the end point of the beam, press TAB, SHIFT, or CTRL, depending on how you want to position the beam.

- With the "**Inverted**" option you indicate whether the beam to be placed is inverted.

From the list of predefined layers
you can select the layer to which the beam to be inserte[illegible] layer "Concrete Beams".

- "**Register - Select**": the user can create his own cross-section library, through the "Register" command, and call it at any time through the "Select" command, without having to define the same cross-sections each time.

- "**Info**" (Details) 

: select to see all the geometric and inertial data of the specific cross-section.

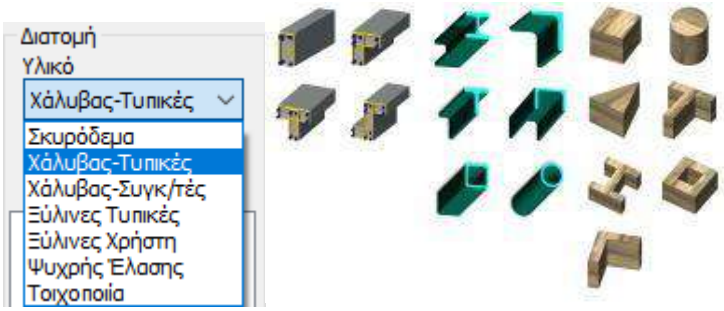
2.2 Metal beams


Similarly, for the metal sections (Hot & Cold Rolled), you define the material, the type of section and the angle.
The 3D view helps to select the correct angle with respect to the local axes of the cross-section, and "Register" helps to create a library of cross-sections.

2.3 Wooden beams


SCADA Pro also features Wooden Sections, located inside the windows of the beams.

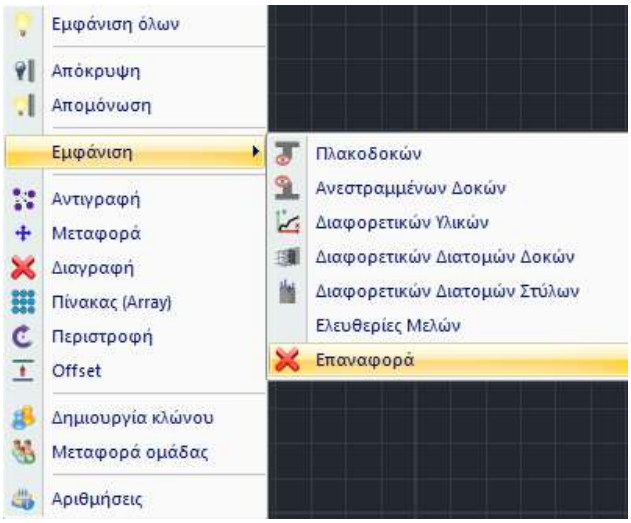
Regardless of the cross-section you select, the dialog box includes all other cross-sections. Based on the material you select, the grades, cross-sections and geometry are adjusted.



You can also model wooden sections, following exactly the same procedure as described for concrete and metal sections.
With SCADA Pro you can model and analyze concrete, metal, wood and load-bearing masonry structures.

RIGHT MOUSE BUTTON

The list includes commands found in the corresponding section that relate to the **beam**.



To deselect, press the right mouse button anywhere on the canvas surface.
The list opens a new list and you select "Show All".

In addition, with the "**Show**" command you can display the different cross-sections of the beams in different colours and the same colour for the same cross-sections and better overview of the elements of the structure, as well as identify slab beams, inverted beams, elements with different material qualities and member freedoms.

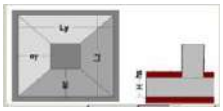
3. Foundation

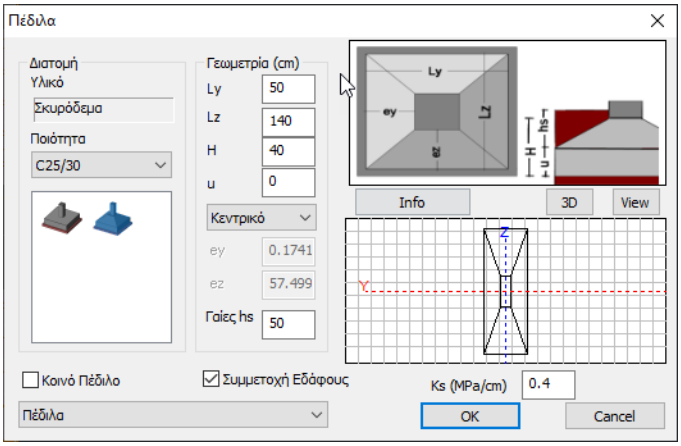
The "Foundation" command group contains the commands to model:
- **Sandals** and
- **Pedestals/Connectors**

Each command includes the corresponding sub-commands that define the type and shape of the cross-section.

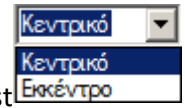
Define **the parameters of** the foundation. More specifically:

3.1 Sandal

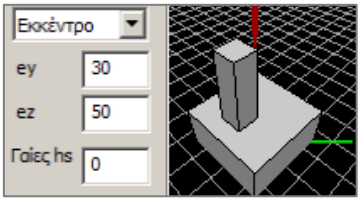


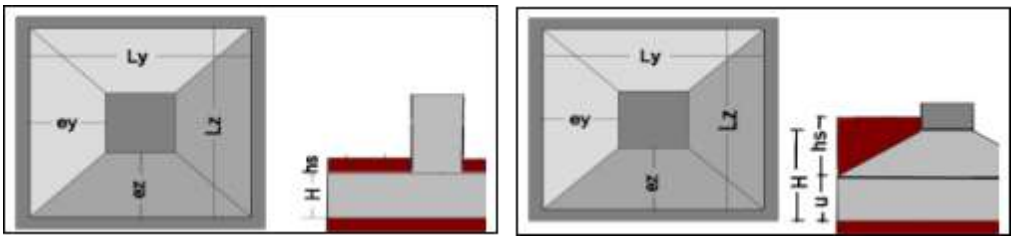


Here are the **parameters of** the field. More specifically:

- "**Cross-section**": you choose the quality and shape.
- "**Geometry**": enter the dimensions of the cross-section based on the graph showing the shape of the pediment.
- From the list 

: select whether the skirt will be central or eccentric to the corresponding column. For an eccentric skirt, enter the corresponding eccentricities by referring to the figures below.

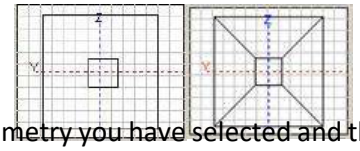




- In the "**Earths hs**" field, type the value you want. The height of the overlying earth is calculated from the base of the foundation to the ground surface.
- In "**Pedicle-Cone**", the difference is that you have to specify the thickness u of pedicle plate (equal to H/3)

ATTENTION:
A prerequisite for the placement of the pedestals is the existence of supports.

The checkbox next to "**Shared Skirt**" is activated to place a shared skirt on two or more columns.

In the window  the cross-section of the pediment is formed based on the geometry you have selected and the dimensions you have entered. Here you also see the local yy and zz axes of the cross-section.

- For elastic supports, check the "**Soil Involvement**" indicator,

and type the value of the **Ks** indicator.

The value given by the designer in the software (in MPa/cm which are the corresponding units of MN/m3 after conversion) is multiplied by the foundation surface (length x width) in order to obtain the vertical spring constant (in kN/m) which is placed at the corresponding node and which constant expresses the soil's intractability.
So the spring constant is Ky= Lx*Lz*ks.
Similarly, two torsion springs about the x-axis and about the z-axis, whose constant is equal to K=ks* the corresponding moment of inertia in each direction of the foundation element, are placed in the node.
For example, for a skirt with ks=1 MPa/cm = 100000 KPa /m and dimensions 1*1 m, the vertical constant of the spring used will be Ky=1 m*1 m*1 MPa/cm
= 100000 kN/m

- From the list of predefined layers 

you can select the layer to which the lap belt you are going to insert will belong. As an automatic default is the "Skirt" layer.

- ***How to place the pedestal***:

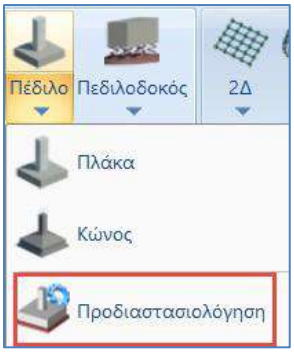
After giving all the elements of the pediment geometry, we select the substructure on which to place it, either by pointing to one of its vertices or by pointing to one its sides. In the case of pointing to the side, the pentagon is placed parallel to it.

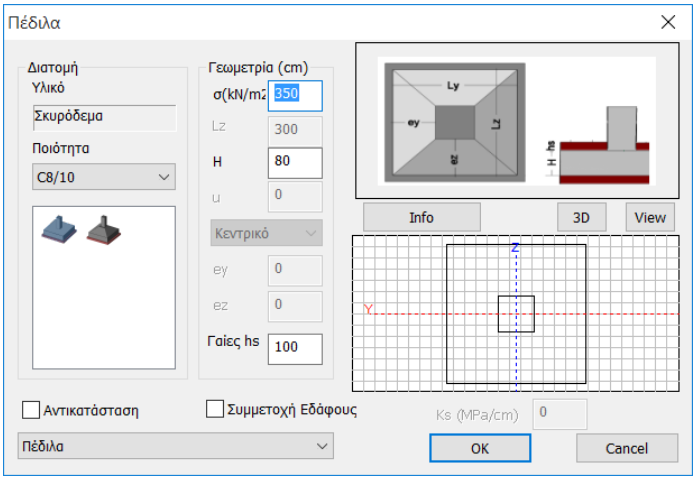
- ***How to place the common pedestal***:

After providing all the elements of the geometry of the pediment, we select the columns on which it will be placed.

OBSERVATION:

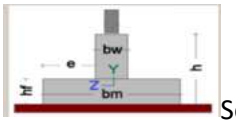
A new feature offered by SCADA Pro, after the creation of the Mathematical Model, is the "Prescaling" of the fields:



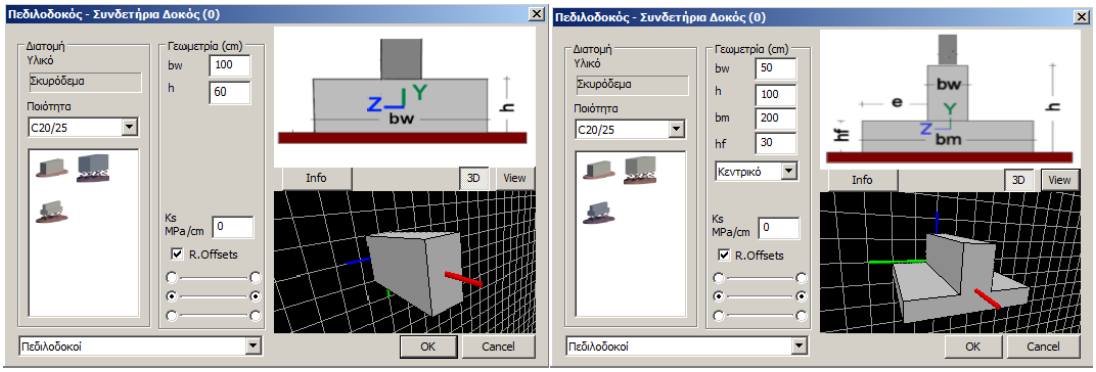


which, according to the soil tension σ(KN/m2), the height of the footings H and the overlying soil hs, pre-standardizes the existing footings, possibly modifying their dimensions.

3.2 Pedilodocus



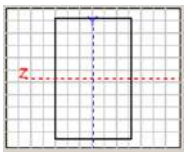
Select the cross-section and in the dialog box:

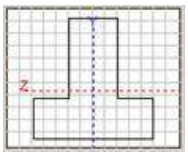


Here are the **parameters of** the pedicle. More specifically:

- "**Cross-section**": you choose the quality and shape.
- "**Geometry**": enter the dimensions of the cross-section based on the graph showing the shape of the footbridge.

In the window, the cross-section of the pediment is formed based on the geometry you have selected and the dimensions you have entered. Here you also see the local yy and zz axes of the cross-section.





The key 

: offers a three-dimensional representation of the pediment. By pressing and holding the left key and moving the mouse, the plane rotates and you can see the perimeter of the perimeter of the perforator.

- The "**R.Offsets**" option enables or disables the existence of rigid sections in the beams.

ATTENTION:
*To insert footings under the basement walls you must disable "**R.Offsets**" and "**Autotrim**".*

- The option  indicates the way of placing the pedestal, i.e. which terrace will be chosen for its design.

NOTE: *For the placement of the pedestals, the same applies as for the beams.*

Where "**Ks**", 

enter the value of the soil index.

From the list of predefined layers 

you can select the layer to which the pedestal to be inserted will belong.

- "**Info**"(Details) 

: select to see all the geometric and inertial data of the specific cross-section

3.3 Connecting beam

As for the pediment, but without the participation of the soil elements.

4. Surface

The "Surface" command group contains the commands to model:
-Surface 2D and
- Surface 3D
Each command includes the corresponding sub-commands describing, defining, editing and calculating the meshes.

OBSERVATION:
3D surfaces offer greater possibilities than 2D surfaces which are more limited.
You can simulate surfaces with both but be aware that with 2D surfaces with common boundaries are not automatically connected, which is the case with 3D (see EXAMPLE: MULTIPLE SURFACES MULTIPLE SURFACES)

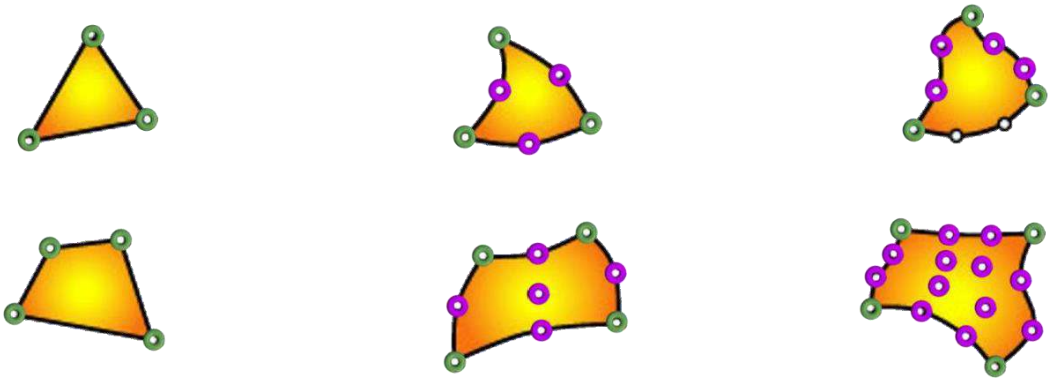
FINITE ELEMENTS

The types of finite elements you can use in **SCADA Pro** are generally grouped into **1D elements, 2D elements and 3D elements** and are identified by their shapes. For example, elements can take the form of a straight line or curve, triangle or quadrilateral, tetrahedron, and many others.
The simplest element is a line consisting of two nodes. All line, straight line or curve elements are called **1D elements** and are capable of displacements and rotations. Examples of 1D elements are the truss element (truss) and the beam element (beam3d)
SCADA Pro includes :

- Ribbed (linear) Network elements with function in space
- Ribbed (linear) beam-supporting elements with spatial function
- Ribbed (linear) elements of a pedestal on elastic ground with operation in space
- Boundary utilities for the simulation of elastic supports at the nodes of linear elements .

2D elements are usually surface elements triangular or quadrilateral. Examples of 2D elements are 3-node triangular element, 6-node triangular element and many others. These surface elements can have regular or irregular shapes as shown in the figure
2D elements are flat elements. Therefore, the linear approximation of the displacements considered are u (x, y) and v (x, y) while the rotations are θ (x, y). Since they correspond to plane stress and simple strain, they are often used to solve 2D elasticity problems.
SCADA Pro includes :

- Finite surface shell elements (quadrilateral or triangular)
- Finite surface shell elements on elastic ground (quadrilateral or triangular)
- Finite surface flat deformation elements
- Finite plane intensity surface elements for simulating surfaces generated by rotation.
- Finite surface elements of flat intensity.
- Boundary ancillary elements for the simulation of elastic supports at the nodes of surface elements

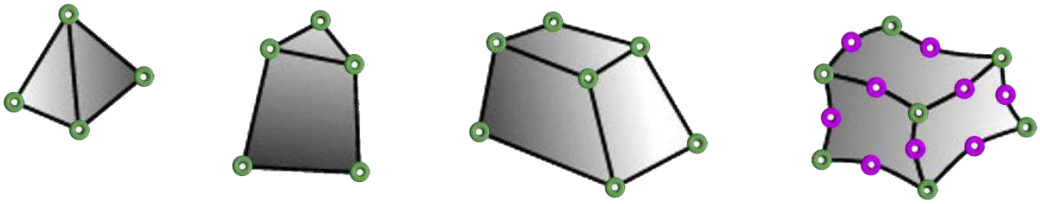


3D Elements. 3D elements are commonly used simulate volumes. They are derived from 2D elements and are used in more complex simulation problems

3D solid elements have only displacements and no rotations. The three unknown displacement functions are u (x, y, z), v (x, y, z) and w (x, y, z). Examples of 3D solid elements are 4-node tetrahedral element, 10-node tetrahedral element, 8-node isoparametric element, etc.

SCADA Pro includes :

- Three-dimensional, hexahedral, isoparametric finite elements, with varying intensity along their thickness (8-21 nodes)
- Boundary auxiliary elements for the simulation of elastic supports at the nodes of solid elements.

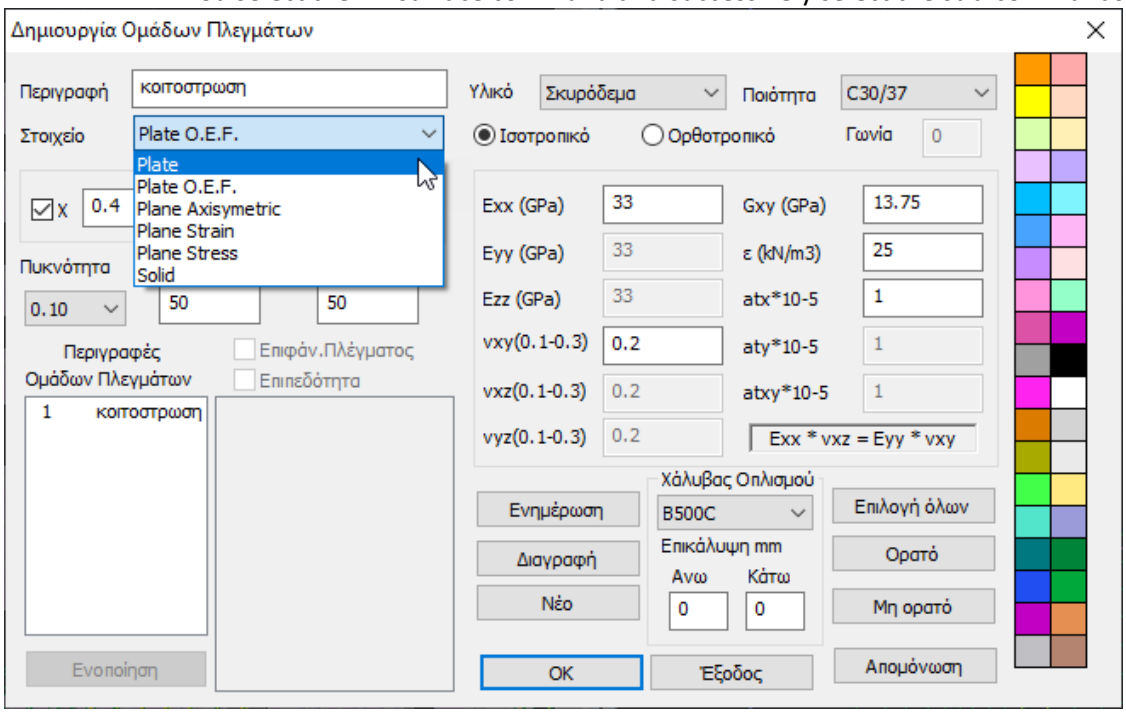


4.1 Surface 2D

With 2D surfaces you can model cavities and generally horizontal surfaces of any shape that do not share edges with other surfaces.

4.1.1 Grid

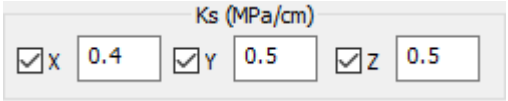
You select the 2D surface command and successively select the sub-commands it contains.



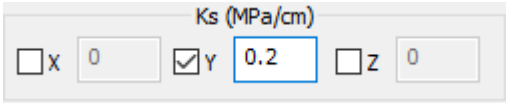
You specify the **characteristics of** the mesh groups you will use:

- "**Description**": type a name describing the group
- Select "**Material**" and "**Quality**"
- "**Item**" you select the type of item you want to place. If you select "Plate (O)n (E)lastic (F)oundation" you must enter a value of the soil index "Ks" in the corresponding field. This option is suitable for general depressions and the 'Plate' option for all other cases.

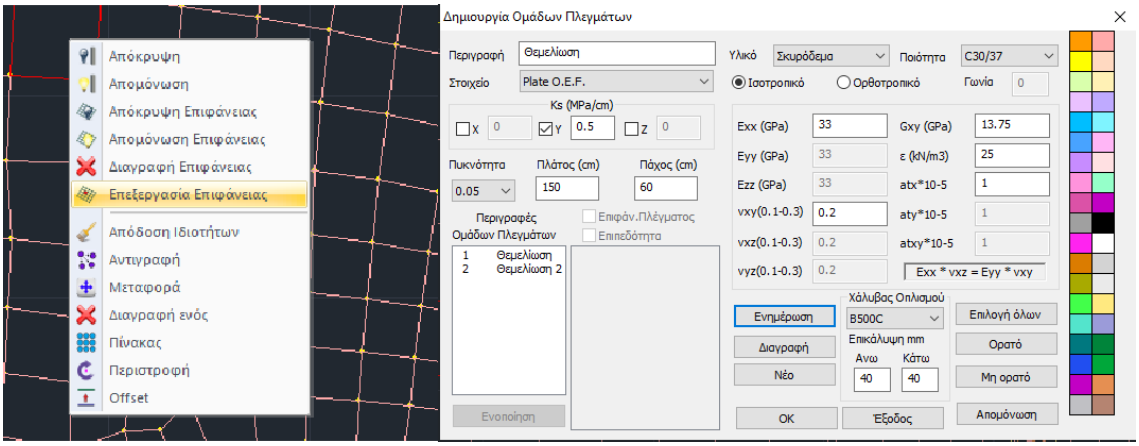
In addition, the new version added the possibility of horizontal springs in pavement where the user can define a soil index, Ks,x & Ks,z. The values of horizontal springs are defined in the respective fields activated by selecting the corresponding checkbox .



Also, in the elements on elastic ground, when no spring constant against Y is entered, the program automatically sets a value of 0.2

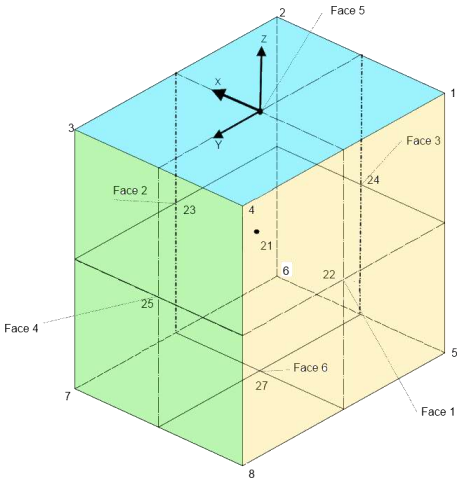


Selecting a finite element from a corresponding surface by right-clicking opens the Edit command where selecting it takes us directly to the Create Grid Groups window with the specific sub-surface selected for further editing.



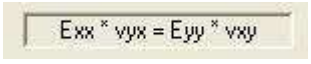
OBSERVATION:

In **Plane Stress** elements the loads are applied within the plane of the element (perpendicular to its thickness) considering zero stresses perpendicular to the plane of the element.
In **Plane Strain** elements the deformations perpendicular to the plane of the element are assumed to be zero and **Axisymmetric** elements are used to simulate surfaces produced by rotation.
In the new version of SCADA Pro a new surface element was added, the **Solid element** (see Solid element below). This element can be defined in its generic form, with a minimum number of nodes of 8 and a maximum number of 21.



- "**Density**", "**Width**" and "**Thickness**" refer to the geometry of the surface.
  "Density", expresses the smooth transition from an area with dense surface elements to an area with sparser surface elements. A larger size expresses a smoother "flow" surface elements and, of course, a greater number of surface elements. A low density can be used in cases where you want to use a few elements in order get an estimate of the intensive state (e.g. at the preliminary design level).
  In the "Width" field you enter in centimeters the width of each element and in the "Thickness" field the thickness of your surface.

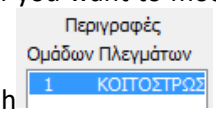
- The options "**Display. Grid**" and "**Flat surface**" options are disabled and are only used when importing 3D surface elements.
- In the "**Material**" and "**Quality**" fields, make the corresponding selections.
- The next option is whether the material is "**Isotropic** or **Orthotropic**". The **Orthotropic** material allows you to set different material properties per direction. If you specify an orthotropic material, the numerical values of the properties you specify must obey the relationship .



- "**Corner**": in the Orthotropic material will be activated in a future version of the program.
- "**New**": after you have completed the definition of all the above data, press the "**New**" button and in the "**Grid Group Descriptions**" section the grid you have just given is entered with a serial number. Following the same procedure you can create other mesh groups with different geometric and physical properties.

- "**Update**": to modify the details of the surface grid or grids you have already defined.

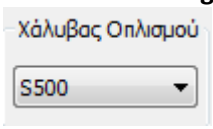
EXAMPLE:

For example, if you want to modify the thickness of the mesh "CUTTING" from 50 to 70 cm, first select the mesh 

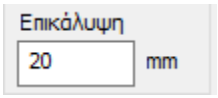
and then type the new value in the corresponding field. Then you press 

and the mesh now has the new value for its thickness. A similar procedure is used to change any other geometric or physical attribute of the mesh.

- **"Steel Armouring and Coating":**



is the field where you select the quality of steel with which to reinforce your mesh.



is the field where you select the thickness of the overlay.

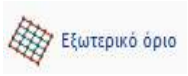
- "**Delete**": delete the grid or grids you have already created. From the "Grid Group Descriptions" section, select the grid and press the "Delete" button.

OBSERVATION:

The grid does not disappear from the list, but the word "Delete" is added to its name to indicate that it has been deleted. This fact allows you to restore a grid that you have already deleted by simply selecting it and pressing the "Delete" button. The word "Delete" from its name disappears and the grid becomes active again.

To permanently delete a grid, after the "Delete" command, make a study entry via the 

4.1.2 Outer Limit



Instruction to set the **Outer Limit of** the surface.

OBSERVATION:
If there is no grid, the dialog box for defining the grid as previously described opens. If there is a defined grid, it is used as described below.

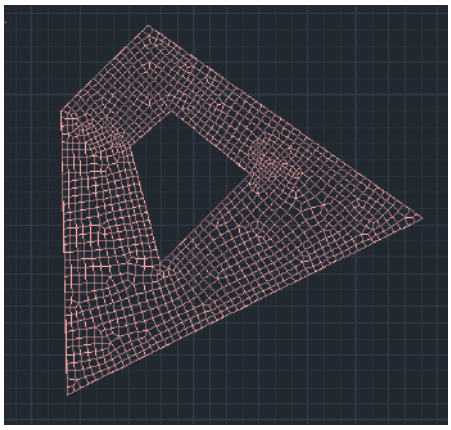
To define the outer boundary of the surface grid, after selecting the command, you create an outline by left-clicking successively on the points that are the vertices of the outline and using the osnaps. You complete the command by pointing as the last point, the point with which you started, or simply move the mouse closer to the starting point and press the right key.

If you want to delete an external boundary, you must delete the corresponding grid group that you have created.

4.1.3 Holes

Command to specify contour contingency contours **of holes** on the grid surface. Call the command and define the contour of the hole, as for the outer boundary.

OBSERVATION:
The definition of the holes can be done afterwards, after the surface grid has been created. Using the "Calculate" command, which will be explained below, the surface grid is recalculated taking into account the hole.

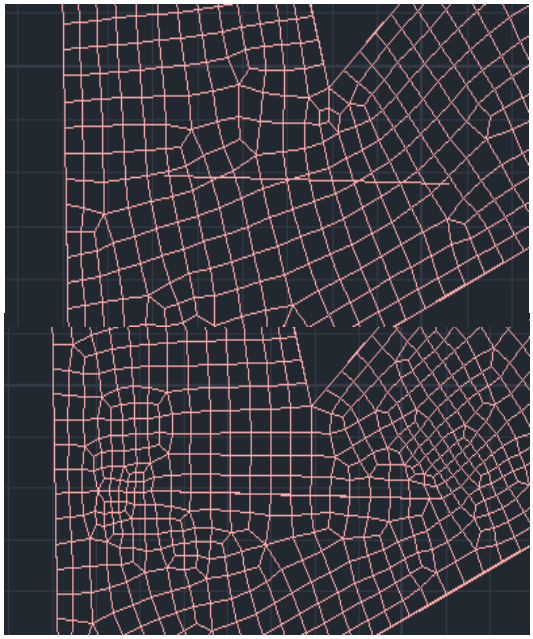


4.1.4 Line

Command to specify with **Lines**, region or regions where the surface grid will be densified. You call the command and draw a line or lines within the boundary of the area of the surfacers you have already specified.

OBSERVATION:

The definition of the lines can be done afterwards, after the surface grid has been created. Using the "Calculate" command, which will be explained below, the surface grid is recalculated based on the line as well.

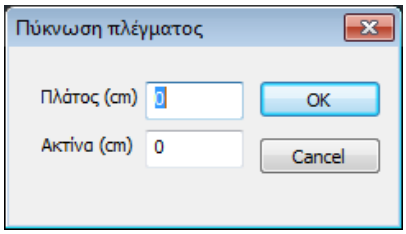


This creates the new grid configuration based on the taper line.

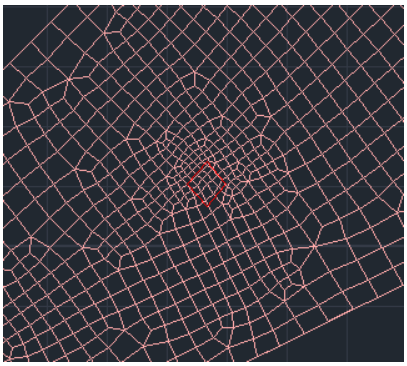
4.1.5 Point

Command to define **Points** within the surface grid which will be densification points.

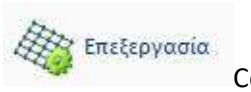
You select the command and define the thickening area around the point. Then, you point to a point within the surface of the surface. You select "Calculate" and obtain the thickening.



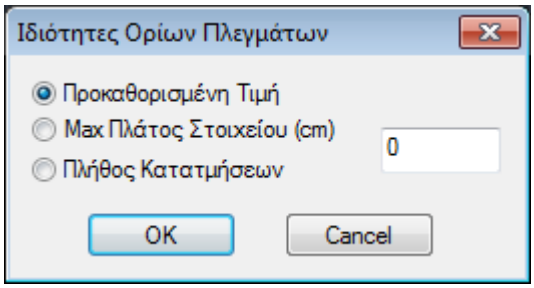
OBSERVATION:
Point definition can be done afterwards, after the surface grid has been created. Using the 'Calculate' command, which will be explained below, the surface grid is recalculated based on the point as well.



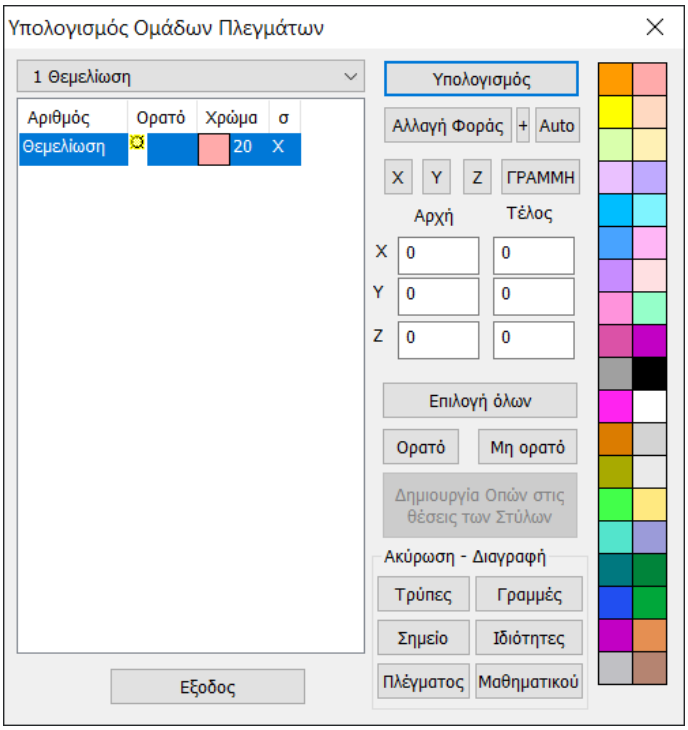
4.1.6 Edit



Command to **Edit** the mesh or meshes of the finite surface elements you have already entered. This editing must be done after creating the surface mesh and before creating the mathematical model. Using the command displays the following dialog box



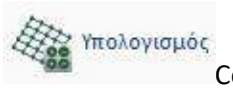
- "**Default Value**": you can specify a specific number of surface elements that your grid will contain. You select it and in the field to the right, type the number of elements you want. Then you press the "OK" button and point the mouse successively to the sides of the contour. Press right mouse button to indicate that you have completed the selection and then call the "Grid Calculation" command. In the dialog box that appears, press the "Calculate" button.



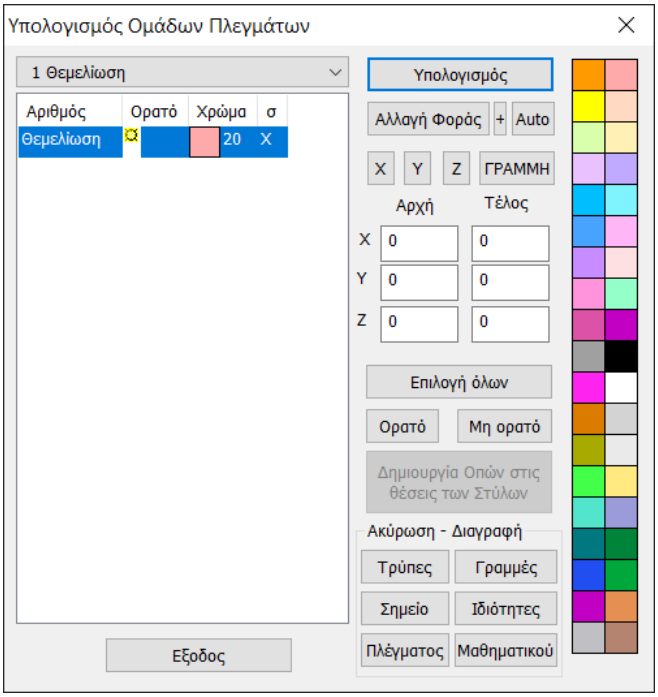
The surface grid is redefined based on the number of elements you specified.

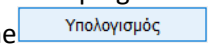
- "**Max element width**": type in centimetres the maximum width you want your surface to have. You press the "OK" button and point with the mouse to one or more sides of the surfacer outline where you want the surfacer to have this maximum width. Finish your selection by pressing the left mouse button and then call the "Grid Calculation" command. In the dialog box that appears, select the grid (it turns blue) and press the "Calculate" button. The surface grid is redefined based on the maximum width of the elements on the sides you specified.
- "**Number of segments**": enter the number of segments (not the number of items). You press the "OK" button and point with the mouse to one or more sides of the surface contour, which you want to have the number of segments you set previously. You finish your selection with the left mouse button and then call the "Grid Calculation" command. In the dialog box that appears, select the grid (it turns blue) and press the "Calculate" button. The surface grid is redefined based on the number of segments you specified.

4.1.7 Calculation



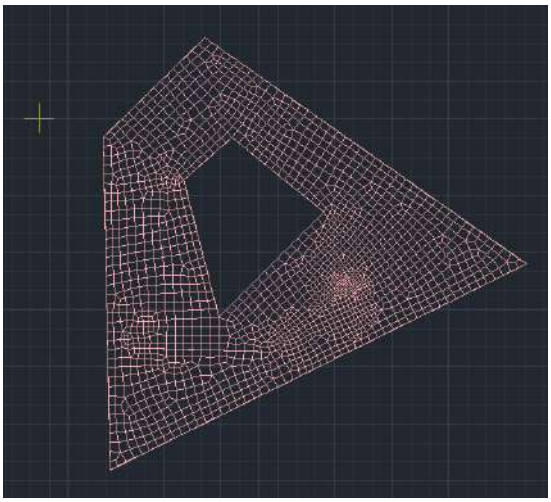
Command to Calculate the 2D surface grid, taking into account the contour you have previously defined and any holes, points and lines.
Using the command, the following dialog box appears



In the top list of the dialog box, select the surface grid you want to calculate. Note here that each grid group, refers to a single discrete surface that you have defined with a specific boundary. This practically means that you cannot use the " plegma" surface grid for two areas.
To make the grid calculation, press the 

button. The surface grid is thus created.

The result of the simulation is shown in the image below:

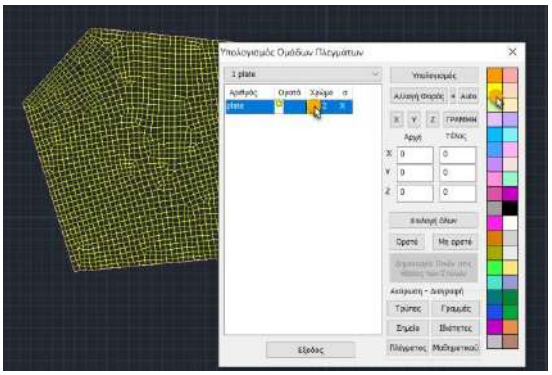


OBSERVATION:

*In the above way, only the grid is created. The mathematical model of the surface has not yet been created, which is done with the command in "Tools>> "**Calculate**".*

The same dialog box, in addition to the Calculator, contains additional useful tools:

- To change the colour of the grid, select it from the list and click the colour you want from the colour palette.



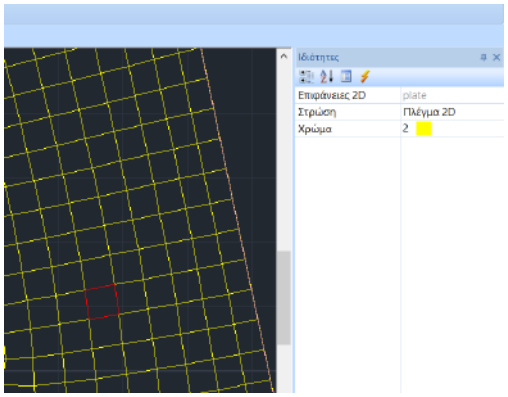
- By using the buttons , 

, you can make the grid appear or not appear respectively. Select the grid you want to make visible or not visible and press the corresponding key. The indicator under the 

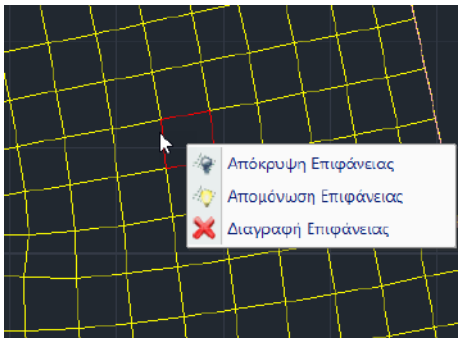
changes from  (visible) to  (not visible).

NEW POWERS and IMPORTANT REMARKS:

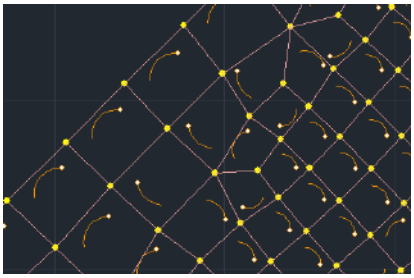
- A sub-surface mesh may be referenced before the mathematical model is created. By left-clicking on the mesh, the name, color and
  layer. The last two can also be modified.



- It can be drawn at the vertices of the grid (only the nearest works), before the mathematical model is created.



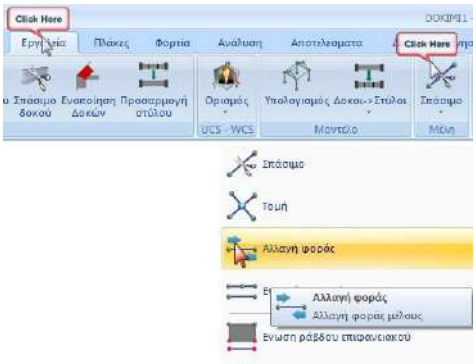
- By right-clicking on the grid, a command menu appears from which Hide can be done - Isolation of the surface mesh before creating the mathematical model.



- The local axes are now shown even before the mathematical model is created and are activated through the "**Appearance**">> section "**Switches**" >> "**Local Axes**".

- By using the "Change Face" button the local axes a direction of the surface overall.

- If you want to change the time on individual elements in the grid, use the "Tools>> "Members' **Tools**>> "**Change of Shirt**".

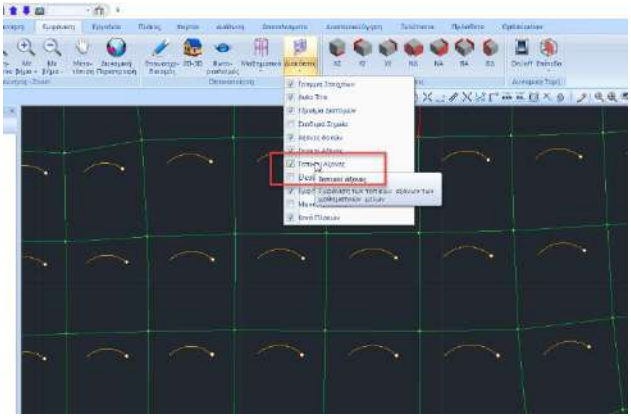


- With the "**Auto**" command the program adjusts the local axes of the surface axes throughout the grid so that they have the same direction.

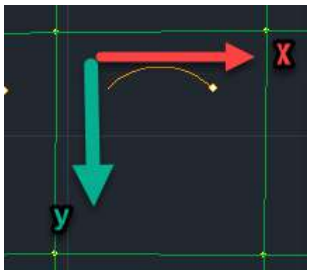
- In the finite surface and solid elements the Face Change, Auto option and display local axes now and only works with the existence of the mesh and before the creation of the mathematical model.

- Also, next to the Change Mode button, the (+) button was added which turns the tadpole 90 degrees with each press.

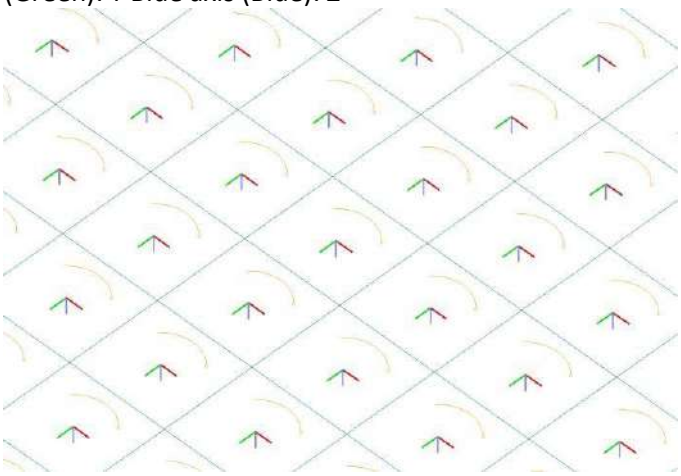
To display the local axes of the surface elements, activate the corresponding option within the **Switches**.



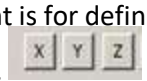
The arcs that appear define the local axes of the surface features, according to the clockwise screw rule.
The direction of the arc indicates the **x** and the point at its edge, the direction.

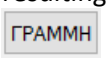


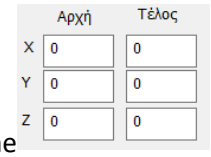
In the new version of the program, the local axes in the surface finite elements are now displayed in 3D for a better overview, following the well-known convention of the right-handed screw and in colours:
Red axis (Red) : X Green axis
(Green): Y Blue axis (Blue): Z



Also, the display of local axes for both linear and surface finite elements has been enabled in the Results section.

- The field on the right is for defining the main direction of the reinforcement (direction X,Y,Z) for the vertical meshes. 

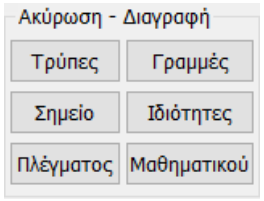
  - For vertical grids // on the X-axis : I select the grid and from the list and the column ''s'' is updated, respectively,
  - For vertical grids // on the Z-axis: select the grid and  from the list and the column ''s'' is updated, respectively
  - For meshes that are not // on either X or Z if I do nothing the program will project the resulting armature by reducing it to the 2 main axes. Alternatively I can use the command . 



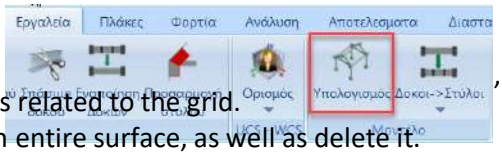
- The 

key works in combination with the field, where you define the coordinates of the beginning and end of the line, which the program will consider as the main direction of the reinforcement, in cases where the surface is not parallel to the universal axes, hence its reinforcement.

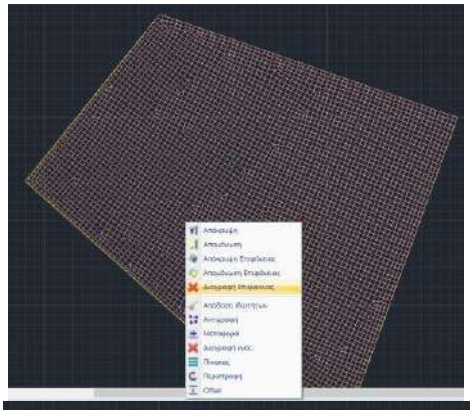
- The "Cancel-Delete" field allows deleting holes, lines, points, as as, properties, grid or mathematical model that you have already created and wish to delete.

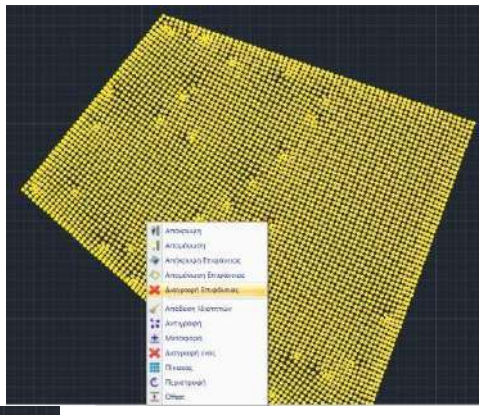


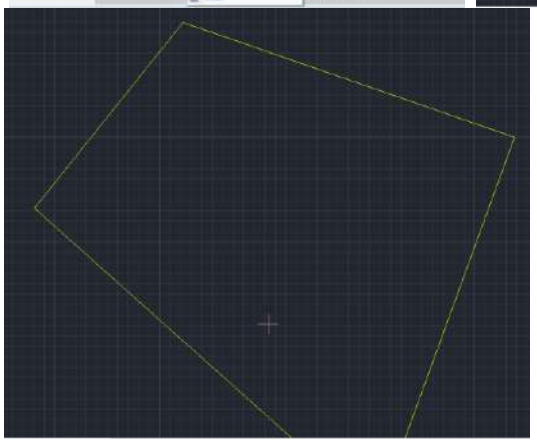
- After the creation of the Mathematical Model, right-clicking inside the grid opens a list of commands related to the grid.



With a right click you can Hide or Isolate a finite or an entire surface, as well as delete it.
The **Delete Grid** command allows you to delete both the mathematical model (if it exists) and its calculation, and the grid itself from the list.







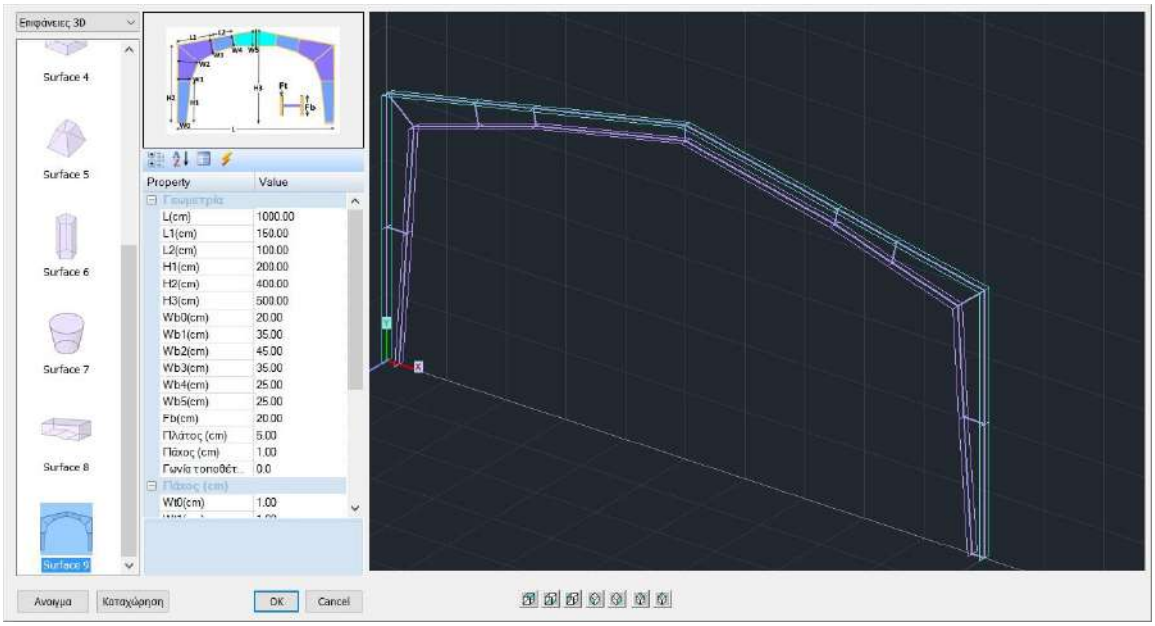
4.2 Surface 3D

4.2.1 Grid

With 3D Surfaces you can model surfaces of any shape (horizontal, vertical, inclined, concave) and surfaces with common limit.

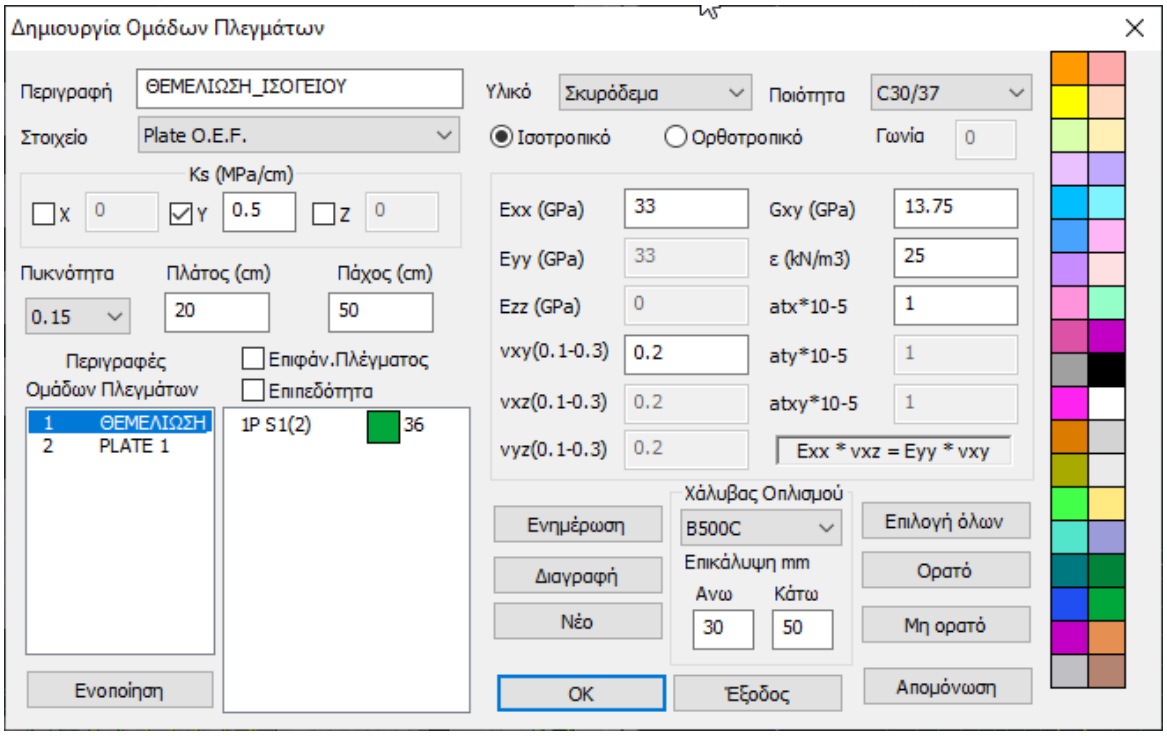
Three-dimensional surface modelling is suitable for the design of load-bearing masonry structures.

OBSERVATION: *There is also the possibility to automatically simulate a typical metal frame of variable cross-section with finite surface elements*



SCADA Pro offers multiple possibilities for the use of 3D surfacing through "**Standard Constructions**" and "**Automatic Face Recognition**", commands that are explained in detail later in the same chapter.

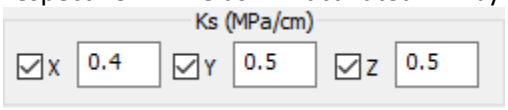
You select the Surface 3D command and successively select the sub-commands it contains.



You specify the **characteristics of** the mesh groups you will use:

- "**Description**": type a name describing the group
- "**Item**" you select the type of item you want to place. If you select "Plate (O)n (E)lastic (F)oundation" you must enter a value of the soil index "Ks" in the corresponding field. This option is suitable for general depressions and the 'Plate' option for all other cases.

NEW In addition, the new version added the possibility of horizontal springs in pavement where the user can define a soil index, Ks,x & Ks,z. The values of horizontal springs are defined in the respective fields activated by selecting the corresponding checkbox

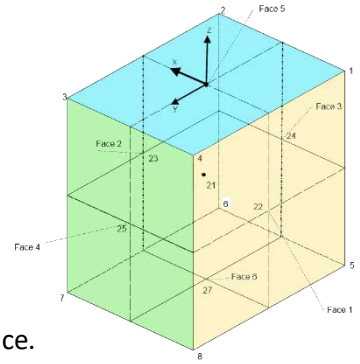


.

OBSERVATION:

In **Plane Stress** elements the loads are applied within the plane of the element (perpendicular to its thickness) considering zero stresses perpendicular to the plane of the element.
In **Plane Strain** elements the deformations perpendicular to the plane of the element are assumed to be zero and **Axisymmetric** elements are used to simulate surfaces produced by rotation.

In the new version of SCADA Pro a new surface element was added, the **Solid element** (see Solid element below). This element can be defined in its generic form, with a minimum number of nodes of 8 and a maximum number of 21.



"**Density**", "**Width**" and "**Thickness**" refer to the geometry of the surface.

- "**Density**", expresses the smooth transition from an area with dense surface elements to an area with sparser surface elements. A larger size expresses a smoother "flow" of surface elements and, of course, a greater number of surface elements. A low density can be used in cases where you want to use a few elements in order to get an estimate of the intensive state (e.g. at the preliminary design level). In the '**Width**' field enter in centimetres the width of each element and in the 'Width' field enter in centimetres the width of each element.

- "**Thickness**" the thickness of your surface.

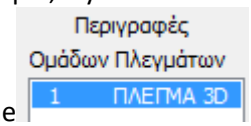
- In the "**Material**" and "**Quality**" fields, make the corresponding selections.
- The next option is whether the material is "**Isotropic** or **Orthotropic**". The Orthotropic material allows you to set different material properties per direction. If you specify an orthotropic material, the numerical values of the properties you specify must obey the relationship . $Exx * vyx = Eyy * vxy$

- "**Corner**": in the Orthotropic material will be activated in a future version of the program.

- "**New**": after you have completed the definition of all the above data, press the "New" button and in the "Grid Group Descriptions" section the grid you have just given is entered with a serial number. Following the same procedure you can create other mesh groups with different geometric and physical properties.

- "**Update**": to modify the details of the surface grid or grids you have already defined.

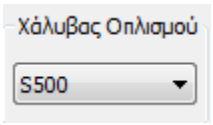
EXAMPLE:

For example, if you want to modify the thickness of the "PLEGMA 3D" mesh from 50 to 60 cm, first select the 

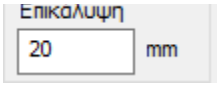
grid and then type the new value in the corresponding field. You then press 

and the grid now has the new value its thickness. A similar procedure is used to change any other geometric or physical attribute of the mesh.

- **"Steel Armouring and Coating":**



is the field where you select the quality of steel with which to reinforce your mesh.



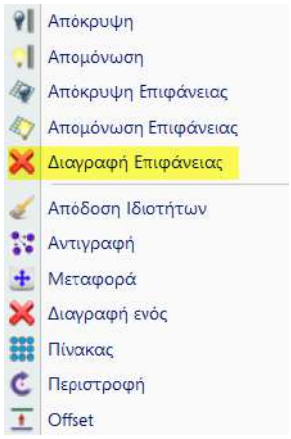
is the field where you select the thickness of the overlay.

There is now also the possibility of defining the overlap by surface subgroup.

- "**Delete**": delete the grid or grids you have already created. From the "Grid Group Descriptions" section, select the grid and press the "Delete" button.

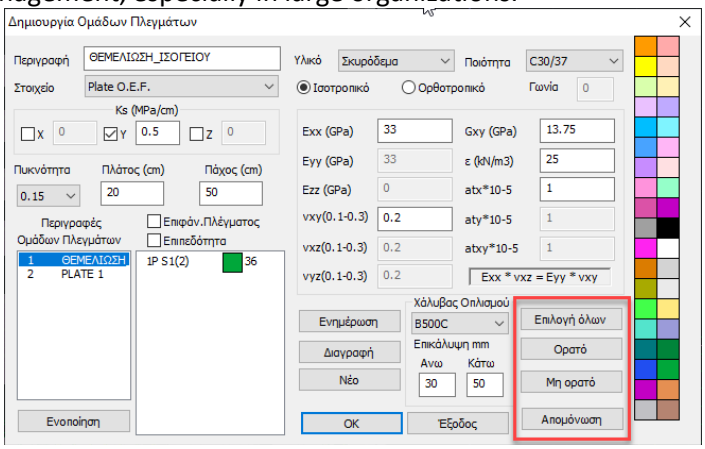
OBSERVATION:
The grid does not disappear from the list, but the word "Delete" is added to its name to indicate that it has been deleted. This fact allows you to restore a grid that you have already deleted by simply selecting it and pressing the "Delete" button. The word "Delete" from its name disappears and the grid becomes active again.
To permanently delete a grid, after the "Delete" command, make a study entry via the 

OBSERVATION:
Right-clicking inside the grid opens a list of commands related to the grid.
The ***Delete Surface*** *command allows you to delete both the mathematical model (if it exists) and its calculation, and the mesh itself from the list.*



- The options "**Display. Grid**" and "**Flat surface**" options are active and are only used when importing 3D surface elements. These options make sense when the mesh consists of several Grid Surfaces.

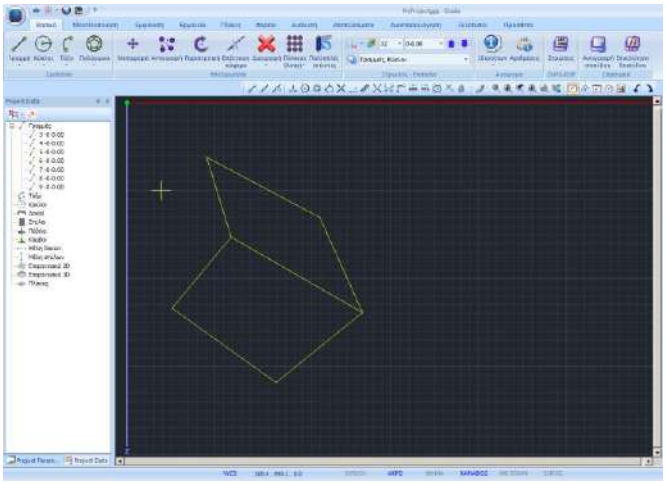
NEW The new version added the ability to change the color, isolation, as well hide - display one or more sub-surfaces through the dialog box of their definition and creation, for better supervision and management, especially in large organizations.



4.2.1.1 Mesh with more mesh surfaces

When we want to model meshes with common edges, we create a mesh with more mesh surfaces, so that the nodes on the edges, which will be created by calculating the matrix surface, will be created in the mesh. Model, are common to both surfaces.

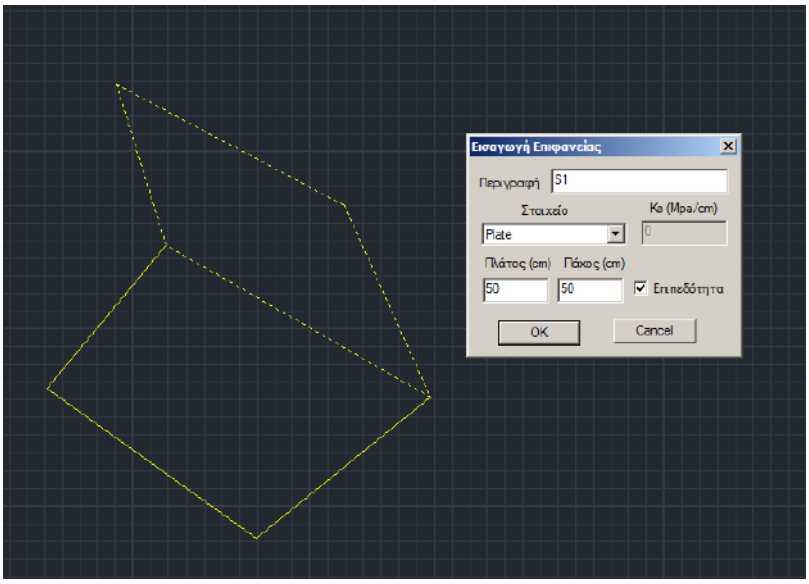
EXAMPLE:
Draw on the desktop, using a line or polyline, two boundary surfaces, as in the example.



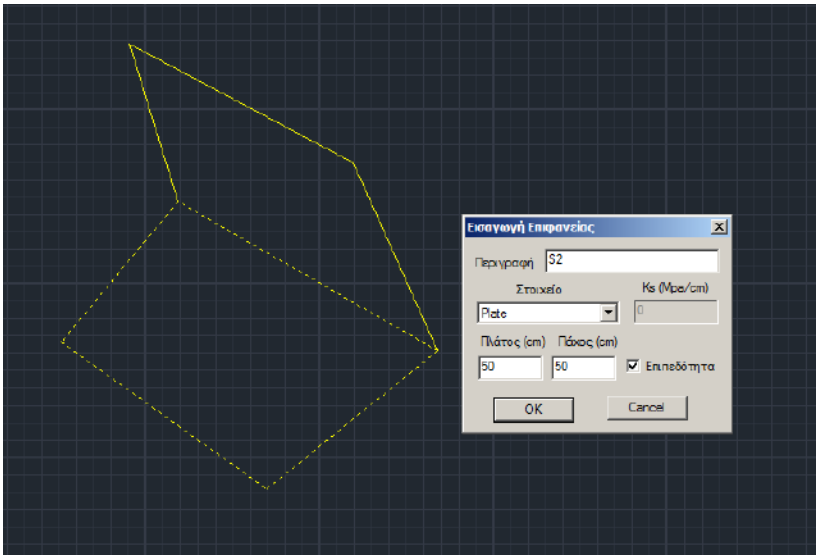
With the command "**Modeling**">>"**3D Surface**">>"**Mesh**", you define the mesh group.

Select from the "**Surface 3D**" group the command "**External** Boundary" 

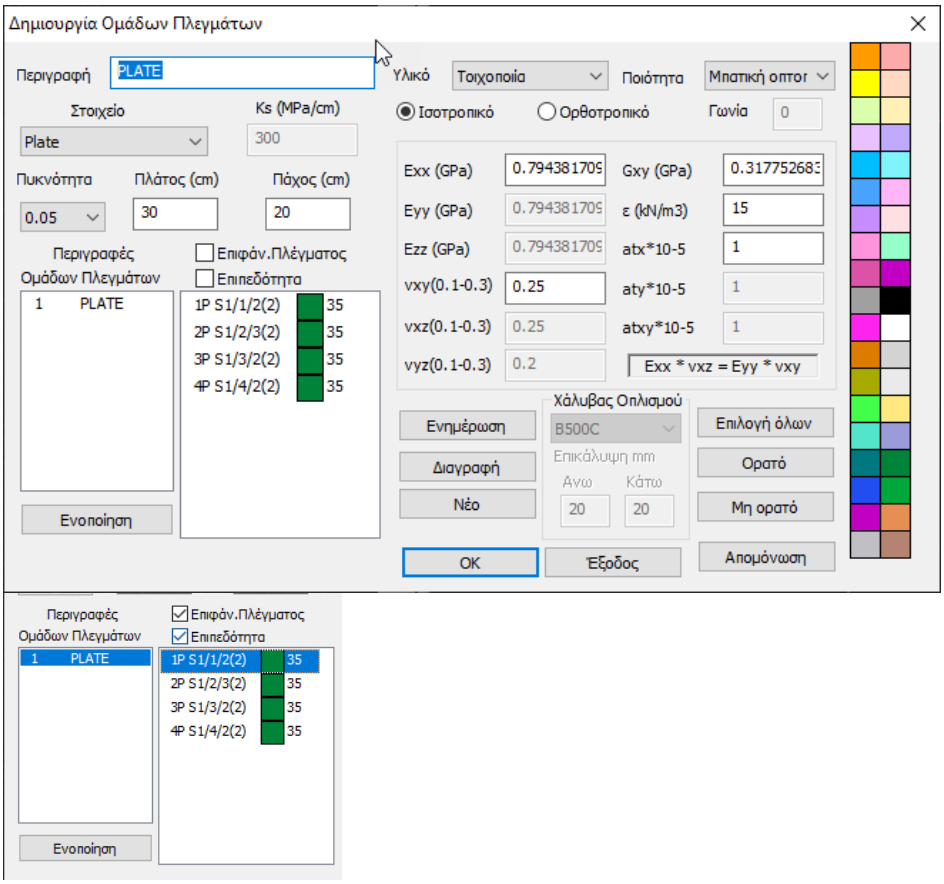
and with left button select one by one the lines of the first surface (1st subgroup) and right button to complete. The dialog box that appears is for the specific subgroup and its characteristics. The check next to Flatness indicates that the surface belongs to the layer.



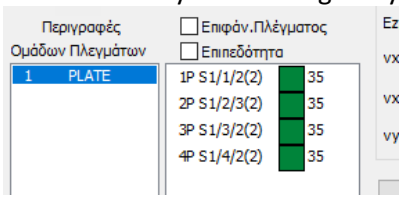
Select OK and continue the process in the same way for the second surface (2nd subgroup).



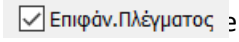
Selecting once again the command "**Modeling**">>"**3D** Surface>> "**Grid**", notice that in the box next to the group, the two subgroups are displayed.



You can modify the entire grid by clicking on the Grid Group



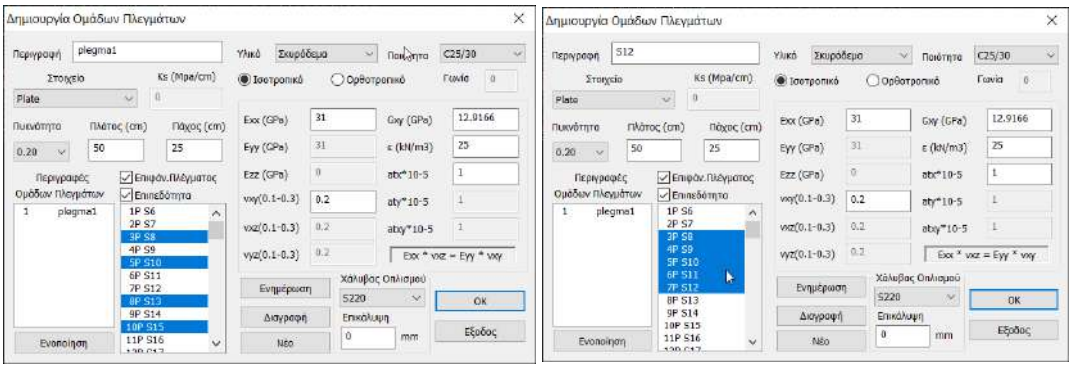
The individual sub-surfaces, by activating the checkbox of the Grid Surface and



subgroup to select one,

There is now also the possibility of multiple subgroup selection in the surface grids:

- Ctrl and click on more subgroups to select more,
- Shift and scroll to select sequential



That is, you can modify the geometry, material, flatness, reinforcement steel, coating, of a Group or one or more Grid Surfaces, by selecting them in one of the above ways and changing the options and then pressing "Update".
Similarly, you have the option, by selecting Delete, to delete the selected sub-tab(s).

If a mesh surface is flat then check the Flatness checkbox

4.2.1.2 Grid consolidation

In the new version of the program you now have the possibility to consolidate two independent surface groups. All their subgroups are now integrated into a new single group. This tool is particularly useful in the case of multi-storey buildings made of load-bearing masonry.
This new feature can also be applied in the case of a master group. One click automatically corrects subgroup outlines where required (deleting duplicate lines that match and partitions where they need to be performed).

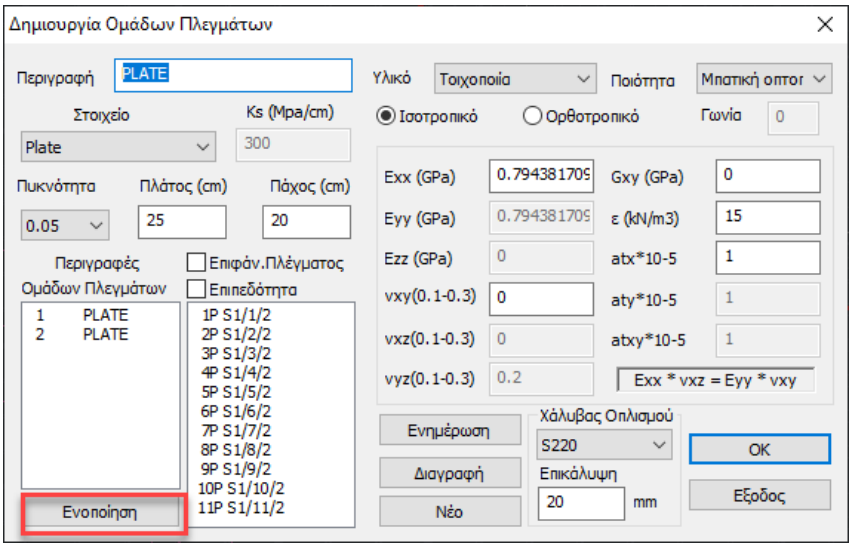
The new 

command is a new command that comes to solve several problems that occurred when the boundaries of sub-surfaces that were in contact were not certain "correctly" (e.g. at their common border the line of the lower wall is not broken, there is a double line).

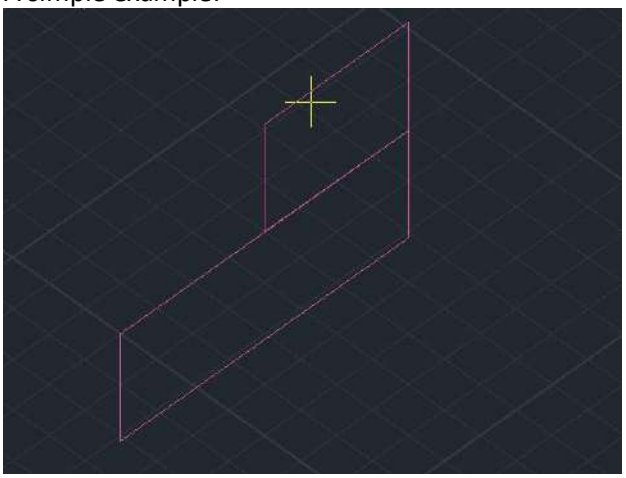
The following examples explain and inform about these issues. 

consolidates major groups and corrects automatically.

The problem of creating different groups (each with different subgroups) of surface groups is more general and as such was addressed by creating a new "Consolidate" command.

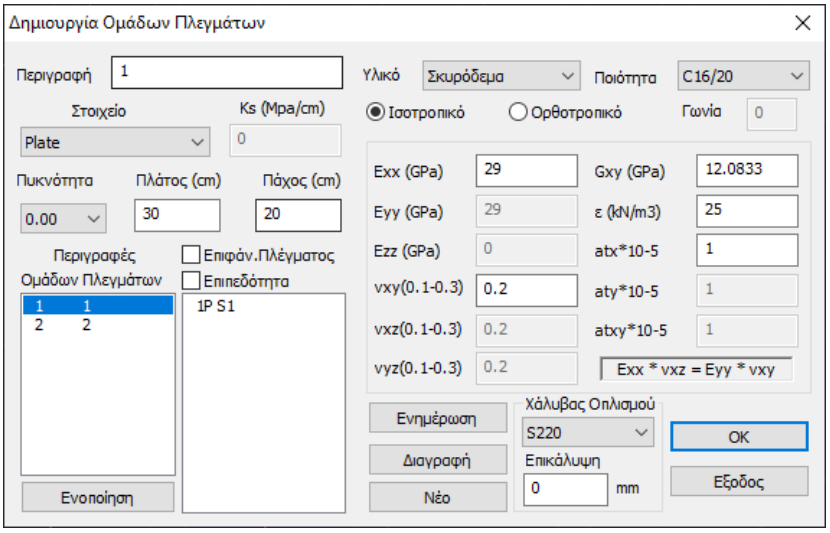


A simple example:

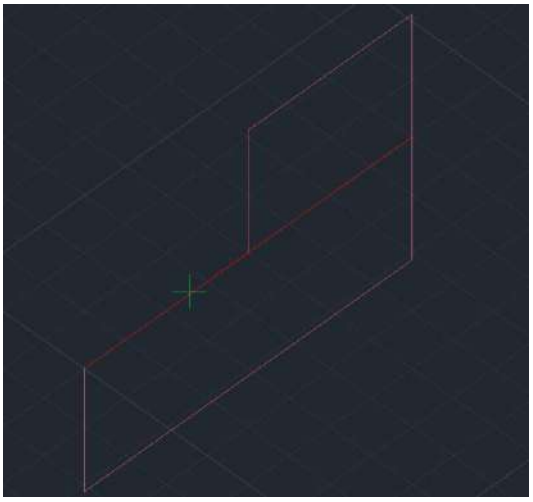


Let's consider the two walls above which:

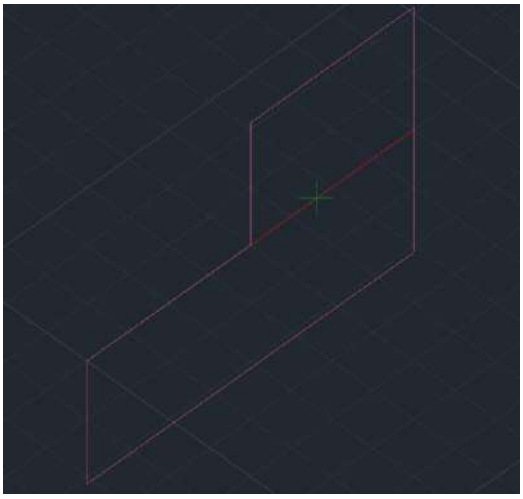
- each belong to two different main groups (each group has only one subgroup)



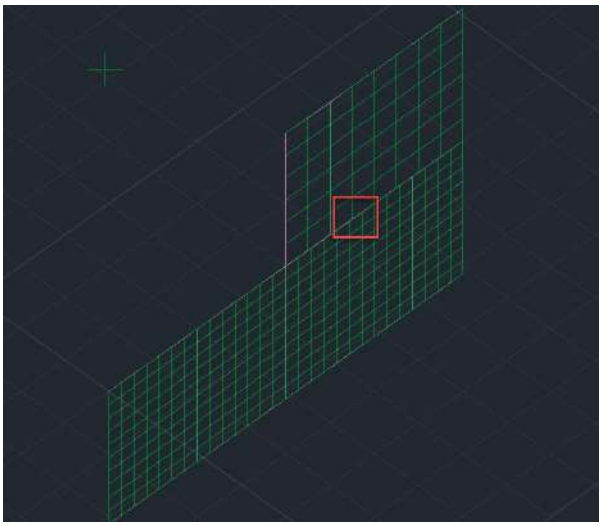
- At their common border the line of the lower wall is not broken



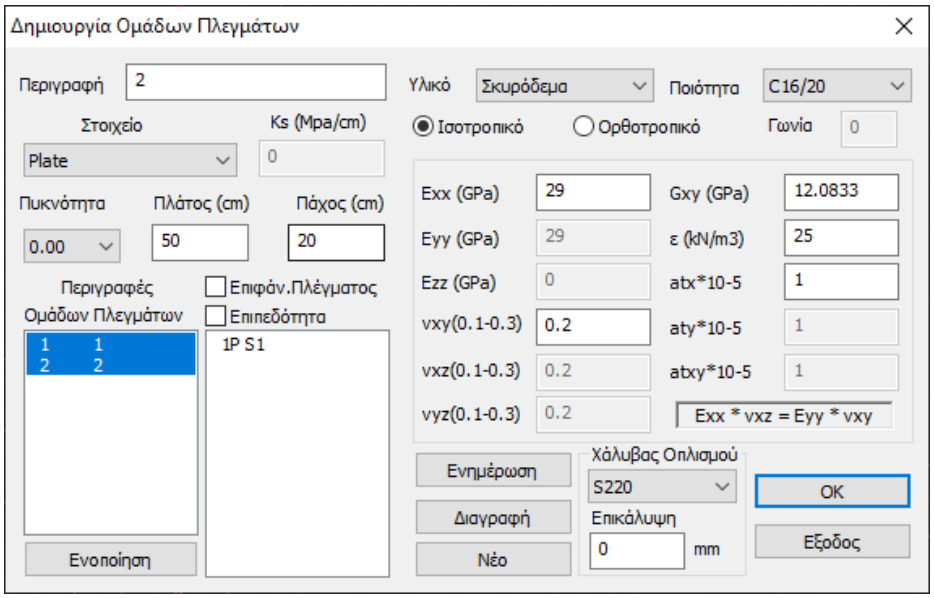
- There is a double line where the top wall touches



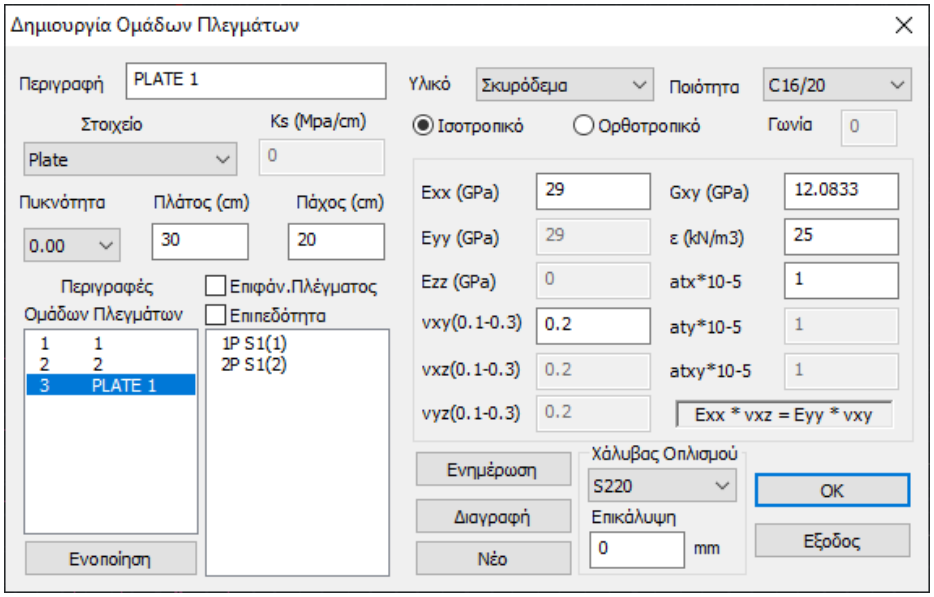
By calculating the grids as they are, the grids are created but the subsurfaces are not connected correctly.



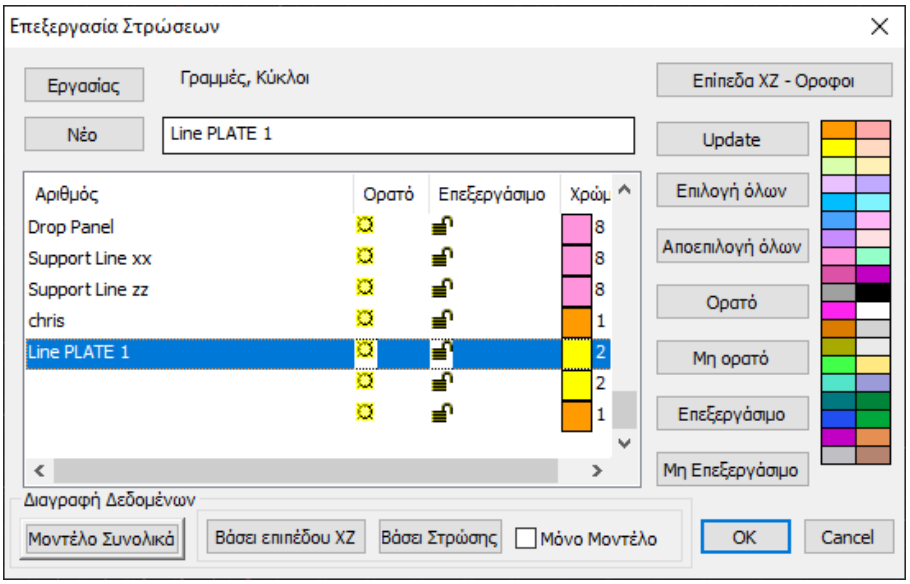
Using the "Consolidate" command, I select the main surfaces I want to consolidate



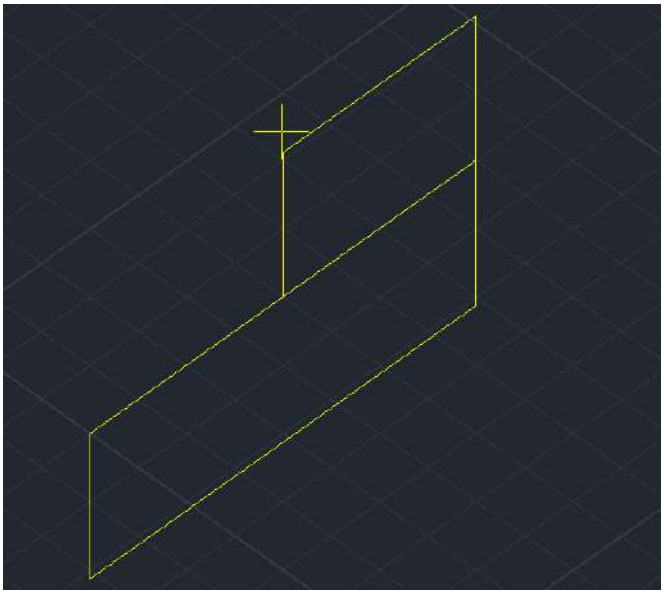
By pressing the "Consolidate" button the program creates a new group with the name PLATE 1



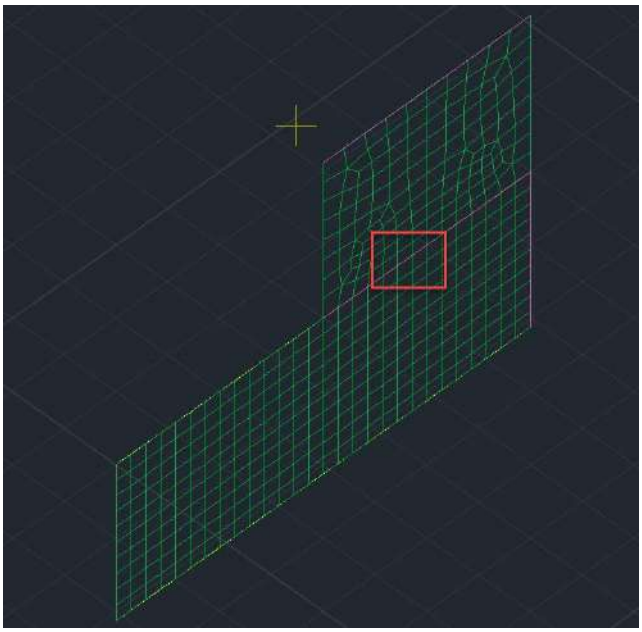
which includes both of the previous two subgroups. The name is standard (PLATE) with a serial number (1,2...). At the same time, the program creates a new layer with the name of the group (PLATE 1) and the word Line in front of it (Line PLATE 1),



which includes all the lines of the outlines of the two sub-surfaces redrawn correctly and properly. The names of the sub-surfaces shall include in parentheses the name of the original group from which they originated.



The lines are now automatically drawn correctly, i.e. without duplicate sections and broken where they should be.
Now calculating the grid of the new surface



We see that the sub-surfaces have been connected

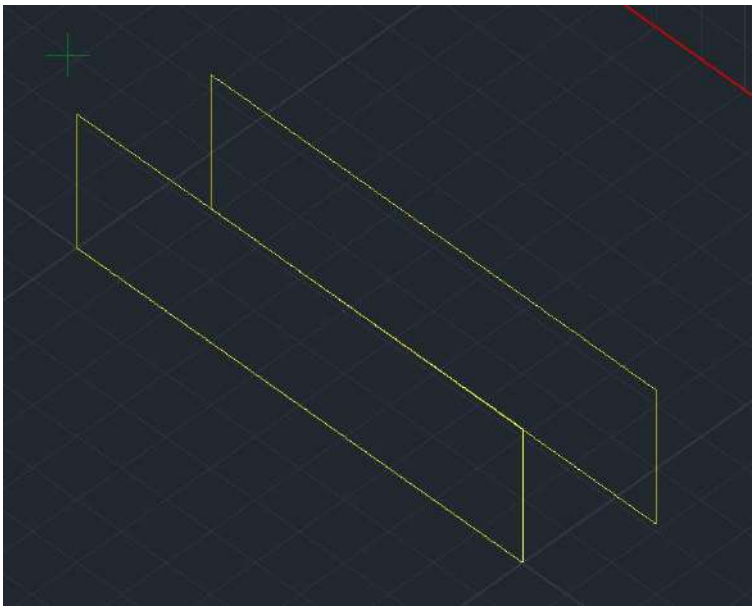
correctly. Some **OBSERVATIONS** about the process:

- When we consolidate two or more groups, the program creates a new main group that includes all subgroups cumulatively, keeping the properties of each subgroup independent of the general properties of the original groups. If, for example, I consolidate two groups that one has a thickness of 40 cm and the other 50 cm, the subgroups will keep their thicknesses correct, regardless of the thickness I see in the new main group.
- The command creates a new main group and new lines - outlines which are also placed on a new layer. All existing elements (original groups and original lines - outlines) still exist and if the user wants to the user can delete them manually after the consolidation.

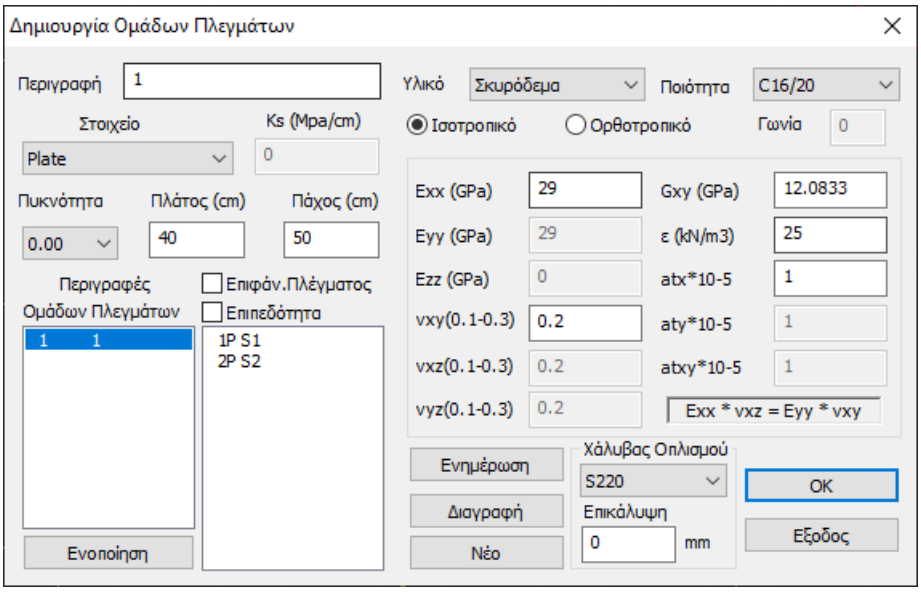
4.2.1.3 Grid consolidation for contour correction

The unification command can be applied to a master group AND to a master group and has the meaning of reordering i.e. correcting the outlines (double lines and breaks where they should be).

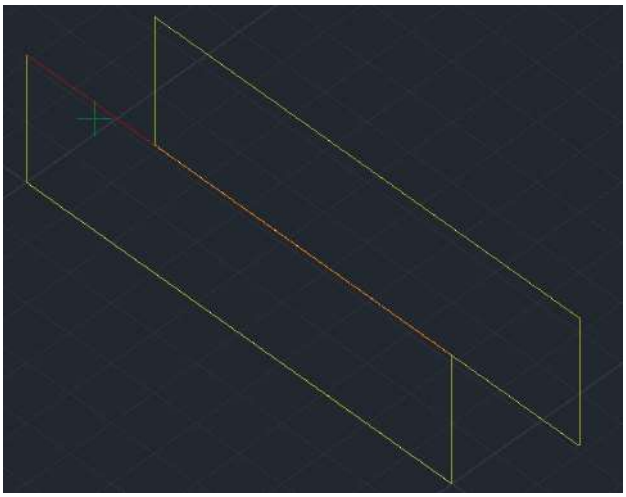
Let's look at an example: We have the following two walls



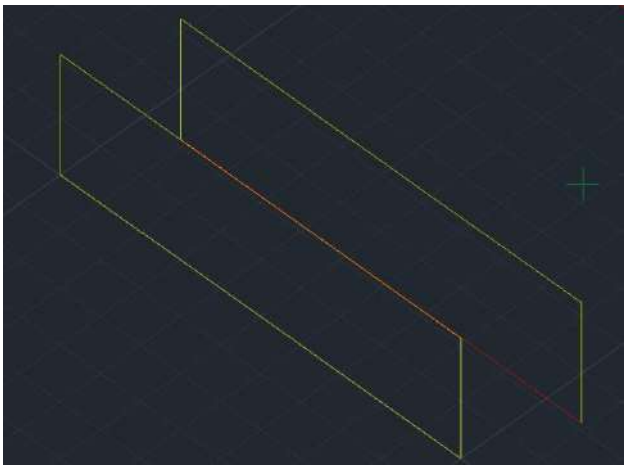
Walls are two sub-surfaces in the same main group



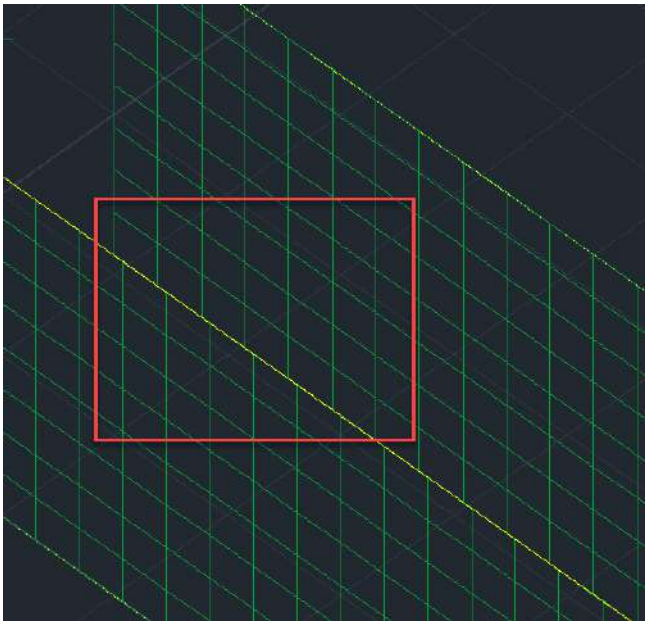
BUT their contours are two independent contours i.e. on their common border I have the entire top line of the lower wall



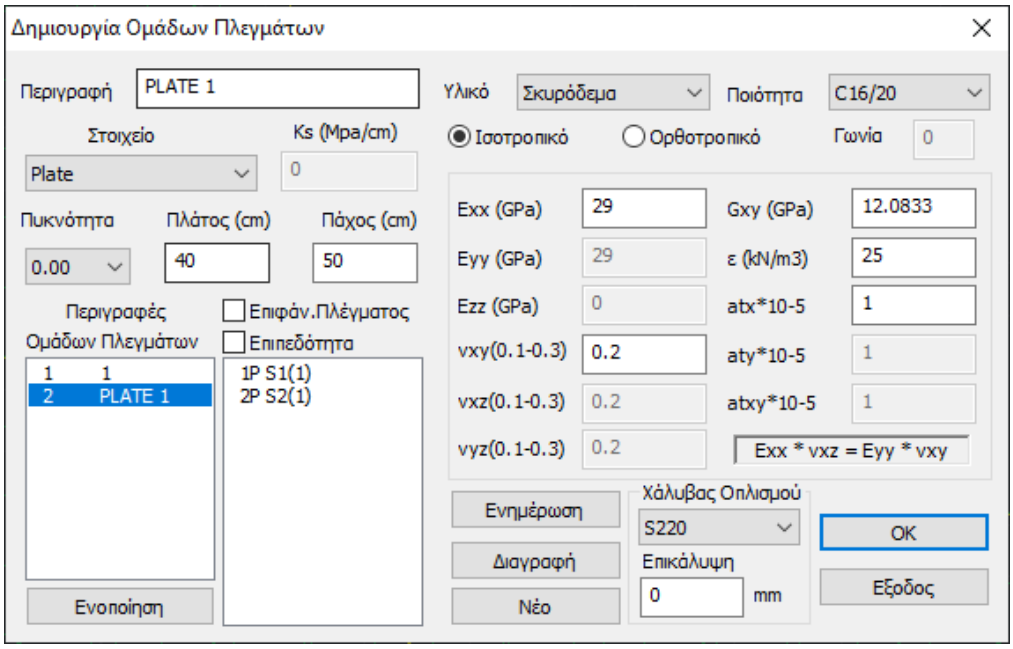
and the entire bottom line of the top wall



If we create the grids, we have the usual problems:

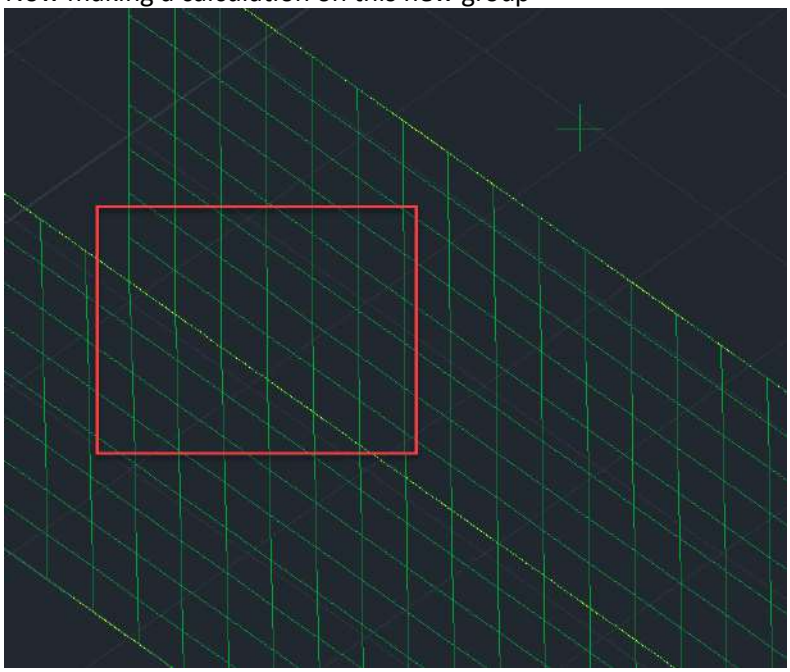


What were we supposed to do until now? Erase the short line and break the long line.
But now if we apply the "Consolidate" command by selecting the one and only main group



the program creates, based on what was mentioned above, a new main group with two sub-surfaces BUT with new correct contours.

Now making a calculation on this new group



we see that now the meshes of the subsurfaces have been connected correctly!

4.2.1.4 Consolidation of grids in cases where a 2nd dwg is imported in masonry studies

In cases where in masonry studies a 2nd dwg is imported, the surfaces are not included in the 1st group and a second independent group is created.
The procedure to follow when you have two or more floor contours from different dwg is as follows:

- Import the first dwg,
- you do, as you know, a face recognition and create the ground floor.
- Then you bring in the second dwg,
- you identify faces and "glue" the first floor on top of the ground floor.

You now have two main groups and lines that are identified and/or need to be broken. Same process for as many floors as I have.

Finally, using the "Consolidate" command, you select all the main groups that have been created and create a new one that includes all the subgroups with their outlines now as they should be.

If you want, you can now erase the original groups and their outline lines.

4.2.2 Outer Limit with Lines-Bow

Instruction to set the **Outer Limit** of the surface.

OBSERVATION:
If there is no grid, the dialog box for defining the grid as previously described opens. If there is a defined grid, it is used as described below.

To define the outer boundary of the surface grid, you first draw it with the help of lines, polylines or arcs. Then select the command and:

- If the outline is closed and there are no common lines with other outlines and/or branches, then it is sufficient to left-click on one of the lines to set it and right-click to complete the command.
- If the contour is closed and there are common lines with other contours and/or branches, then left-click and point to the lines of the surface contour one by one.
- The program provides automatic error detection in finding closed contours for the definition of surface contours and displays in red X the points where the selected contour is not closed.

- If the grid consists of more than one subgroup, follow the procedure in the previous example.
- If you want to delete an external boundary, you must delete the corresponding grid group you have created.

4.2.3 External Boundary with Points

The definition of subgroup surfaces can now also be done using points - vertices of the contour, a command particularly useful in cases of complex floor plans and facades where lines are not easy to select. Contours are automatically created and automatically placed in a new layer named "Plate 3D Line".

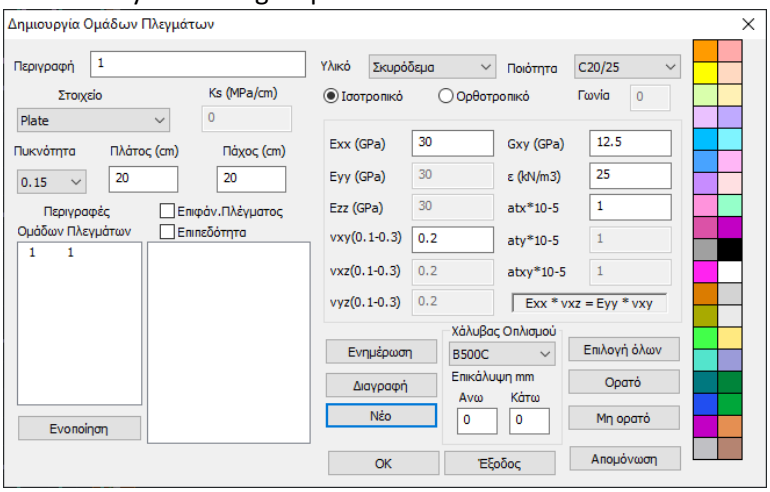
Added a new command to the 3D surface menu 

The previous command was renamed to "Outer boundary with Lines-Arcs".

This command obviously cannot include arcs but can include contours that only include lines.

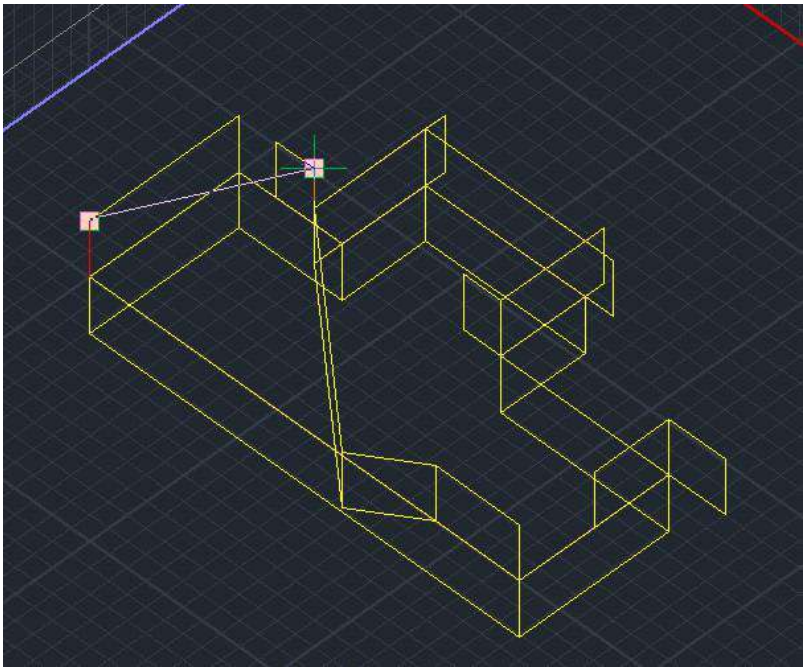
Let's look at the use of the command with an example:

You create your main group as usual

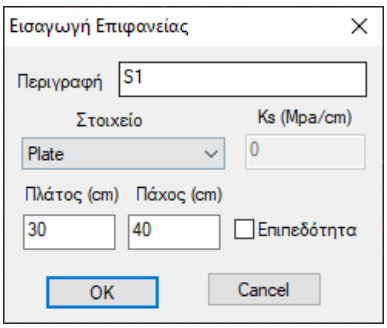


and define the contours of the sub-surfaces that make up the sub-surfaces.

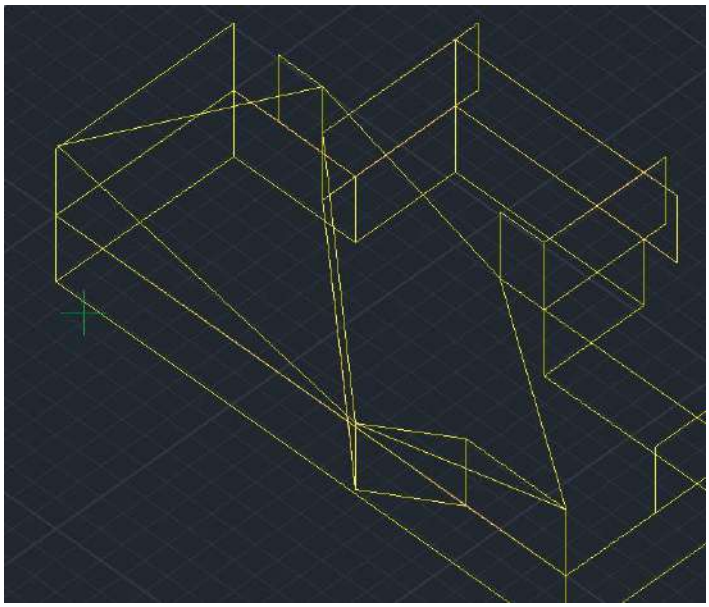
You select the "Outer boundary with points" command and start, using the osnaps that are convenient, to point successively to vertices of the contour of the sub-surface you want to define, creating an elastic line to see where you are moving:



When you reach the penultimate point of the outline press the right button to close the outline and the familiar sub-surface finalization dialog box will appear.

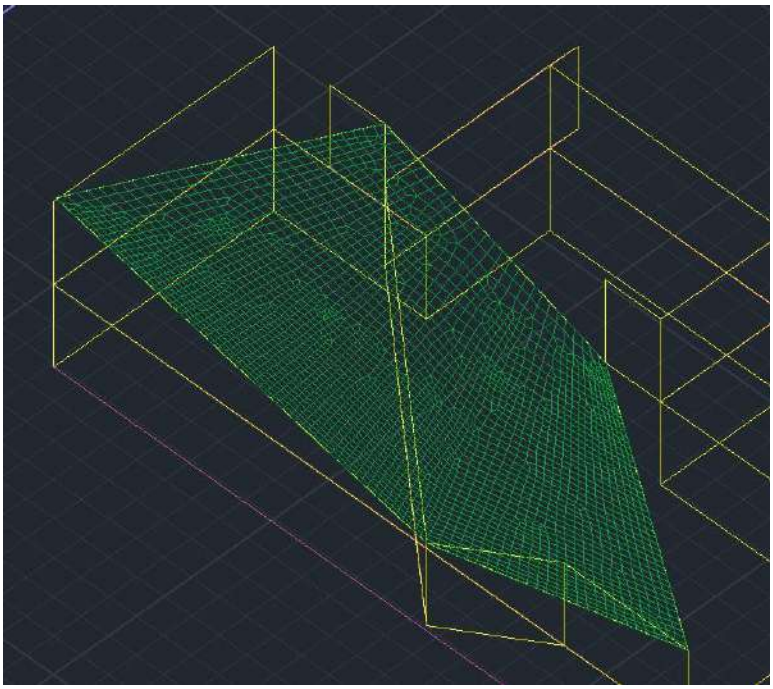


By selecting the OK automatically created outline is displayed



This one is a random warped contour in space.

With the calculation of the surface



the corresponding grid is created.

The process behind this command is that the program, upon inserting the points, automatically creates the contour with lines which are automatically placed in a new layer called "Plate 3D Line".

OBSERVATION:
To avoid errors, when defining an outer boundary with points on the surface finite elements, and when there is already a boundary with lines, the program does not allow a second boundary to be created.

4.2.4 Holes

Command to specify contour contours of contingent **Points** on the grid surface.

Call the command and define the outline of the hole, as for the outer boundary, using a "line" or "polyline". Then you select the command and point, with a left click, to one of the lines of the hole outline. Right click to complete.
Using the "Calculate" command, which will be explained below, the surface grid is recalculated based on the hole as well.

OBSERVATIONS:

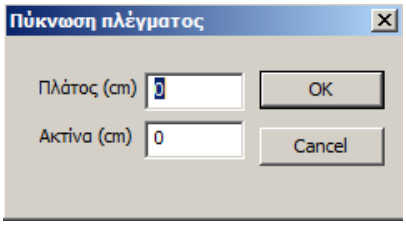
- 

The () next to the S symbol of the grid indicates the existence of a hole, while the number inside it defines the surface to which it belongs.
- *The definition of the holes can be done afterwards, after the surface grid has been created. Using the "Calculate" command, which will be explained below, the surface grid is recalculated taking into account the hole.*

4.2.5 Point

Command to define **Points** within the surface grid which will be densification points.

You select the command and define the thickening area around the point. Then, you point to a point within the surface of the surface. You select "Calculate" and obtain the thickening.

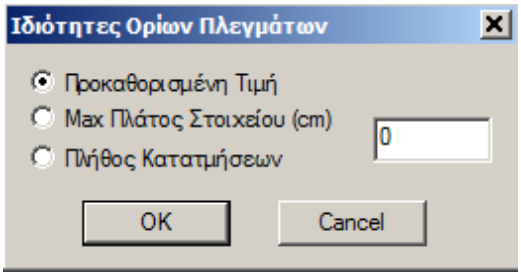


OBSERVATION:

Point definition can be done afterwards, after the surface grid has been created. Using the 'Calculate' command, which will be explained below, the surface grid is recalculated based on the point as well.

4.2.6 Edit

Command to **Edit** the mesh or meshes of the finite surface elements you have already entered. This editing must be done after creating the surface mesh and before creating the mathematical model. Using the command displays the following dialog box

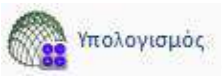


"**Default Value**": you can specify a specific number of surface elements that your grid will contain. You select it and in the field to the right, type the number of elements you want. Then you press the "OK" button and point the mouse successively to the sides of the contour. Press right mouse button to indicate that you have completed the selection and then call the "Grid Calculation" command. In the dialog box that appears, press the "Calculate" button. The surface grid is redefined based on the number of elements you specified.

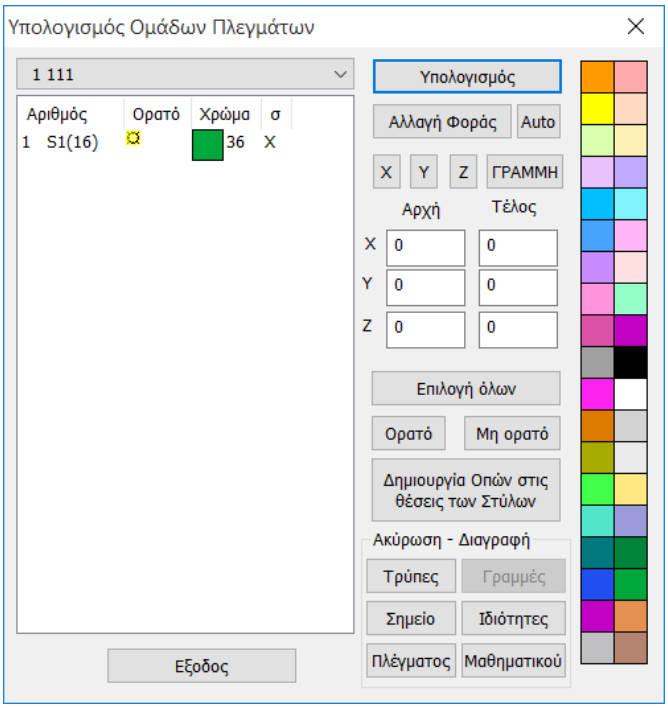
"**Max element width**": type in centimetres the maximum width you want your surface to have. Press the "OK" button and point with the mouse to one or more sides of the surfacer outline where you want the surfacer to have this maximum width.
Finish your selection with the left mouse button and then call the "Grid Calculator" command. In the dialog box that appears, select the grid (it turns blue) and press the "Calculate" button. The surface grid is redefined based on the maximum width of the elements on the sides you specified.

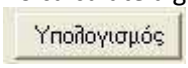
"**Number of segments**": enter the number of segments (not the total number of items). Press the "OK" button and point with the mouse to one or more sides of the surface contour, which you want to have the number of segments you have previously defined. You finish your selection with the right mouse button and then call the "Grid Calculation" command. In the dialog box that appears, select the grid (it turns blue) and press the "Calculate" button. The surface grid is redefined based on the number of segments you specified.

4.2.7 Calculation

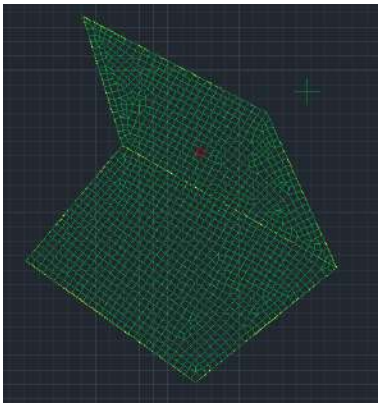


Command to Calculate the 3D surface grid, taking into account the contour you have previously defined and any holes and points. Using the command, the following dialog box appears



To calculate a grid, after selecting it from the top list and displaying its subgroups, press the 

button. The same procedure is followed for the other grid groups. The result of the simulation is shown in the image below:

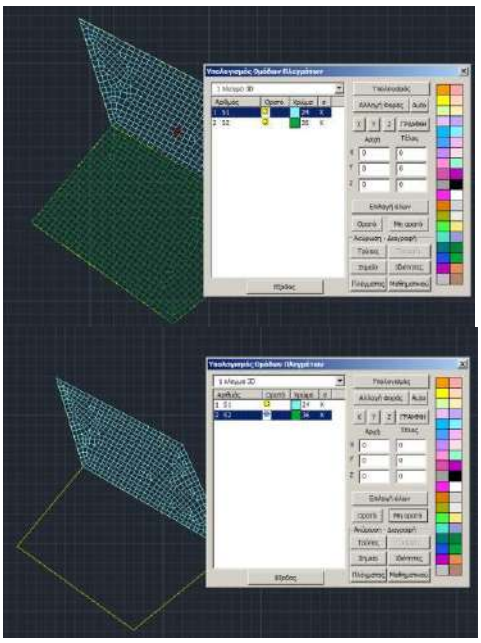


OBSERVATION:

In the above way, only the grid is created. The mathematical model of the surface has not yet been created, is done with command in "Tools"
>> "Calculation".

The same dialog box, in addition to the Calculator, contains additional useful tools:

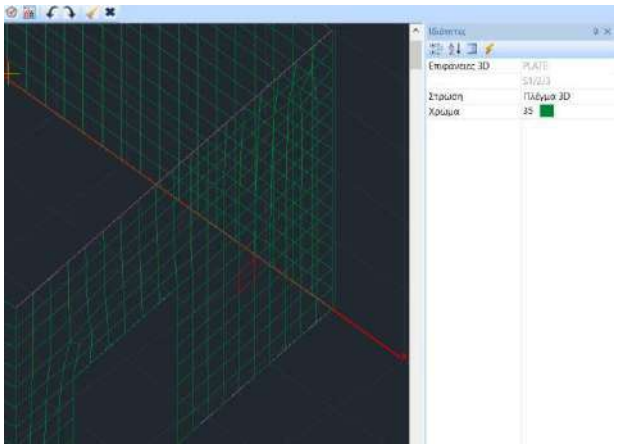
- To change the colour of a grid surface, select it from the list and click the colour you wish from the palette of colours.



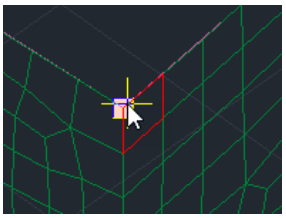
- By From Using of keys Ορατό, Μη ορατό, a grid surface may or may not be displayed respectively. You select the one you want to be visible or not visible and press the corresponding button. The indication under the column Ορατό changes from (visible) to (not visible).

NEW POWERS and SHMANTIC OBSERVATIONS:

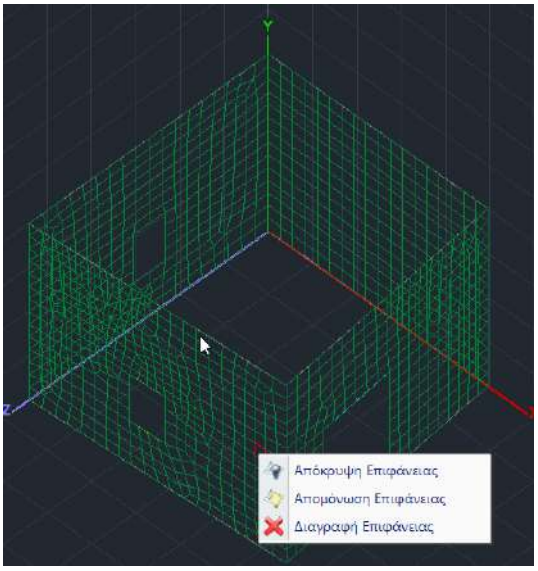
- A sub-surface mesh may be referenced before the mathematical model is created. By left-clicking on the mesh, the name, color and layer. The last two can also be modified.



- It can be pulled on the tops of the grid (only the nearest), before creating the mathematical model.



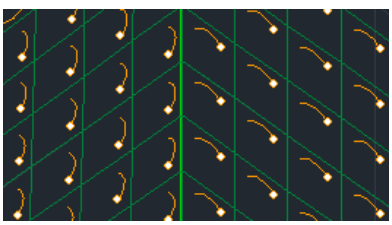
- By right-clicking on the grid, a command menu appears from which Hide can be done -
  Isolation of the sub-surface mesh before creating the mathematical model.
  Capability particularly useful when defining the outer boundary for 3D surfaces, for easier identification of sub-surfaces



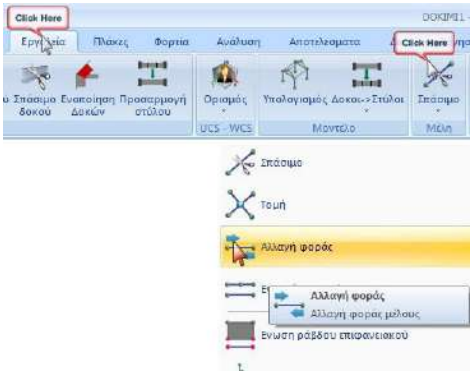
- It is now possible to select multiple subgroups in the surface meshes with Ctrl and with Shift
- Automatic deletion of problematic elements (coincident nodes or nodes that are identical) of the finite surface elements is performed

- The local axes are now appearing even before the creation of mathematical model and
  are activated through the "Appearance">> section "Switches" >> "Local Axes".

- By using the "Change Face" button the local axes are changed. The above command changes the direction of the surface total.

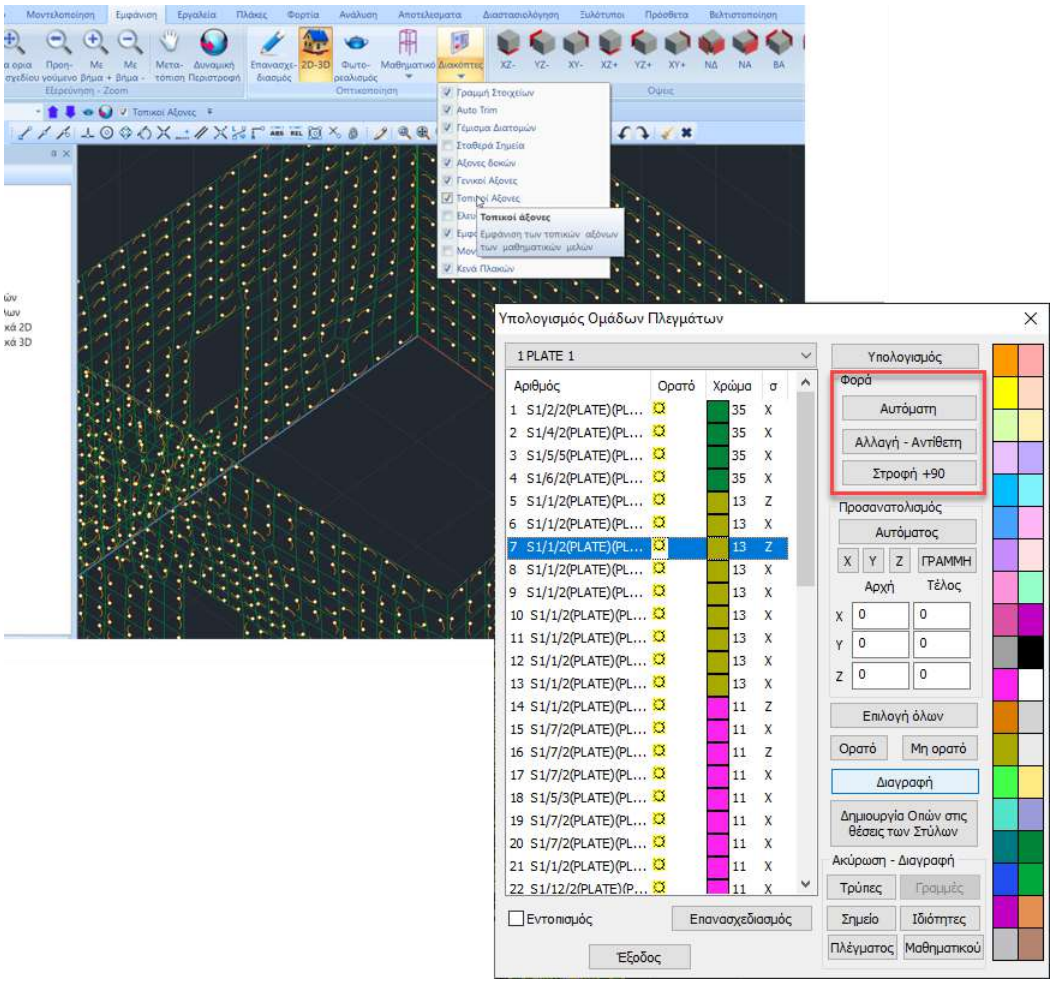


If you want to change the direction of individual elements in the grid, use the command "Tools">> "Member ***Tools****>> "****Change Face****".*



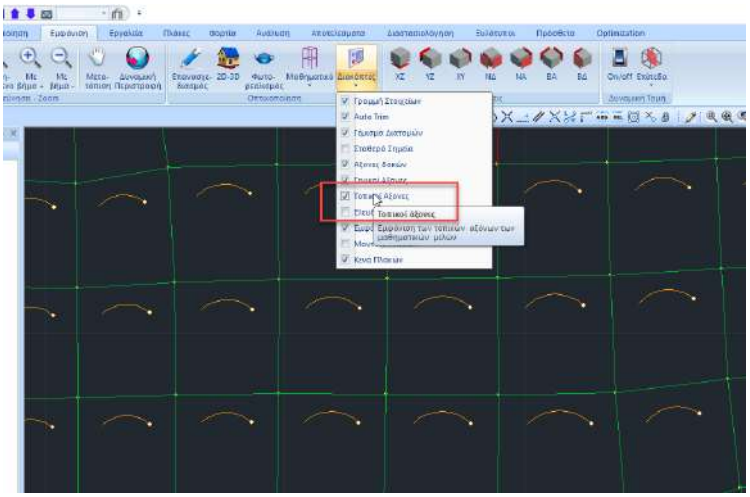
- With the "**Auto**" command the program adjusts the local axes of the surface axes throughout the grid so that they have the same direction.

- In the finite surface and solid elements the Fear, the Auto option and the display of local axes now works only with the existence of the mesh and before the creating the mathematical model.

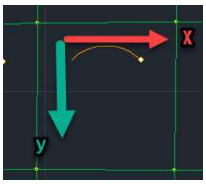


Also added the Change - Opposite button changes the local from +x to +y and vice versa, while the Rotate 90 command rotates the gyro homophore 90 degrees with each press.

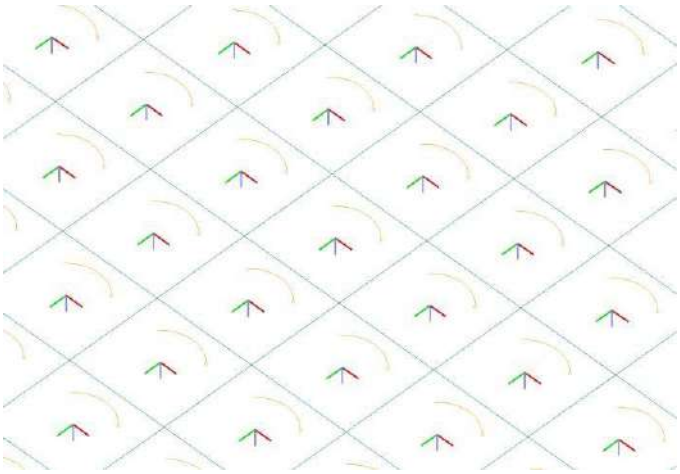
- Every time you define a 3D surface grid, to automatically redefine the correct direction of the surface elements, you should ALWAYS select the "**Automatically**" command.
  To display the local axes of the surface elements, activate the corresponding option within the **Switches**.



The arcs that appear define the local axes of the surface features, according to the clockwise screw rule.
The direction of the arc indicates the **x** and the point at its edge, the direction.



In the new version of the program, the local axes in the surface finite elements are now displayed in 3D for a better overview, following the well-known convention of the right-handed screw and in colours:
Red axis (Red) : X Green axis
(Green): Y Blue axis (Blue): Z

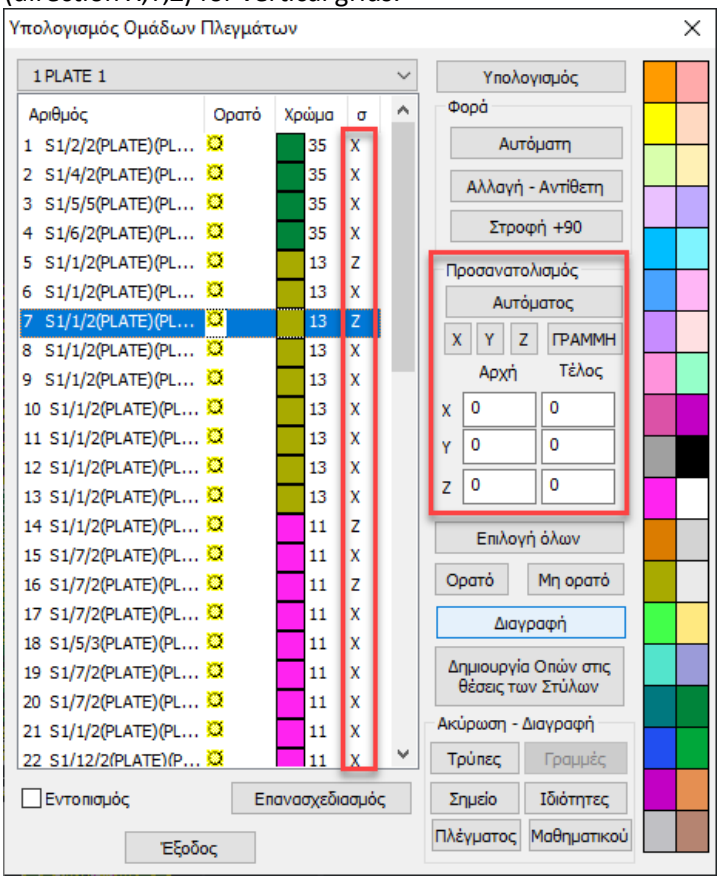


Also, the display of local axes for both linear and surface finite elements has been enabled in the Results section.

- Necessary for determining the main direction of the reinforcement, is the determination of the direction for the vertical grids.
  To automatically redefine the correct direction of the surfaces in relation to the X and Z global axes, ALWAYS select the use of the "**Automatically**" command.

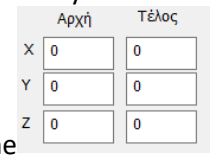
  The field 

, allows the manual definition of the main direction of the reinforcement (direction X,Y,Z) for vertical grids.

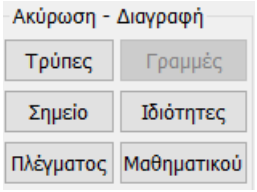


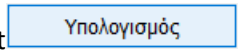
- For vertical grids // on the X-axis : I select the grid and  from the list and the column ''s'' is updated, respectively,
- For vertical grids // on the Z-axis: select the grid and from the list and the column ''s'' is updated, respectively 
- For meshes that are not // on either X or Z if I do nothing the program will project the resulting armature by reducing it to the 2 main axes. Alternatively I can use the command . 

♦ The 

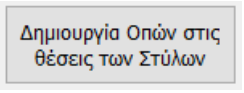
key works in combination with the 

field, where you define the coordinates of the beginning and end of the line, which the program will consider as the main direction of the reinforcement, in cases where the surface is not parallel to the universal axes, hence its reinforcement.

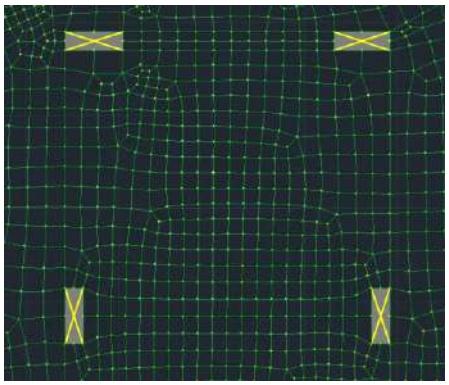
♦ The "Cancel-Delete" field allows you to delete holes, lines, points, as well as, the properties, the mesh or its mathematical model, that you have already created and wish to delete. 

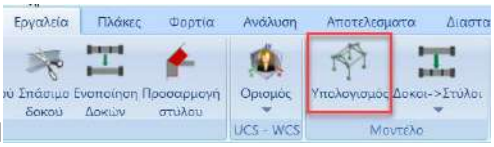
Then, if you select 

the program will recalculate the grid with the new data (e.g. if you delete the "lines" the grid will be calculated without the lines)

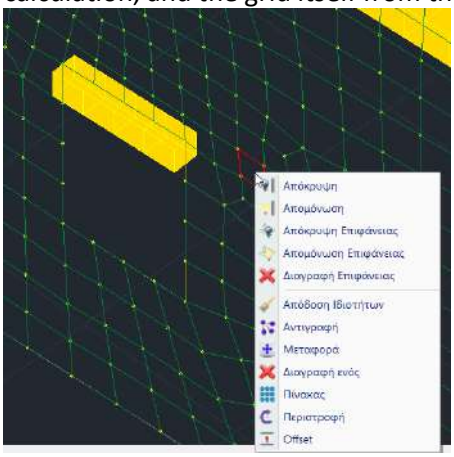
♦ The **Create Holes at Column Positions** command 

is for "**Flat Plates**" and allows automatic creation of holes in the flat plate area where there are columns. (see Chapter 9 - Flat Slabs).

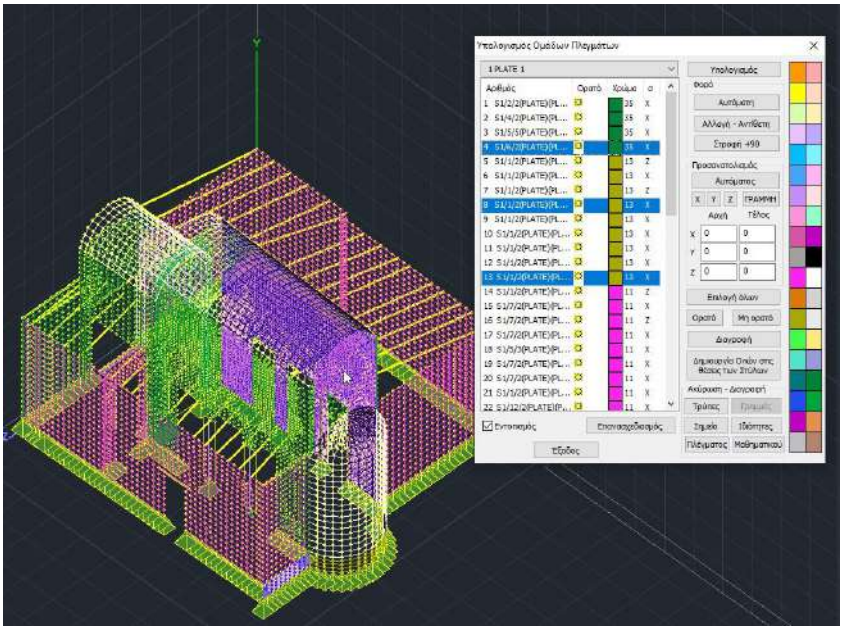




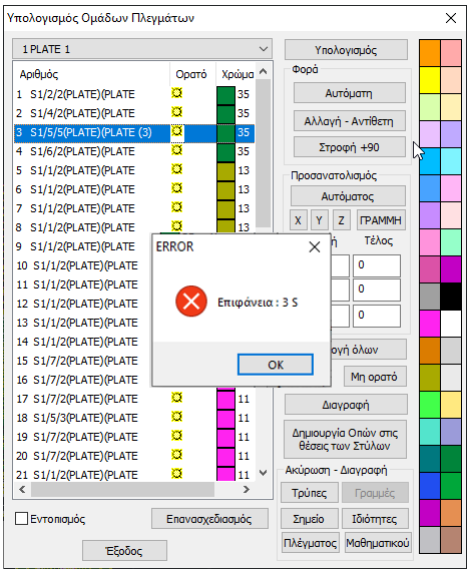
After creating the Mathematical Model you can insert linear members to simulate diaphragms, beams and columns in cases of load-bearing masonry and generally intervene in the mesh.
With a right click you can Hide or Isolate a finite or an entire surface, as well as delete it.
The **Delete Grid** command allows you to delete both the mathematical model (if it exists) and its calculation, and the grid itself from the list.



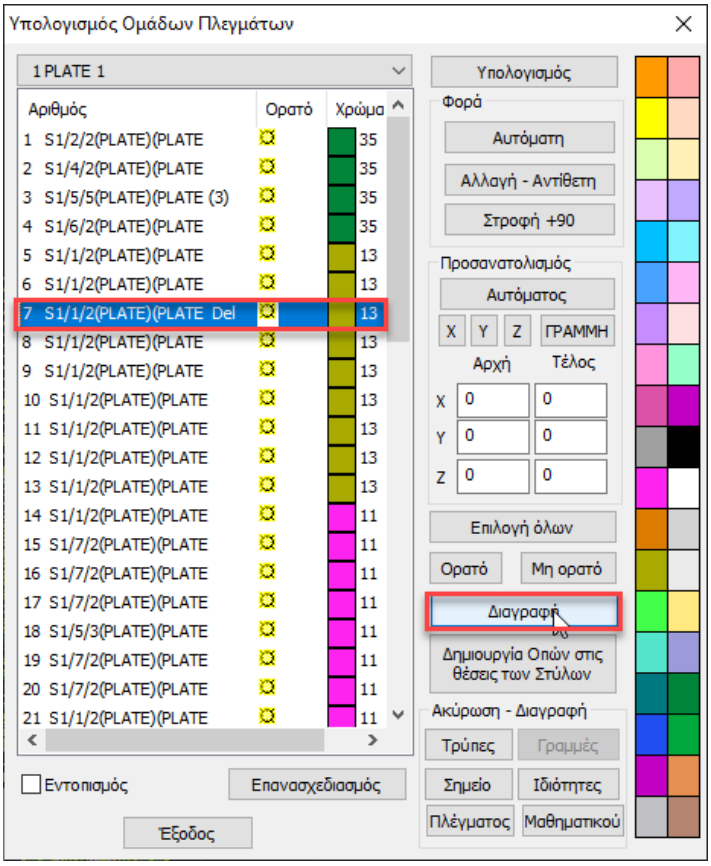
- The new version of the program gives you the possibility to display the areas of the surfaces you have defined, before and after the creation of the meshes, in order to have direct supervision and better control of your model. Select the "Locate" command and left-click the surfaces you want to locate from the list. Then select "Redesign" and deselected.



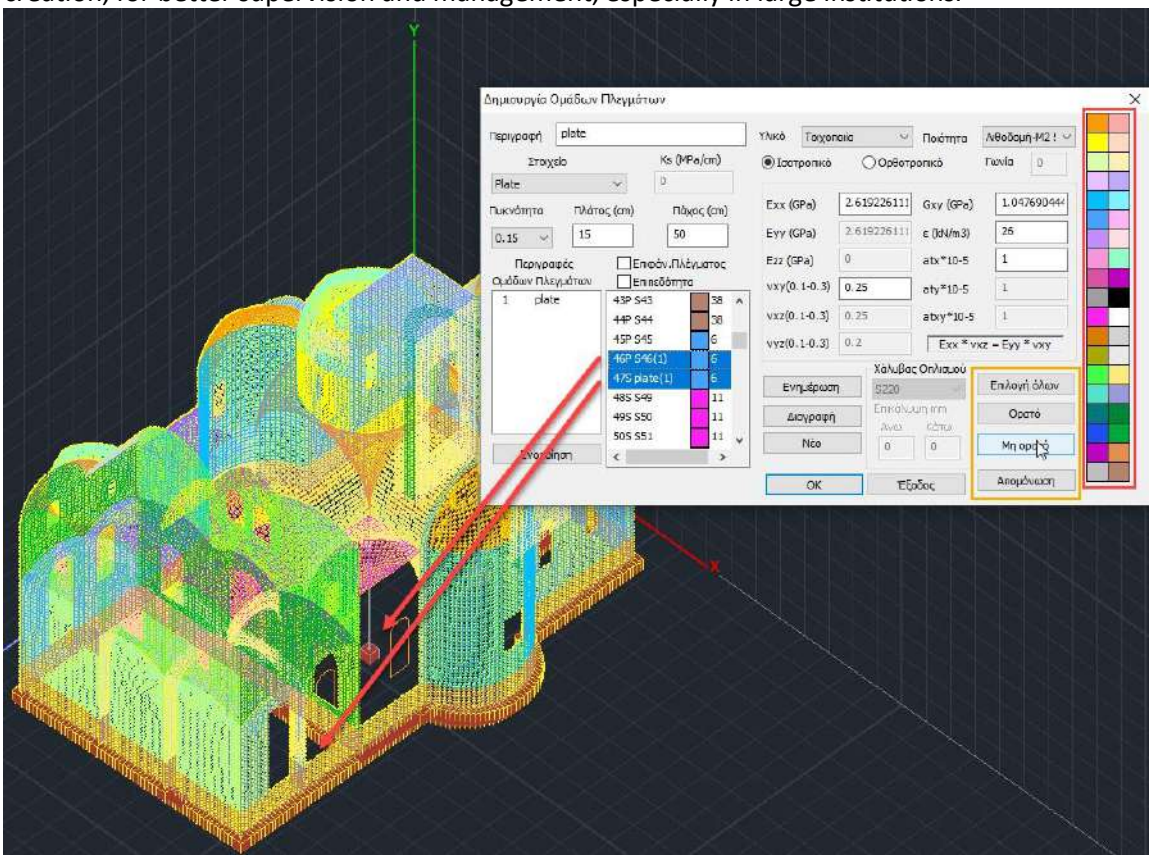
- A new feature that offers great convenience for locating the surfaces of a mesh that for whatever reason does not allow the program to calculate it.



- The Delete command allows you to delete one or more surfaces directly from the list of the grid calculation.



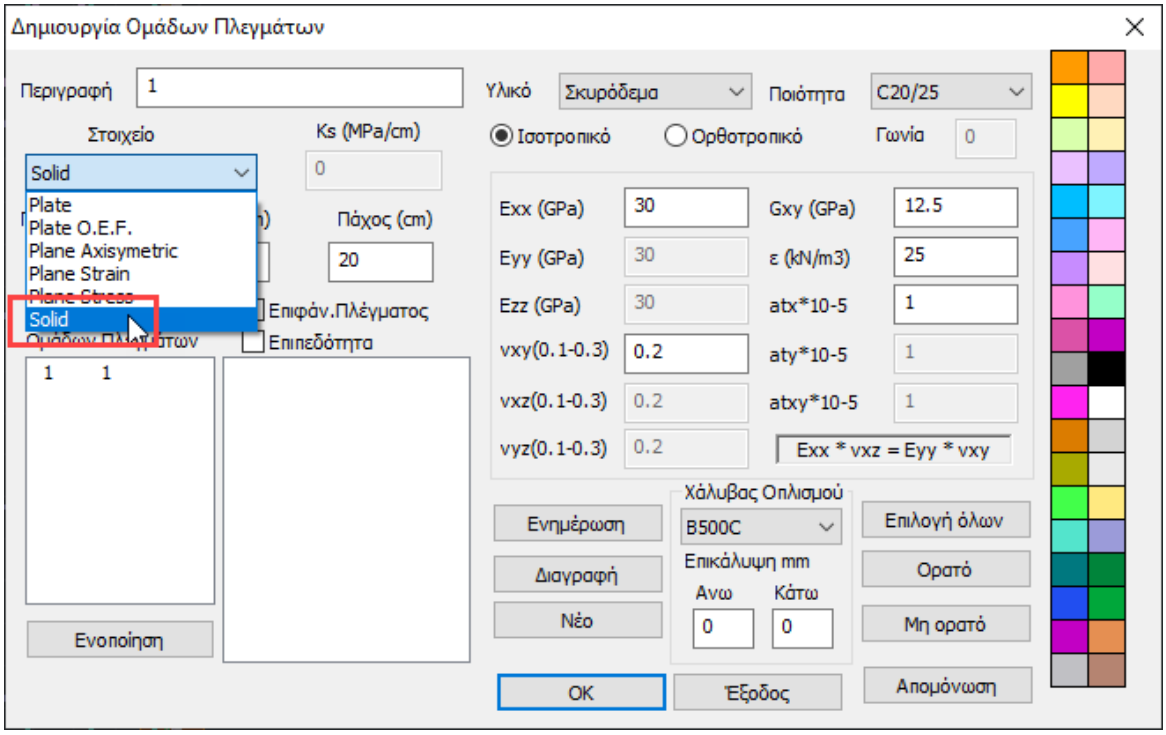
- In addition, there is the possibility to change the color, isolate, and hide - show one or more sub-surfaces through the definition dialog box and creation, for better supervision and management, especially in large institutions.



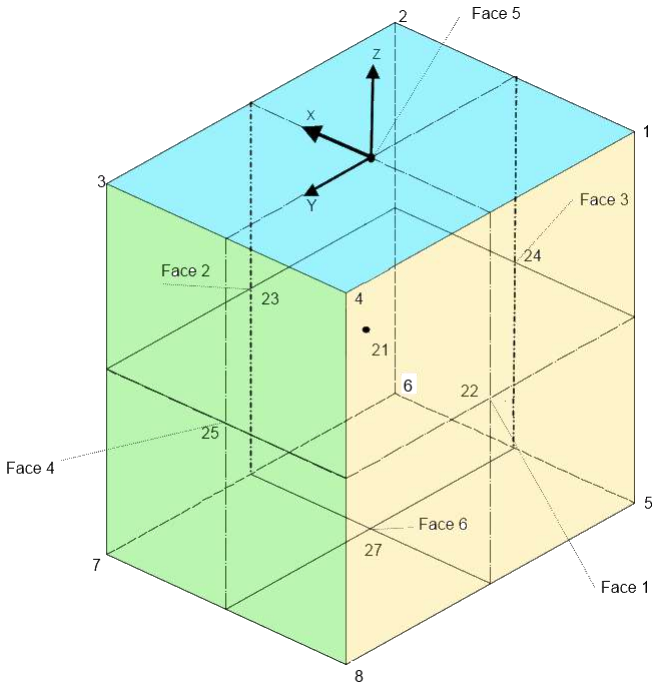
- The surface subgroup selection command

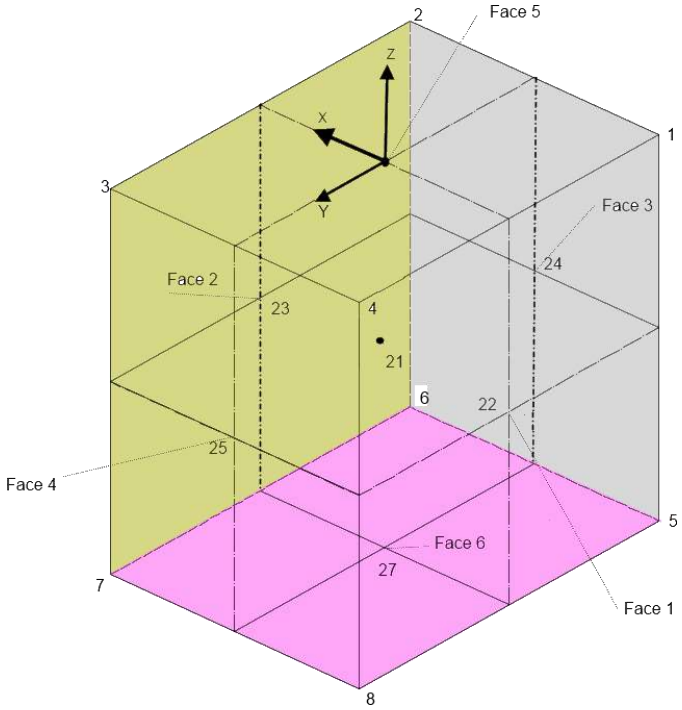
This new option allows you to select a whole subgroup of surface by selecting a single element of it, a very useful tool especially in complex operators and when you want to make a specific subgroup Delete, Change direction, Change material etc.

SOLID ELEMENT



This element can be defined in its generic form, with a minimum number of nodes of 8 and a maximum number of 21.





In SCADA Pro it is defined as an octagonal element with 6 seats with the numbering and axis convention shown in the figures above. The first face with nodes 1,2,3,4 is initially defined (always counterclockwise) and based on the thickness given by the designer, the face with nodes 5,6,7,8 is generated in correspondence with the first one (the "generator") and the solid element is created. This element has 6 faces with the numbering shown above. At point 21 is the centre of gravity of the element (centroid).
The centres of faces are:
Face 1: 22 (Face 1 is perpendicular to the back side of X)
Face 2: 23 (Face 2 is perpendicular to the front side of X)
Face 3: 24 (Face 3 is perpendicular to the back side of the Y)
Face 4: 25 (Face 4 is perpendicular to the front side of the Y) Face 5: 26
(Face 5 is perpendicular to the front side of the Y) Face 6: 27 (Face 6 is
perpendicular to the back side of the Y)
In all the above points there are no nodes but these points are used to display the results as we will see below.

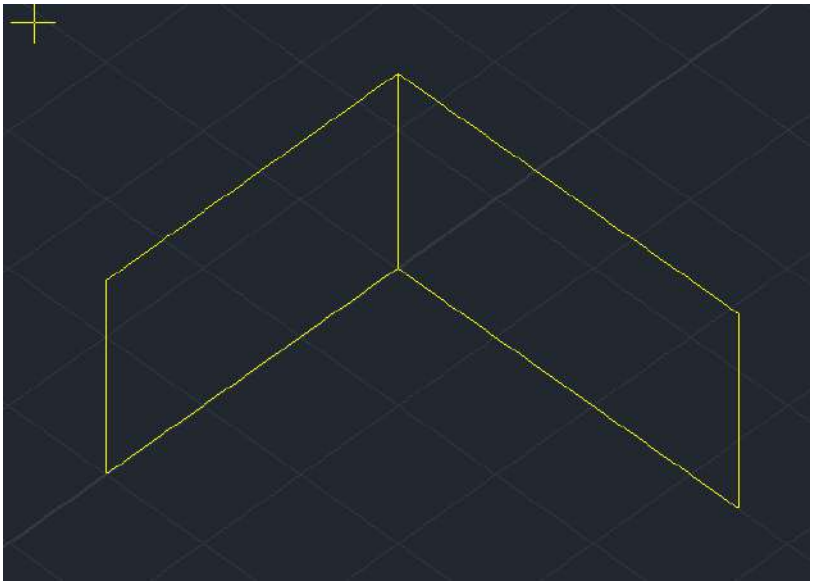
At the centroid and at the above centres (7 points in total) the results are displayed both in Post Processor (in the form of isosceles curves) and in the out file.

The nodes of the element have freedom of movement but their nodes do not develop turns.

ATTENTION
The surface 5,6,7,8 is always generated in the direction **opposite to** the local Z. This is important in order to know which side to generate the thickness of the wall. Let's illustrate this with an example.

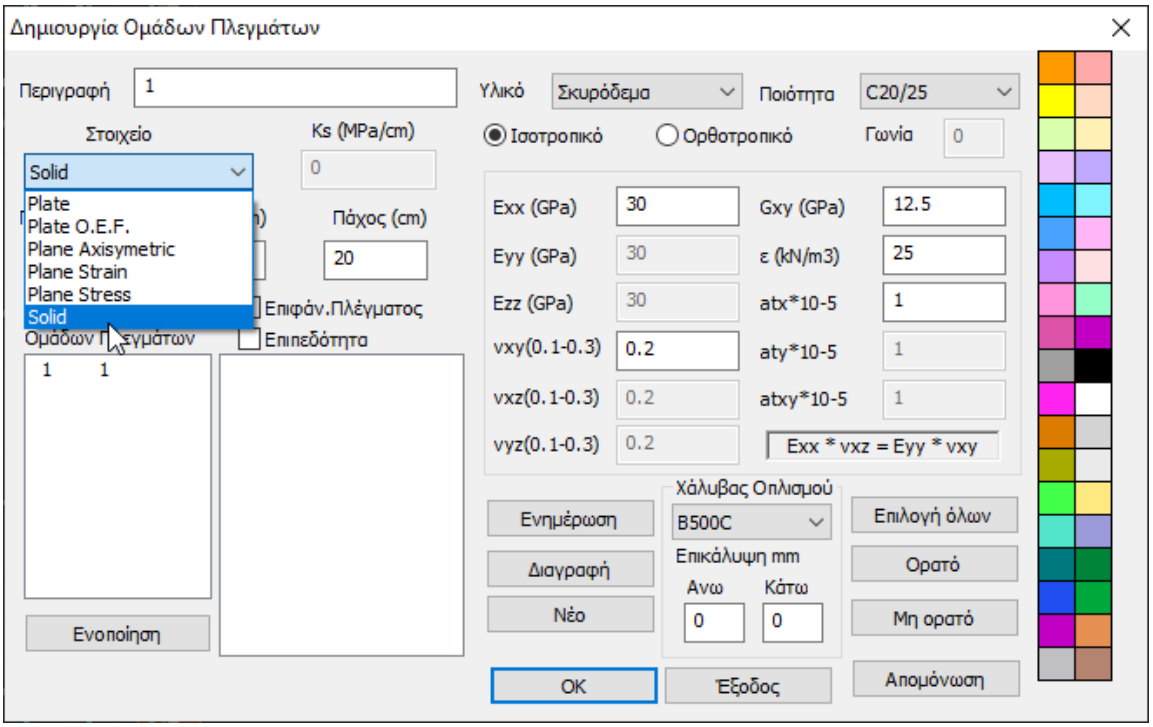
Data Import

In the picture below we have the outer contour of two vertical walls

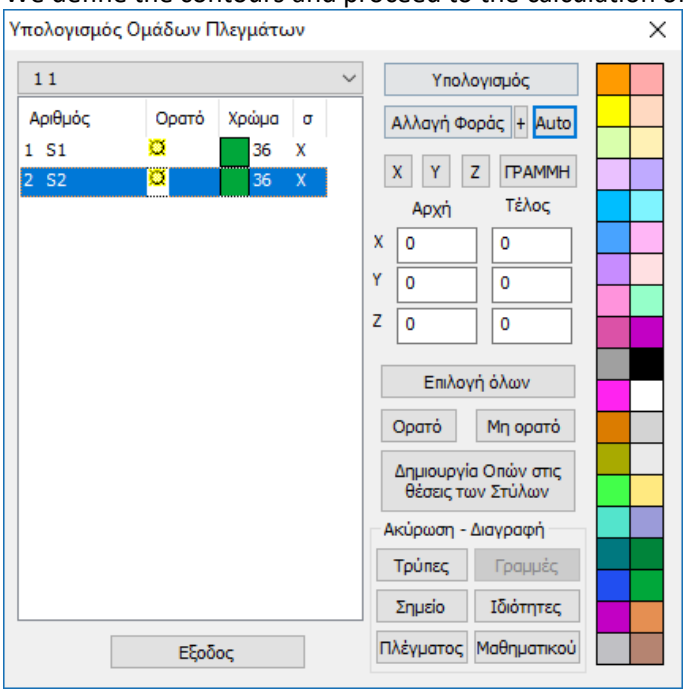


We want the walls to be created from the inside.

We create as usual a new group of superficial with type Solid



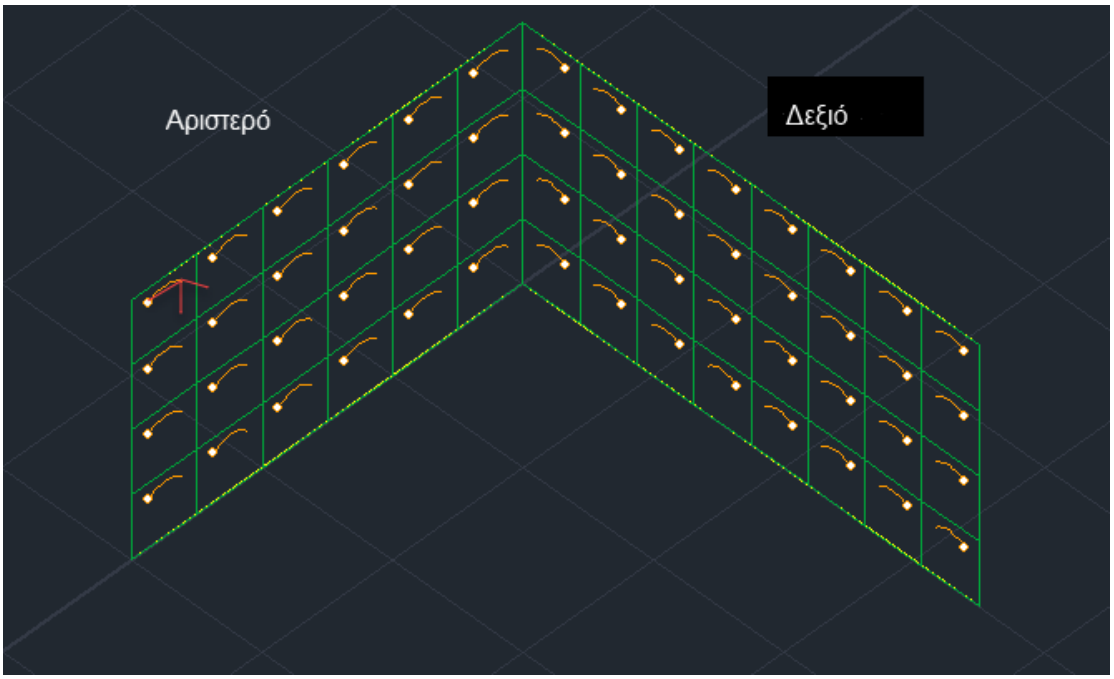
We define the contours and proceed to the calculation of the meshes.



OBSERVATION:

It should be noted that the option has been added so that the Change Face, Auto option and the display of local axes now only works with the existence of the mesh and before the mathematical model is created. Of course this works for all types of surface. Also, next to Change direction button (+) was added which rotates the tadpole concentrically by 90 degrees with each press.

We come back to our grid and the situation after and Auto is this

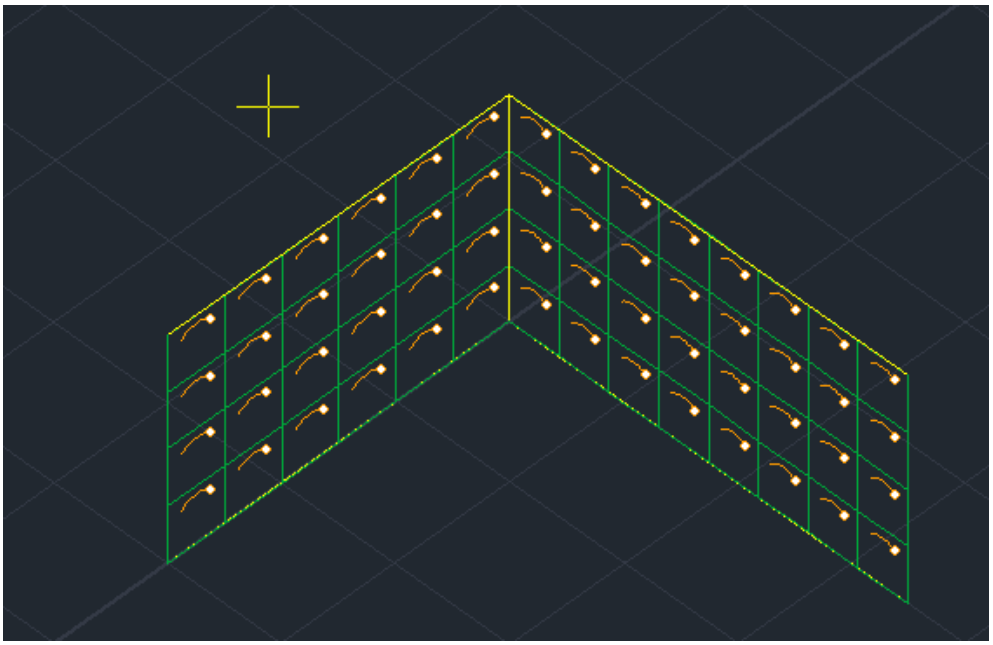


In the left grid you can see the local axes for the first element.
According to the above, and since the local Z is inward, the second surface will be produced on the outside, which is not what we want.
With the Change Face tool we change the overall times of the elements of this subgroup.

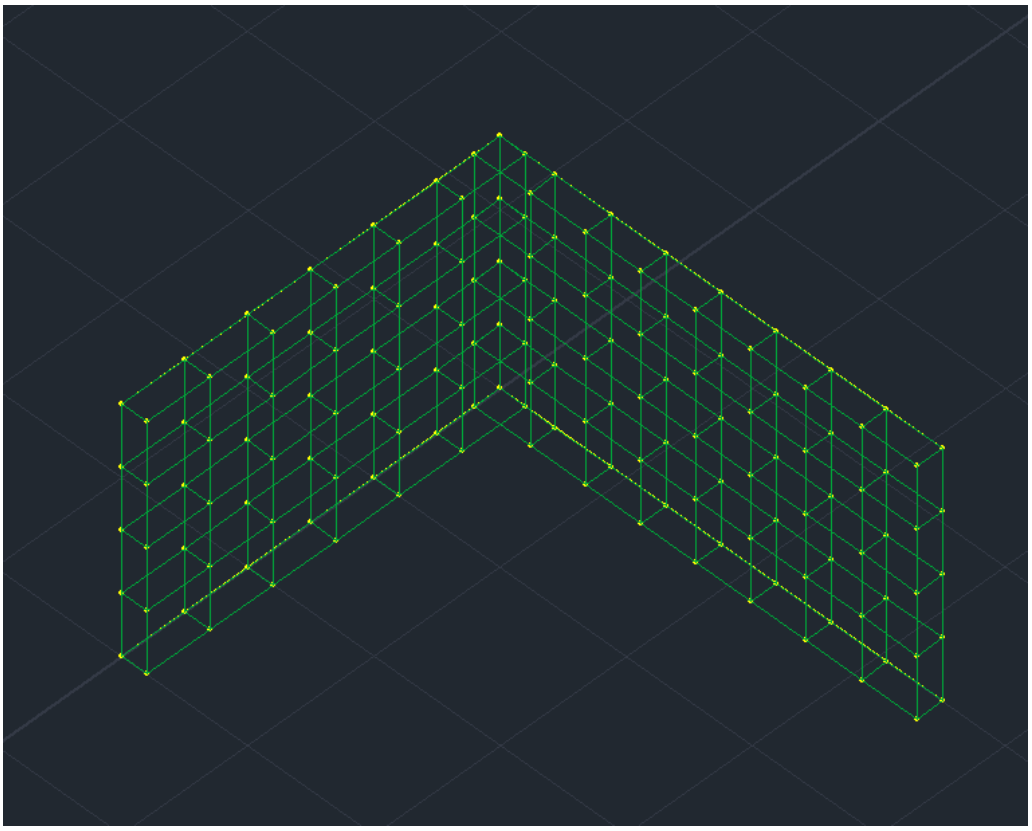
In the right grid the local Z is outwards so the second surface will be created correctly inwards.

OBSERVATION:
It is therefore very important to set the times correctly before creating the mathematical model, because changing times after the mathematical model is created is NOT allowed (the analysis does not run).

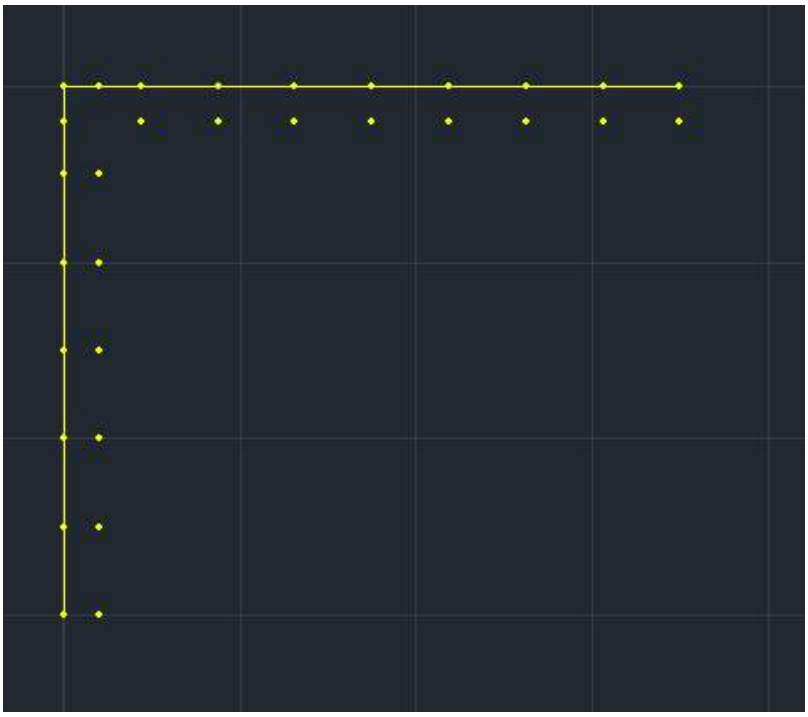
After the change of direction on the first surface, the following situation arises



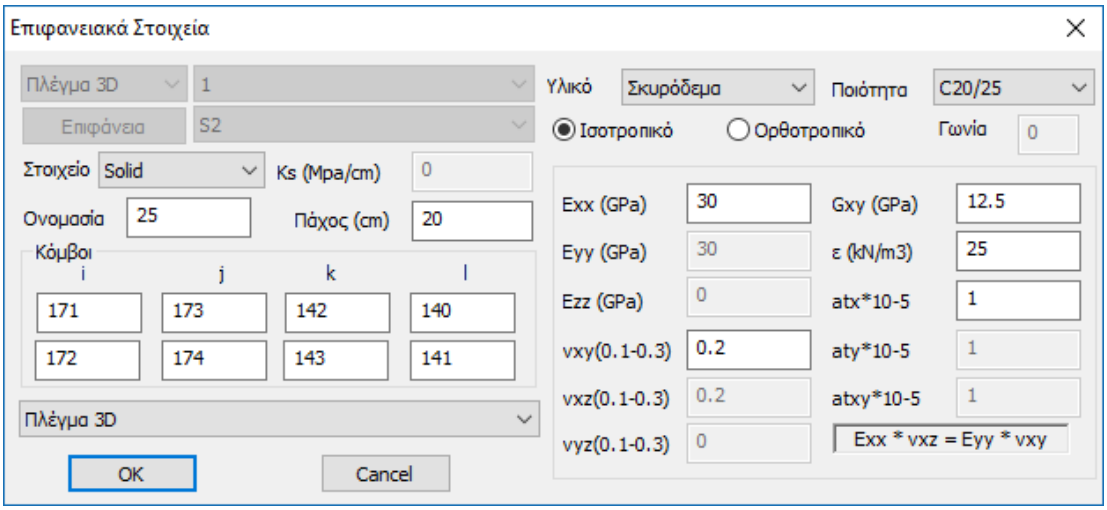
With the creation of the mathematical model, the surface Solids are created using the technique described above.

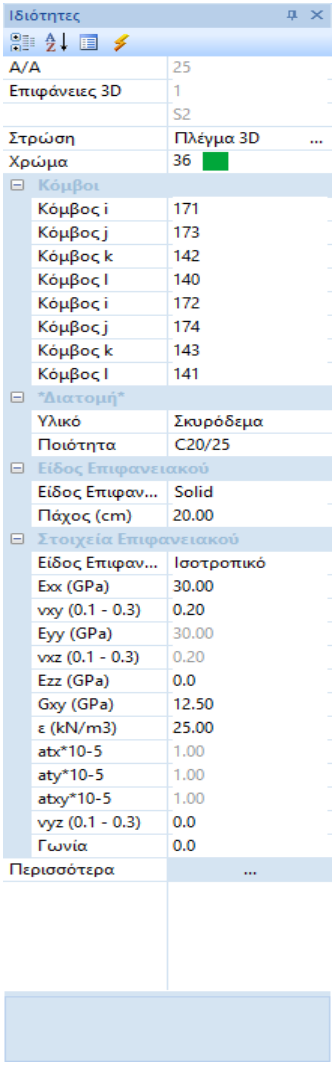


In plan view we can see that the walls have been created correctly inwards



Making a reference to a solid element normally shows the elements of the solid element



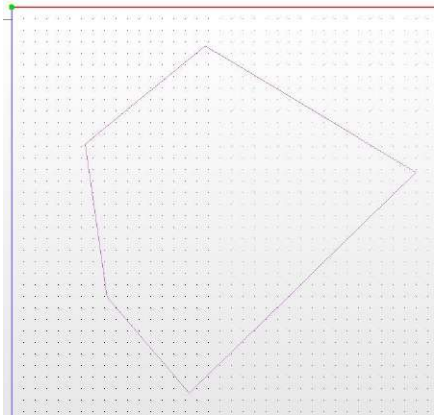


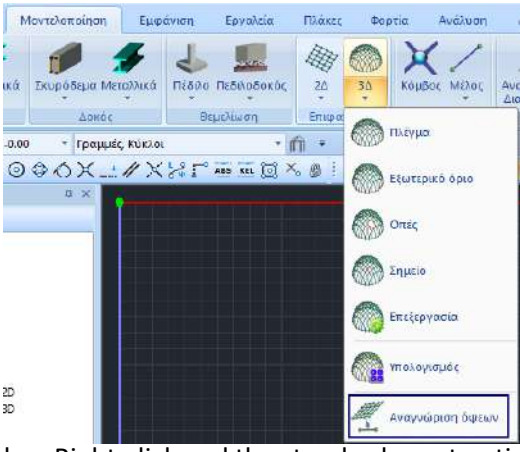
4.2.8 Face recognition

SCADA Pro allows you to create any contour for the masonry and with the help of standard constructions to "build" your carrier easily and quickly.

The procedure is as follows:

- Import a floor plan from an existing .dxf or .dwg file or using the commands within the "*Basic*" section draw a closed surface on the XZ plane of the "**Draw**">>"**Line**">>"Polyline" desktop create a surface right click.





- Select the command at Section "**Modeling**">>"**Surface3D**">> "**Face Recognition**", and with Window  you select the whole floor plan. Right-click and the standard constructions box opens.

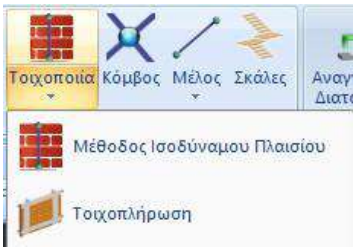
⚠ *(see &6.2.5. Typical Structures - Masonry)*

5. Members

The "Members" command group contains the commands for specifying and importing:

- MIPs and gut payments
- Nodes
- Members (Mathematical-Surface)
- Scala

5.1 Masonry:



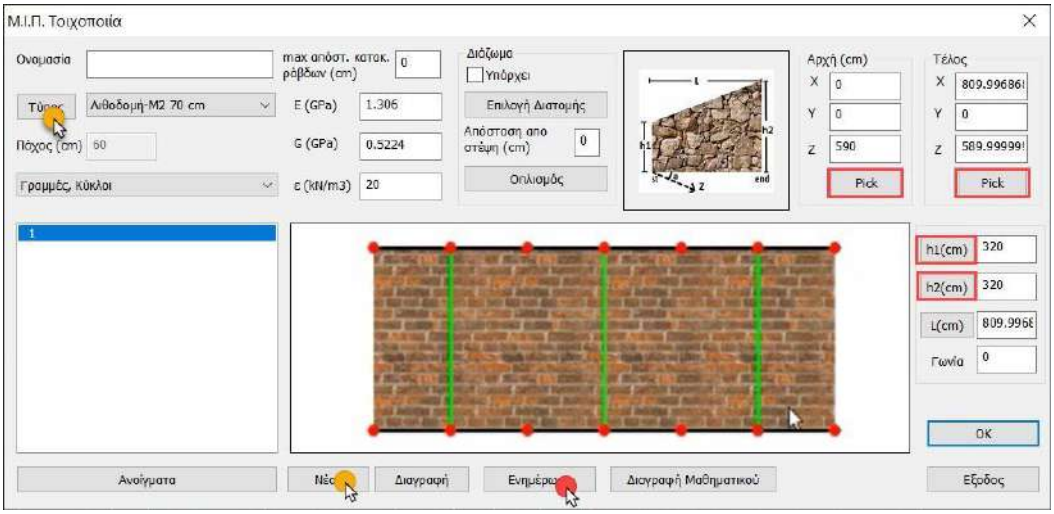
5.1.1 Equivalent framework method

Through the Equivalent Framework Method command.
First the walls are defined:

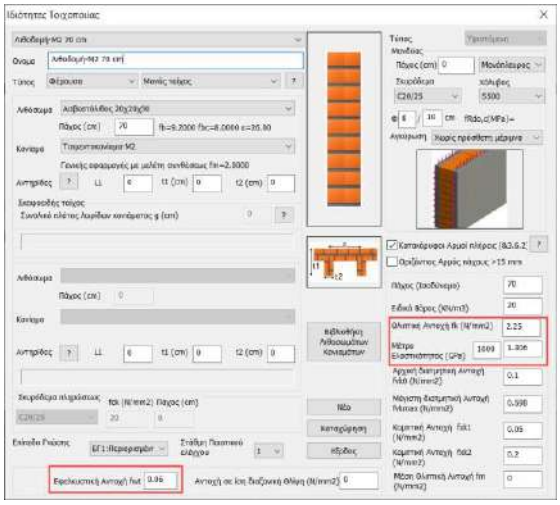
- Press
- Dormitory
- Geometry

The process includes the following steps:

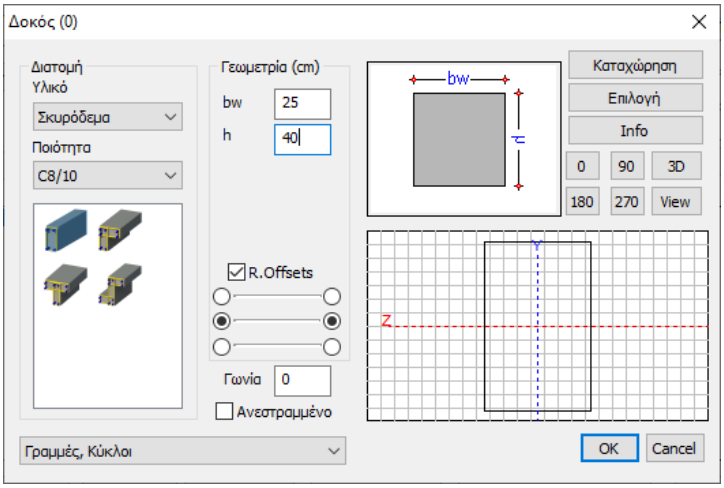
1. We give a name to the wall



2. We select the type of masonry from the pop-up menu or click on ¨Type¨ to open the masonry library and define a wall of our own,
   defining lithosome, mortar and compressive and tensile strengths.



3. We define the geometry of the wall graphically by selecting Pick and h1, h2 and showing the start, end points and heights at the 2 edges of the wall, with the help of pulls and left click. The fields are filled in automatically, and manual entry of values is also allowed.
4. In addition, the fields of the constants E, G, e are filled in automatically, with the possibility of modification by the user, as well as the values L and Angle.
5. ¨Max sender. Cat. The maximum distance between the bars is determined by the maximum distance between the
   vertical ribbed elements for the simulation of the equivalent frame columns.
   Keeping the value 0, the program calculates it by KADET. The user can specify a value of his own and then the program will take it into account.
6. ¨Diasma¨: I activate the checkbox when there is a tier and via the command ¨Cross-section selection¨ define the cross-section of the partition. Then, we indicate the distance of the diaphragm from the crown of the wall.



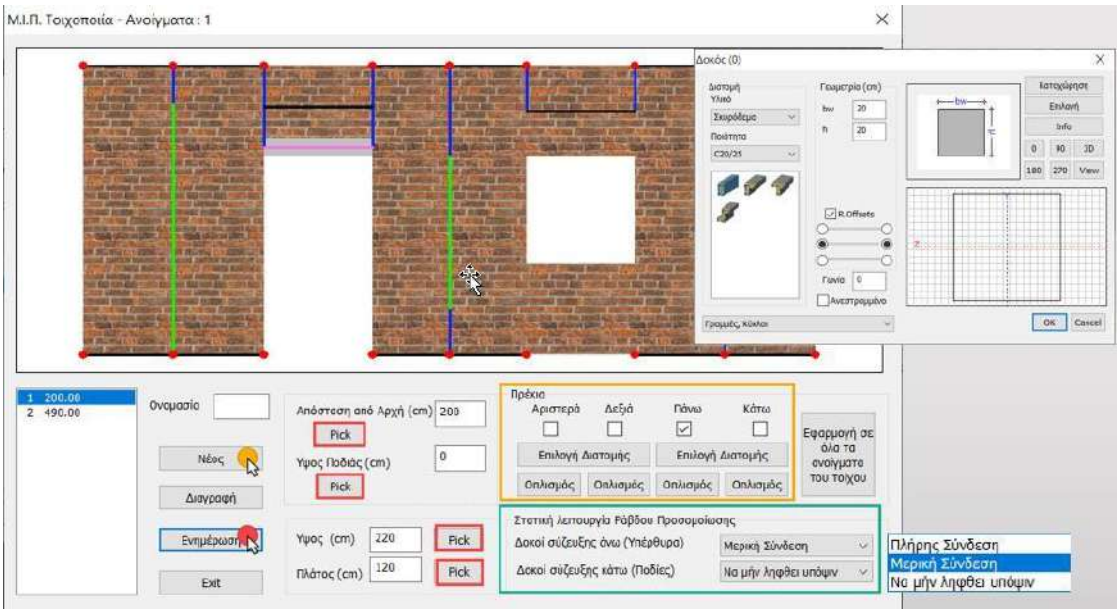
7. By selecting thecommand, the specified wall is added to the list. H ¨Delete¨ deletes the user, ¨Update¨ informs the user of any changes, ¨Deletion¨ "Projection" marks it graphically for easy identification on the vector. Mathematical¨ deletes the mathematical model (after it has been calculated of course, which is done after the walls and openings have been created).
8. The command opens the window for defining the openings of the selected wall.

Then the openings are defined:

- Geometry
- Pretzels
- Static Rod function

The procedure is similar to that of fixing the walls:

1. We give a name to the opening
2. With the help of Pick we graphically define the geometry of an opening
3. If there are Presses we select their position and their cross-section and if they apply to all openings of the same wall, then we select ¨Apply to all openings¨. openings of the wall¨, so that the same presses are automatically applied to the other openings we will make for the same wall.



4. In the field ¨Static simulation rod mode¨ we select one of three ways of participation of the transoms and legs in the mathematical simulation:
    - Full connection
    - Partial connection
    - To disregard
5. By selecting the command, the specified opening is added to the list. H "Delete" deletes it, Updateinforms it of any changes, the "Projection" marks it graphically for easy identification on the vector.

6. The process is repeated until all openings of the selected wall are defined.
7. ¨Exit¨ to close the opening window and continue with remaining walls and openings.

The modelling is completed when the definition of all walls and openings at all levels is completed and the calculation of the mathematical model is done. (See Inc. See Use Chapter F: 'WALL CONSTRUCTION BY THE EQUAL PLANE METHOD')

5.1.2 Wall filling

With this command you can enter the wall fillings of your construction according to the provisions of the CEE:

CANC. §2.1.4.2:

- *They are considered only for seismic actions.*
- *They are mandatory when they adversely affect the operator. CAN §5.4.3c:*

It is forbidden to place the wall fillings selectively, on certain floors or positions of the building, and not in their entirety, so that the morphology is transformed from non-normal to normal.

CAN. §5.9.2:

*In **elastic analyses** it is allowed to be considered in **a crosswise arrangement** (i.e. one diagonal is compressed and the other is flexed), giving each diagonal **half of** the stiffness EAR. In the **inelastic analyses** a pair of cross diagonals with full strain EAR each is used, but is only considered where it operates under compression.*

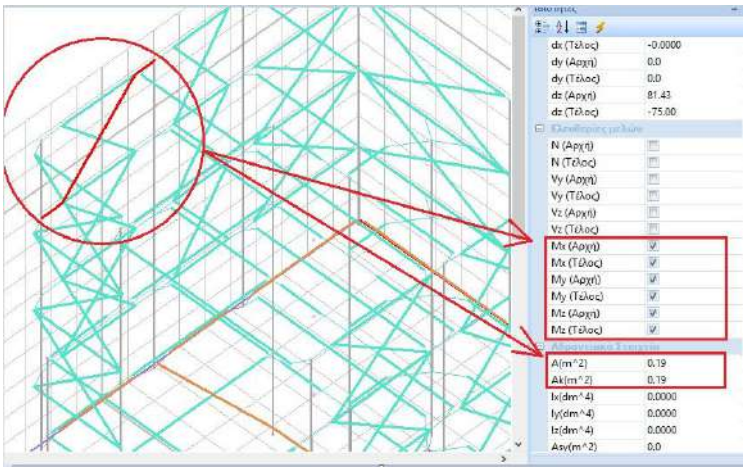
The simulation is performed with two diagonal bars with zero specific weight (since the loads of the wall fillings have been given as linear loads on the members of the beams) and with a surface area of dimensions according to the provisions of the CEE.

The wall fillings are inserted with cross bars with the following properties:

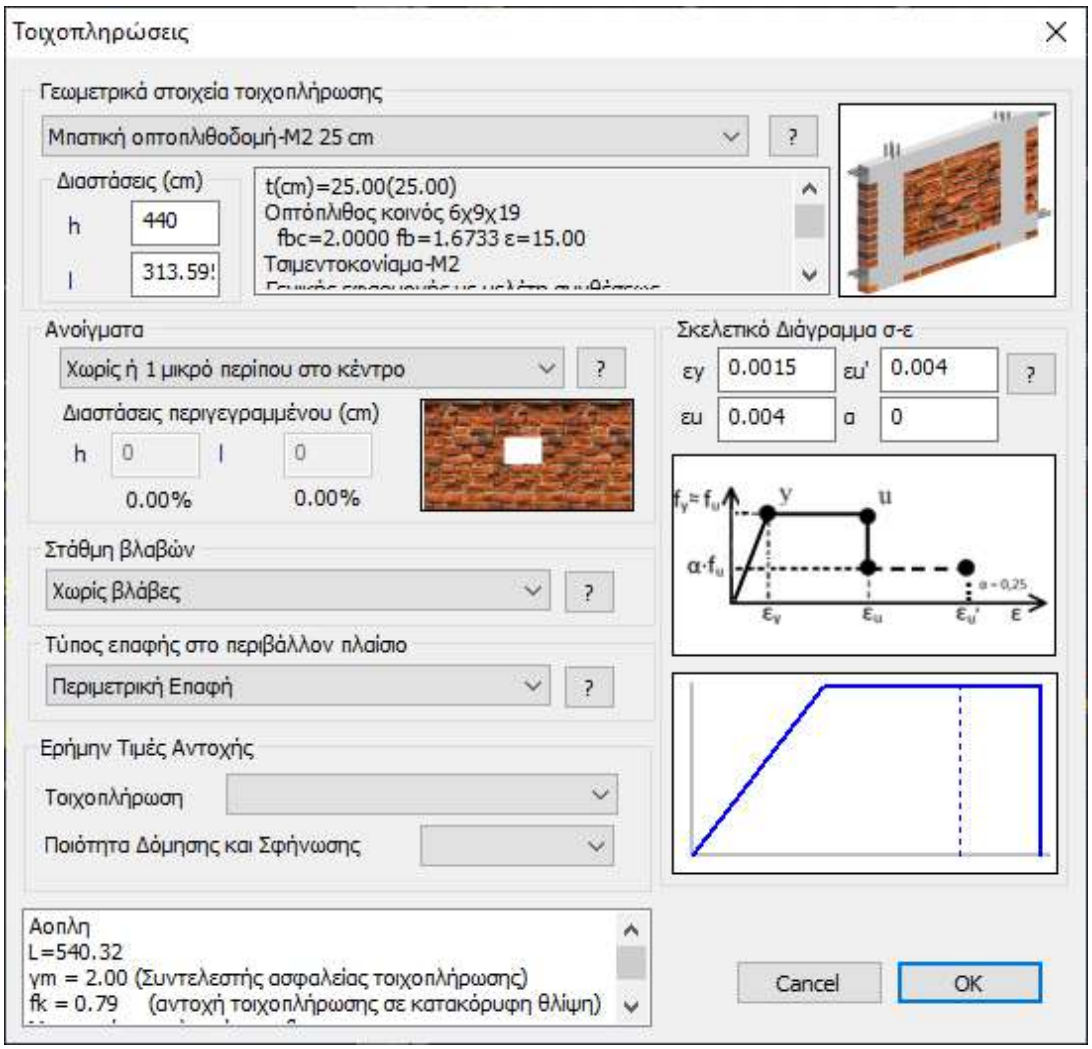
- **Members' freedoms**: Torque joints Mx,My,Mz,
- **Inert Elements:**
- *A: Diagonal area Ar*, which depending the type of analysis, is obtained:
- *Elastic Analysis*: Taken Art/2,
- *Pushover Analysis*: Each step of the analysis is done in 2 stages, in 1^{o} the pushover bars are identified, and in 2^{o} the analysis is repeated ignoring the first ones from the model,
- *E: Modulus of elasticity of wall infill*= 800÷1000 x fwc,s , fwc,s:

Compressive strength in the diagonal direction (S.18)

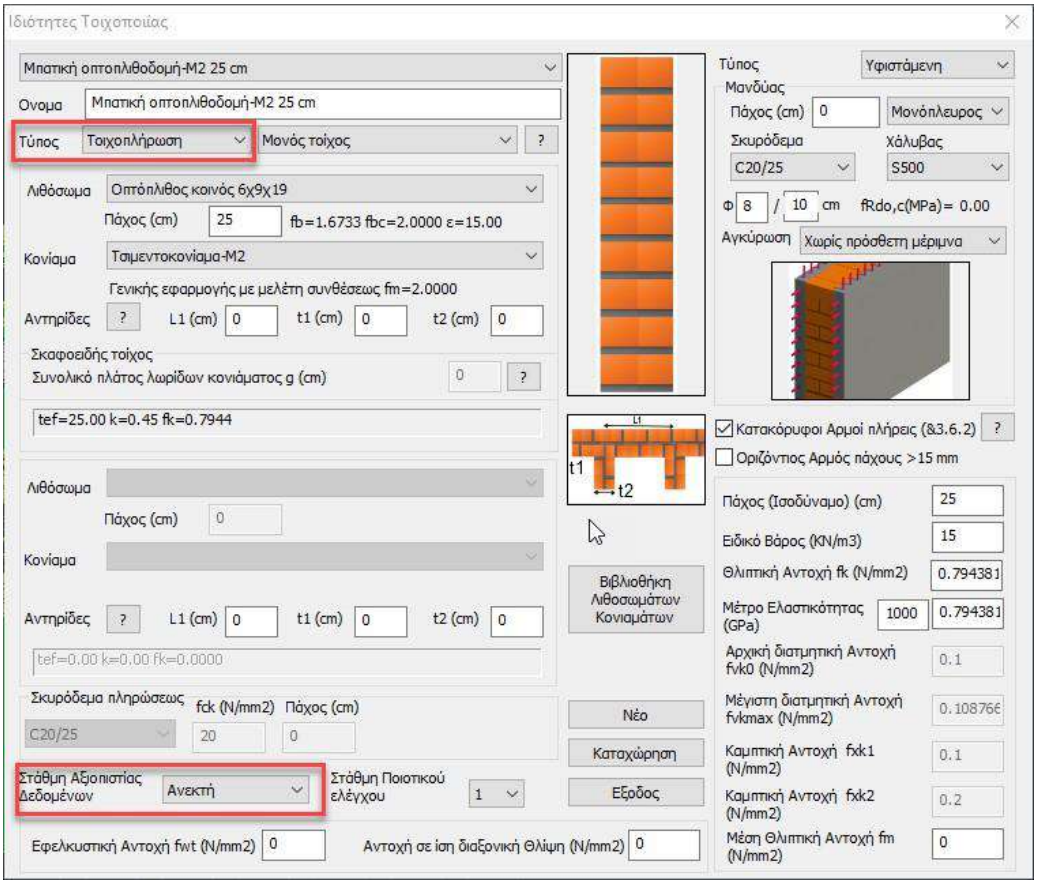
$$\overline{f}_{wc,s} = \lambda_m \lambda_s \lambda_c kf_{bc}^{0,7} f_{mc}^{0,3} \approx 1,25 kf_{bc}^{0,7} f_{mc}^{0,3}$$



After selecting the command, point the mouse at the mathematical member of the upper beam of the facade where the wall infill will be placed. The following dialog box will then appear

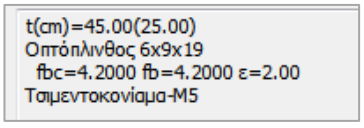


- In the section **Geometric elements of wall infill**, from the list you select the wall infill you have previously created in the masonry library. By pressing the symbol ? the masonry library dialog box appears, where you create and modify the wall fillings.



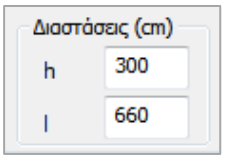
The icon on the right shows the corresponding type of wall infill. In the

next field:

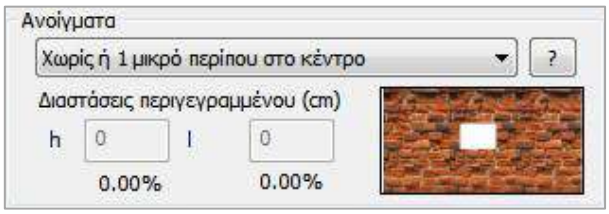


- The properties of the wall fill, such as the thickness t (total with the mantle and the net), the type of lithic body with its strength, as well as and the type of mortar with its corresponding strength.

- The next section automatically displays the height (h) and width (l) of the facet as calculated by the program with the possibility of processing.



- The next section concerns the definition of the openings of the wall infill. Select from the list one of the options.



OBSERVATION:
If you select "Other" you must specify the dimensions of the rectangle outlined in the openings. The choice of openings is made in order to calculate the reduction factor of the compressive strength n1.

- By pressing the ? button, an explanatory text with the corresponding paragraph of the C.E.P.E.

CAN.EPE. §7.4.1 d:

Openings: *Where infill masonry walls have* ***openings****, the respective statutory laws shall be appropriately modified to approximate the generally adverse influence of openings.*

- *Without or 1 small one in the centre: The wall infill is normally taken into account.*
- *2 large at the ends: The wall filling is neglected.*
- *1 large one in the centre: When there is an opening approximately in the centre of the phantom, the dimensions of which exceed 50%, the wall filling is neglected.*
- *Other: When there is an opening approximately in the centre or two small and adjacent openings that can be considered as an equivalent opening, circumscribed in them, where the dimensions h & l are between 20% and 50% of the facade, then the wall infill is taken into account.*

The next section concerns the wall infill damage from which you select the corresponding level to calculate the reduction factor rR.

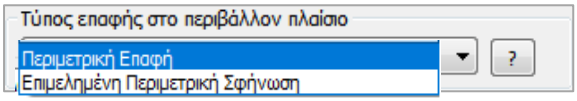
By pressing the ? button, an explanatory text with the corresponding paragraph of the C.E.P.E.

CAN LTD. Appendix 7D, §12:

Level of damage: *To define the extent of damage in existing masonry infrastructures, a classification into* ***damage levels*** *is adopted with reduction factors relating to the resistance of the compressive diagonal bar, rR , and the slope of the elastic branch of the skeletal diagram, rK , according to the table below:*

Στάθμη Βλαβών	Περιγραφή βλάβης	r_K	r_R
Ελαφρές	Ελαφρές (εως μέτριες) ρωγμές, < 2÷3 mm, γύρω από ανοίγματα, ή ρωγμές αποκόλλησης του ΦΟ και ΟΠ. Πολλαπλές ελαφρές ρωγμές, ιδίως σε τοίχους με ανοίγματα.	0,90 0,70	0,90 0,70
Σοβαρές	Έντονη ρηγμάτωση, διαγώνια ή δισδιαγώνια, με εύρος ρωγμής > 5mm, αποκόλληση από τον σκελετό, ρηγμάτωση των διαζωμάτων, απουσία σημαντικών μετακινήσεων εκτός επιπέδου (< 5mm).	0,50	0,50
Βαριές	Έντονη ρηγμάτωση, γενικώς δισδιαγώνια, με εύρος ρωγμής > 10mm, αποκόλληση από τον σκελετό, βλάβες των διαζωμάτων και μικρή μετακίνηση εκτός επιπέδου (μικρότερη των 15mm).	0,20	0,20

- The next option concerns the type of contact of the wall infill to the surrounding frame.

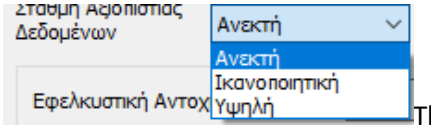


The choice of type affects the calculation of the reduction factor n3 due to looseness. Click on  to display the corresponding explanatory text.

CAN.EPE. §7.4.1 e:

Type of contact in the context environment:

It must be ensured that unreinforced wall fillings do not fail prematurely out of plane. The quantity that has a negative effect on this phenomenon is the looseness λ of the wall: λ= L / teff.

- 

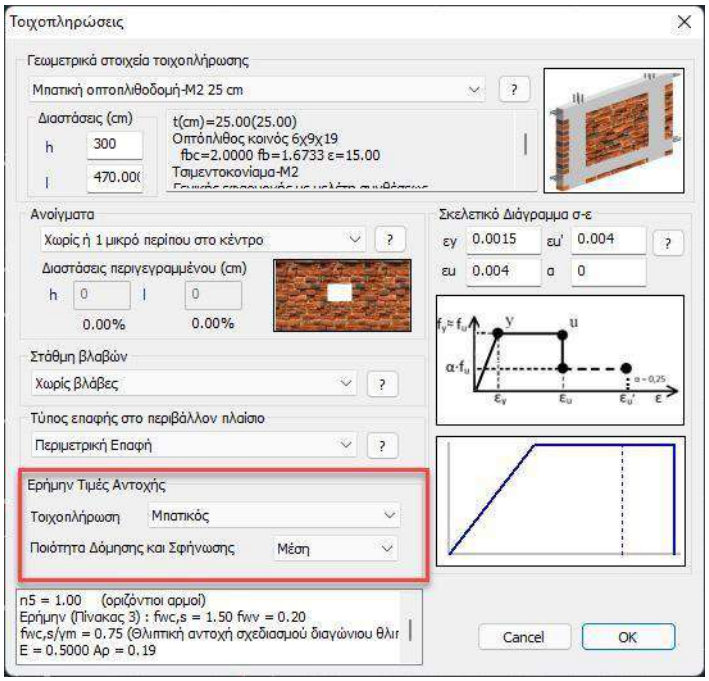
Three **levels of data reliability (DBR)** are introduced, depending on the **extent** and **accuracy of** the information.

- 

you determine the confidence level data according to the KAN.EPE. if it is an existing wall filling and the quality control level if it is new load-bearing masonry or added masonry infill.

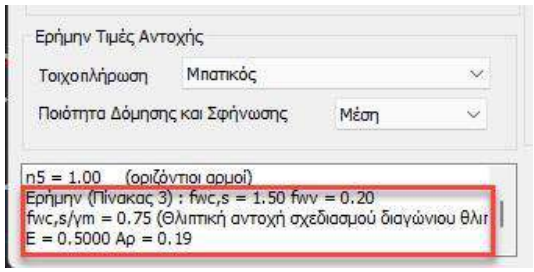
- In the new version of SCADA Pro has been integrated the table of Annex 3.1 of CAN.EPE. concerning "In Absentia" representative values for the strength of existing wall fillings.

More specifically, in the context of the dialogue of the introduction of wall fillings

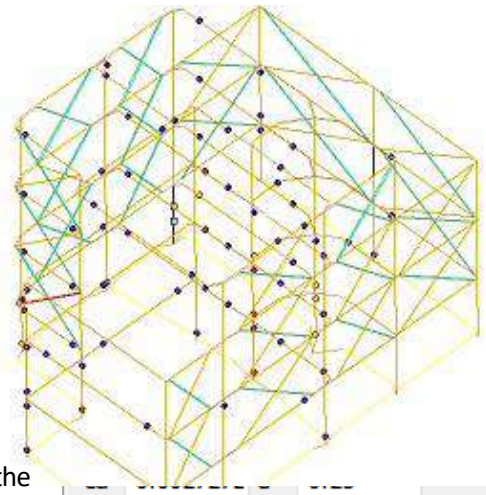


can set the designer the program to take into account "in absentia" strength values, choosing the type of wall filling and the quality of building and wedging.

The strength values to be taken into account are shown at the bottom of the box

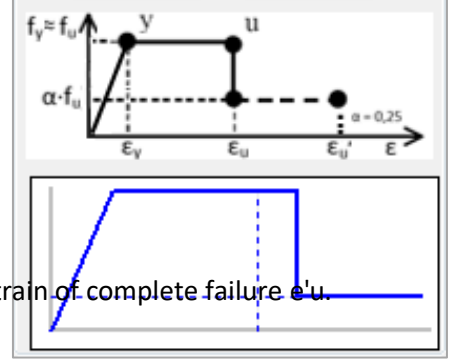


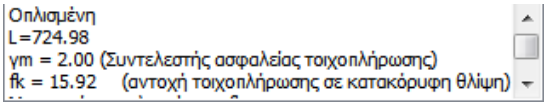
Application in SCADA Pro Retrofit:



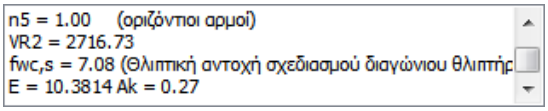
Show Pushover Results: each step of the analysis is done in 2 stages, in 1ο the pushover bars are identified, and in 2ο the analysis is repeated ignoring the first ones from the model. Those ignored by the model are shown in the deformation state of the beam in light blue.

- The next section concerns the creation of the skeletal diagram trends - deformations of the wall filling.
Here and in the corresponding fields you define, for the unreinforced masonry, the reduced yield stress εy and the reduced deformation quasi-uncertainty eu according to the CEEAM.
The skeletal diagram is automatically selected in the graph.
In the case of reinforced masonry, the values εy and εu are calculated automatically. In both cases the values can be optionally modified by the designer.
The coefficient **a** as shown in the graph is the percentage of residual strength after fracture and
applies only to reinforced masonry as well as the reduced strain of complete failure e'u.
Finally, the module



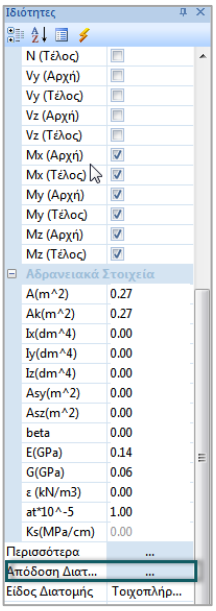


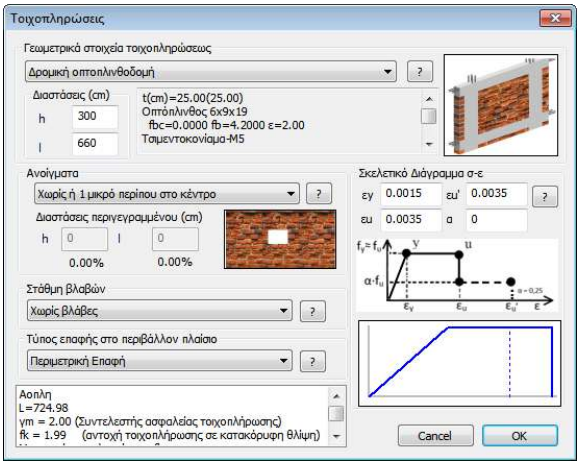
- contains, for the parameters mentioned above, the results of the selections, as well as the final design compressive strength of the diagonal crusher, the corresponding modulus of elasticity, the cross-sectional area Ak of the diagonal bars and their length L.



Clicking OK automatically creates the two diagonal bars in the facet. The wall fillers can be inserted either on the floor plan of each floor, or on the vector in 3D.

Subsequent reference, editing and modification of wall fillings is done through the properties module. By selecting a diagonal bar of the wall infill you select "Cross-section Performance" from the properties on the left and the dialog box





with the corresponding data of the wall infill you have already entered. Here you can change any element you want.

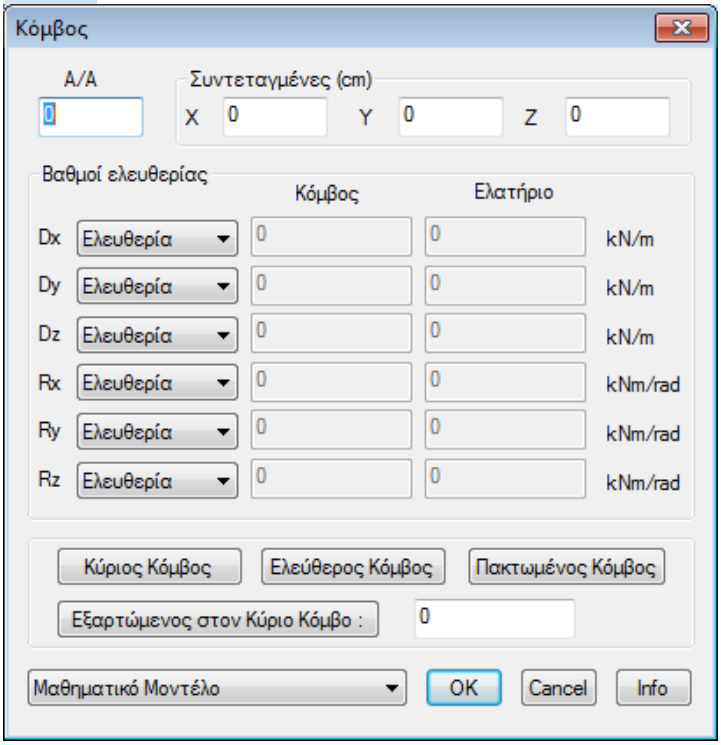
CAUTION:

It does not automatically update the masonry infill of the structure if you change any data in the masonry infill within the Masonry Library.

To update, you must reference each bar of the wall fillings using the procedure described previously, and click OK in the corresponding dialog box.

5.2 Hub

This command is used to insert a mathematical node (node of the mathematical model).
Selecting the command displays the following dialog box:



where you specify the node elements. More specifically, you define :

Enter the serial number "**A/N**" and the "**Coordinates**", or let the program fill them in automatically. In this case, select "OK" and use the mouse to point to the start and end nodes on the screen in either 2D or 3D.

- "**Degrees of Freedom**":
define the degrees of freedom that the node will have. There are four options regarding the status of the corresponding movement or turn of the node: "Freedom", "Packing", "Dependency", "Spring".

- "**Freedom**": allows the hub to move and turn freely in the corresponding direction
- "**Packing**": binds movements and turns
- "**Dependency**" means that the specific movement or turn of the node depends on the corresponding node, whose number you specify in the "Node" column, which is automatically activated when you select "Dependency".

Here you have the option to make movements and turns dependent on more than one node.
If you want the node you enter to be globally dependent on another node press the "Dependent onbutton 

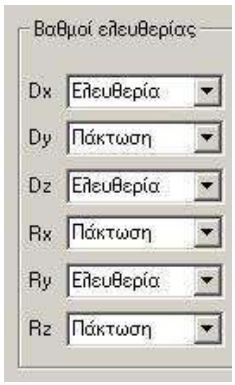
and enter the node number.

- "**Spring**": by selecting this option, the box in the "Spring" column is automatically activated to enter the spring constant.

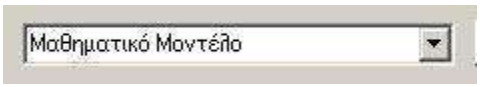
- "**Main Node**"

With the "Master Nodeoption 

the imported node gets the degrees of freedom of the master node. The degrees of freedom of the master node have the following format :



- "**List of default layers**"

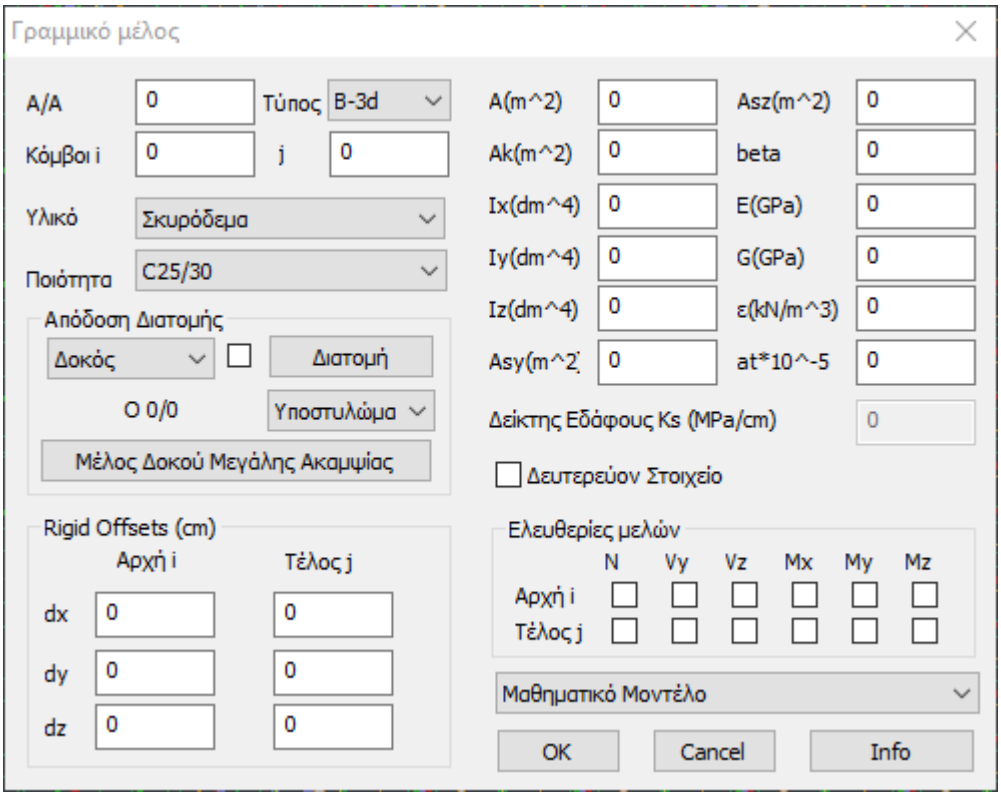


select the layer to which the element will belong that you will enter.

5.3 Member

5.3.1 Mathematical member

This command allows you to enter one or more mathematical members.
In addition to the possibility of directly entering the physical characteristics of the bar, the program provides the possibility of automatic calculation of these elements by entering the corresponding physical cross-section.
In SCADA Pro you can enter 3 types of linear elements: **B-3d**, **Truss** and **B-3def** (beam on elastic foundation).
Selecting the command displays the following dialog box:



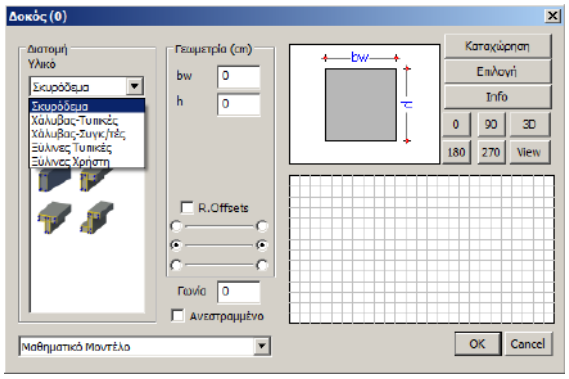
where you enter the details of the member.

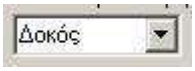
Specify for the member the "**Type**", "**Material**" and "**Quality**". Decide whether to assign the member a cross-section and which one.

Enter the serial number of the member "**A/A**" and the start nodes "**i**" and end nodes "**j**", the Rigid Offsets and the geometric and inertial characteristics, or let the program calculate them automatically. In this case select "OK" and point mouse to the start and end nodes on the screen in either 2D or 3D view.
In detail:

- "**Press**,
  B-3d : The type of rod used in most cases. There are
  intensive quantities from all kinds of works and forces (tension, compression, bending, shearing, etc.) depending on the degrees of freedom of the member.
  Truss : The type of rod which is stressed only by axial forces. *CAUTION not to be used when at least one of its two nodes belongs to a diaphragm.*
  B-3def (Beam 3d on elastic foundation) : The type of beam used in the foot beams. In this case there are no works from axial forces. The start and end nodes are buttressed in x and z displacements and for y bending.

- "**Cross-sectional Performance**",
  You can enter the physical cross-section you want so that the program can calculate the cross-section automatically its inert elements.





First select from the list whether the cross-section you enter will be a beam or column. Pressing the key "Cross-section."

appears respectively, the beam or column input mask where you enter the data, based on what was mentioned in the corresponding paragraph, where you select the material, cross-section and layer:

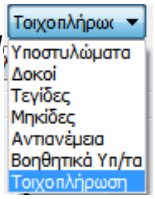
By selecting the "OK" button in the cross-section input mask, you return again to the bar input mask where you can now see in the corresponding fields the inertial elements of the cross-section as well as the type and dimensions of the cross-section.

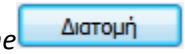
The checkbox next to the "**Cross section**button 

is checked which means that the rod you will enter has a "physical" representative which will be displayed and dimensioned normally. If you wish the rod you insert to participate only in the mathematical model as a take-up of intensive quantities you should uncheck this checkbox.

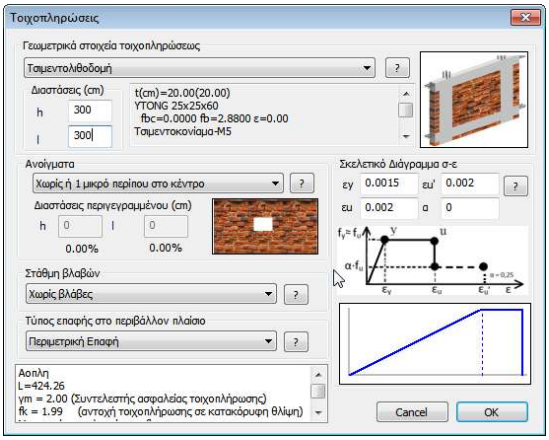
OBSERVATIONS:
Under the "Cross-section" command, there is a list that only applies to ***metallic cross-sections*** *and requires the selection of the corresponding group each time you assign a metallic cross-section to the member.*

*There is also a **wall fill** option if you wish to manually insert diagonal bars to simul the wall fill.*



If you choose your member to be a wallfill then the 

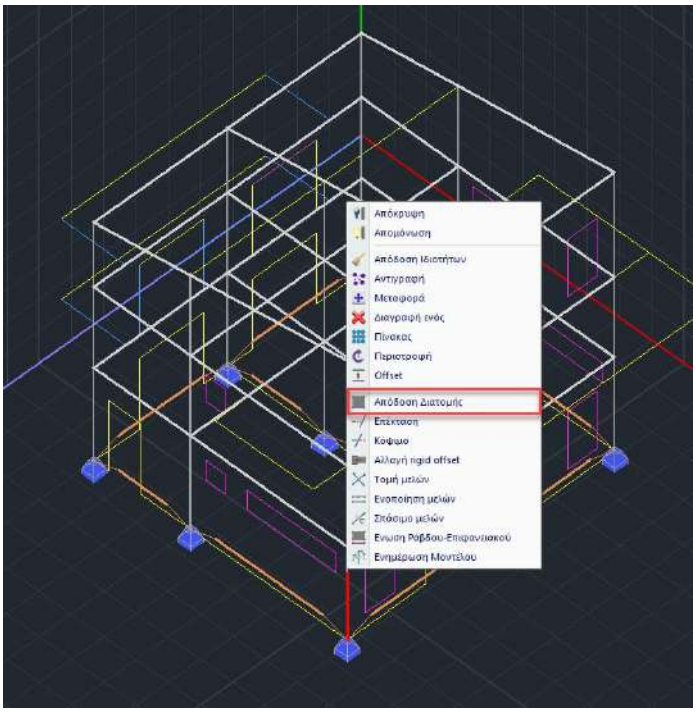
option opens the dialog box of the wallfill import.



The data input procedure is the same as the one mentioned earlier in the automatic procedure, except that you have to enter the length l and height h of the facet yourself. It is understandable that once the wall elements are given the placement of the diagonal bars must be done completely manually from node to node with the procedure followed when entering any mathematical member.

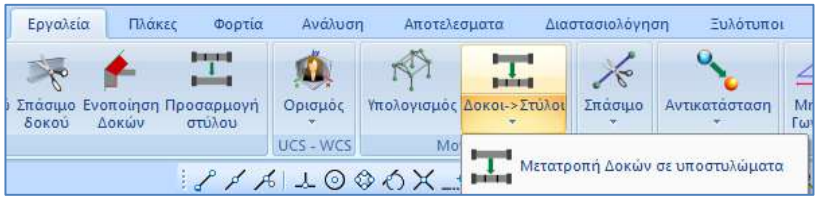
Select Cross-sectional Performance in the right button menu

In the new version of the program, the "Cross-section Performance" option has been added to the right-click menu to quickly and easily change the performance of a cross-section in a linear member.

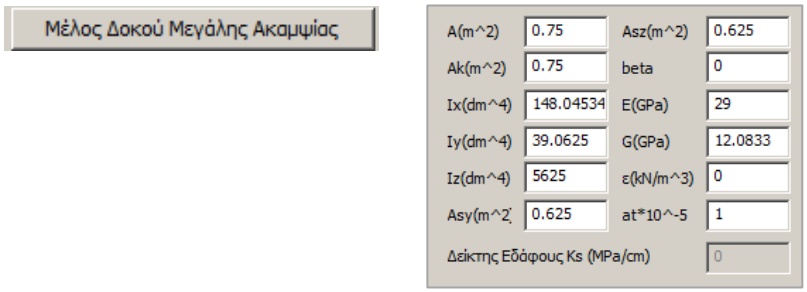


- "**Member of the Great Rigor Beam**"

It is a useful automatic data input tool, which is mainly used when ***simulating basement walls*** with the command "Convert Beams to Columns" and creating connected column cross-sections.



By selecting the button, the field of parameters is automatically filled in for a high stiffness cross-section with zero specific weight, without cross-sectional performance.



- "**Rigid Offsets**"

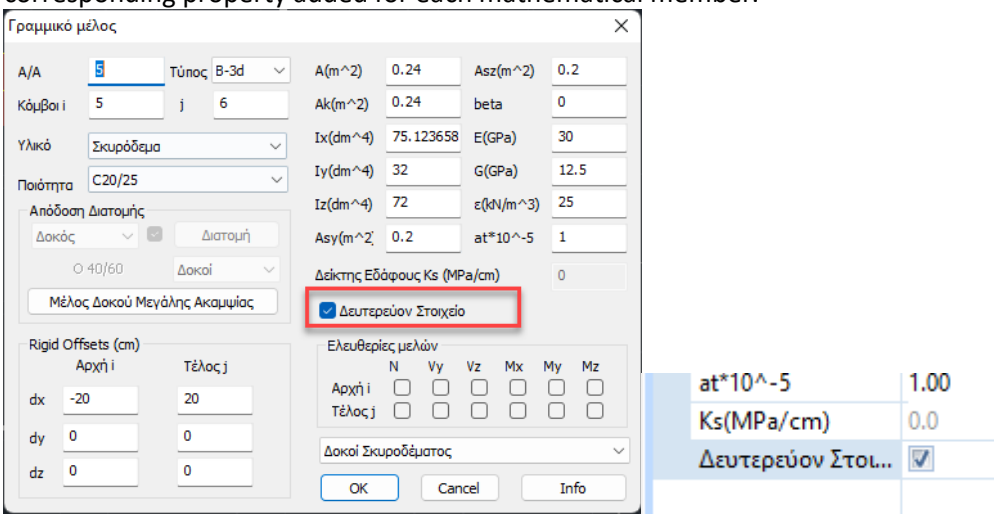
Enter in cm the rigid sections of the beginning and end of the rod respectively.
You can enter the "geometric and inertial parameters" of the linear member yourself or let the program calculate them automatically after you select "Cross-section".

- "**Geometric and inertial parameters**". A : the

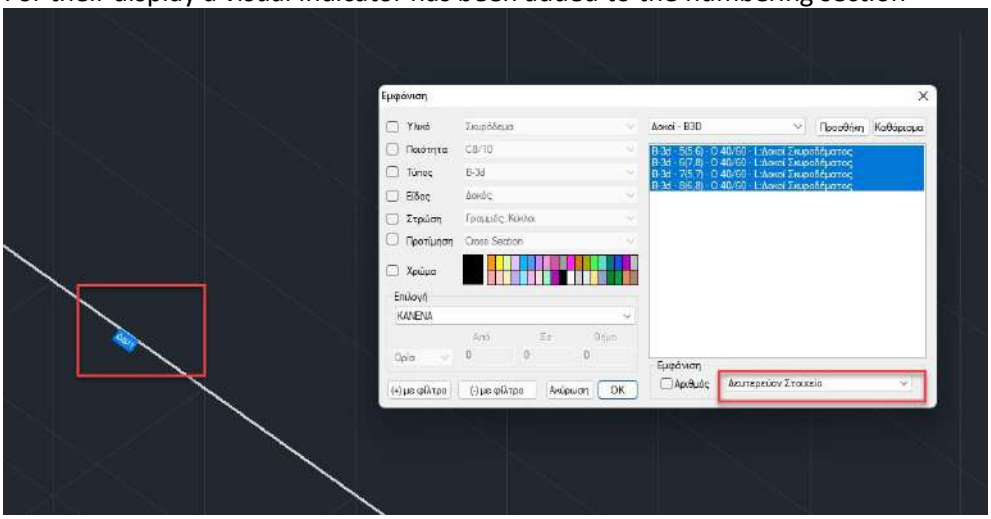
area of the cross-section, (in m2)
From : the area of the cross-sectional area of the trunk, e.g. in the case of a slab (in m2) Ix, Iy, Iz : secondary moments of inertia with respect to x,y,z axes respectively (in dm4)
Asy, Asz : the shear areas of the cross-section along the y and z axes, respectively (in m2)
beta : the beta angle (in degrees)
E, G : the modulus of elasticity and shear modulus of the material to be used, (in GPa) e : specific gravity of the material (in KN/m3)
at : the thermal coefficient
Ks : Soil index (in MPA/cm). The field is activated when you select B-3def as membership type.

- "**Secondary Element**"

The checkbox next to the "**Secondary Element**" button is activated by the designer to identify as secondary any horizontal and vertical elements that, in his/her judgement, do not participate in the absorption of horizontal seismic forces. The designation is made by activating the corresponding property added for each mathematical member.



For their display a visual indicator has been added to the numbering section



The selection is made only in mathematical members of reinforced concrete

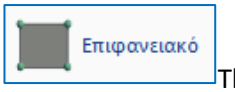
- "**Members' Freedoms**"

The default state is all fields unchecked which means that all fields are present the intensive sizes. Omission of a specific intensive size is done by checking the corresponding checkbox of the beginning or end of the bar.

- "**List of predefined layers**

you can select the layer that will be owned by the rod you will insert. The automatic default is the "Mathematical Model" layer.

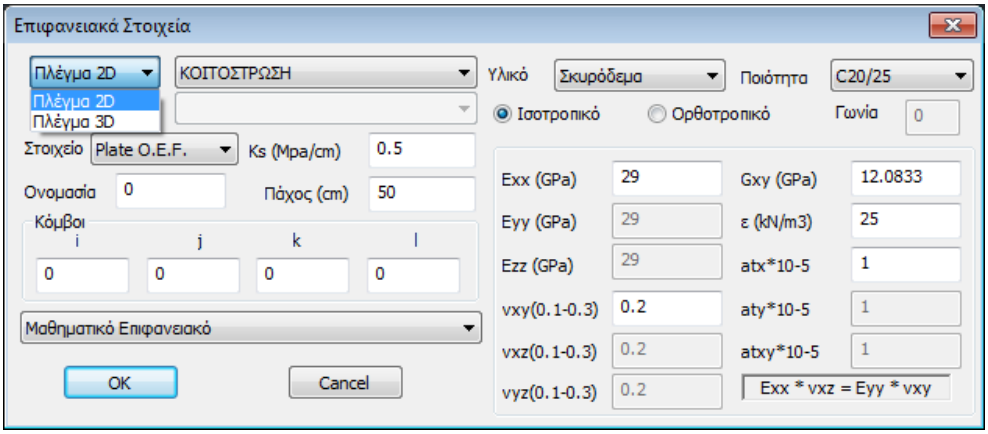
5.3.2 Surface



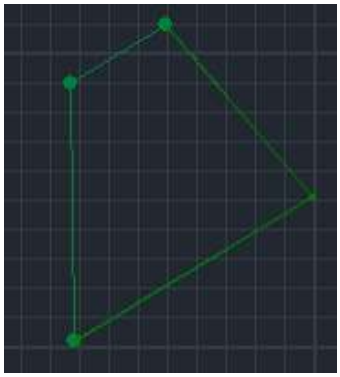
This is the command that allows the manual creation of individual 2D and 3D surface elements.

OBSERVATION:
This command requires that you have already specified, via the "Grid" command in the "Superficially", the characteristics of the grid, while at the same time, it allows some individual modifications, both in the Element, Width, Material, Quality, as well as renaming it.

Selecting the command opens the dialog box:



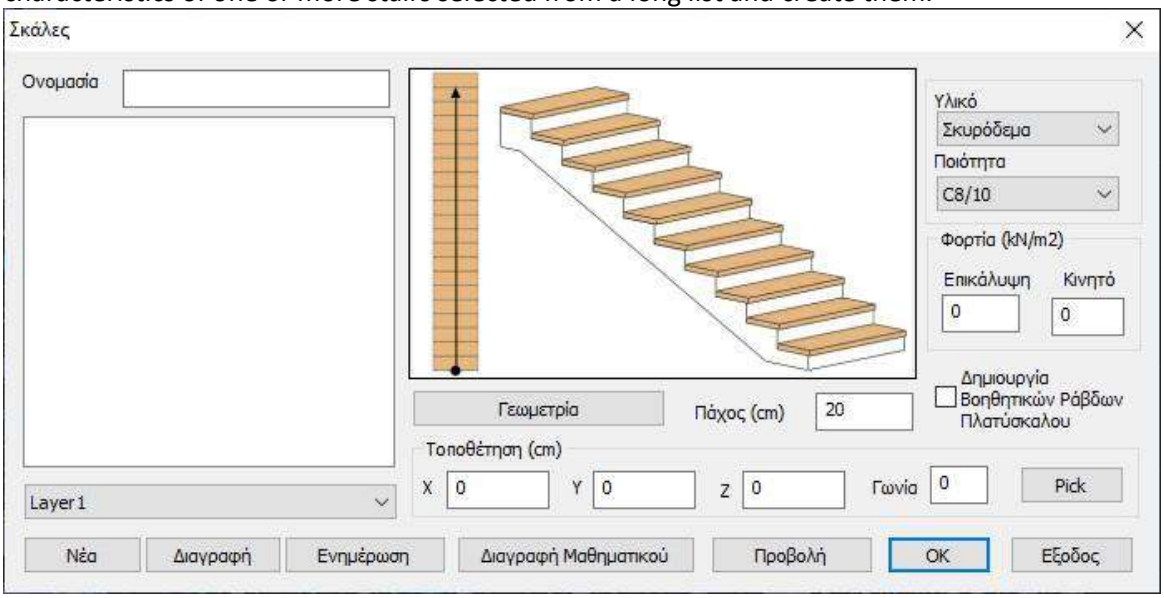
where you first select the grid type and the default grid.
The parameter field is automatically filled in by selecting the grid, and can be manually modified.



In the "**Nodes**" field you can either type the number of nodes of the individual surface, or leave it blank, and on the desktop select the four nodes that make up the surface by left-clicking on them.

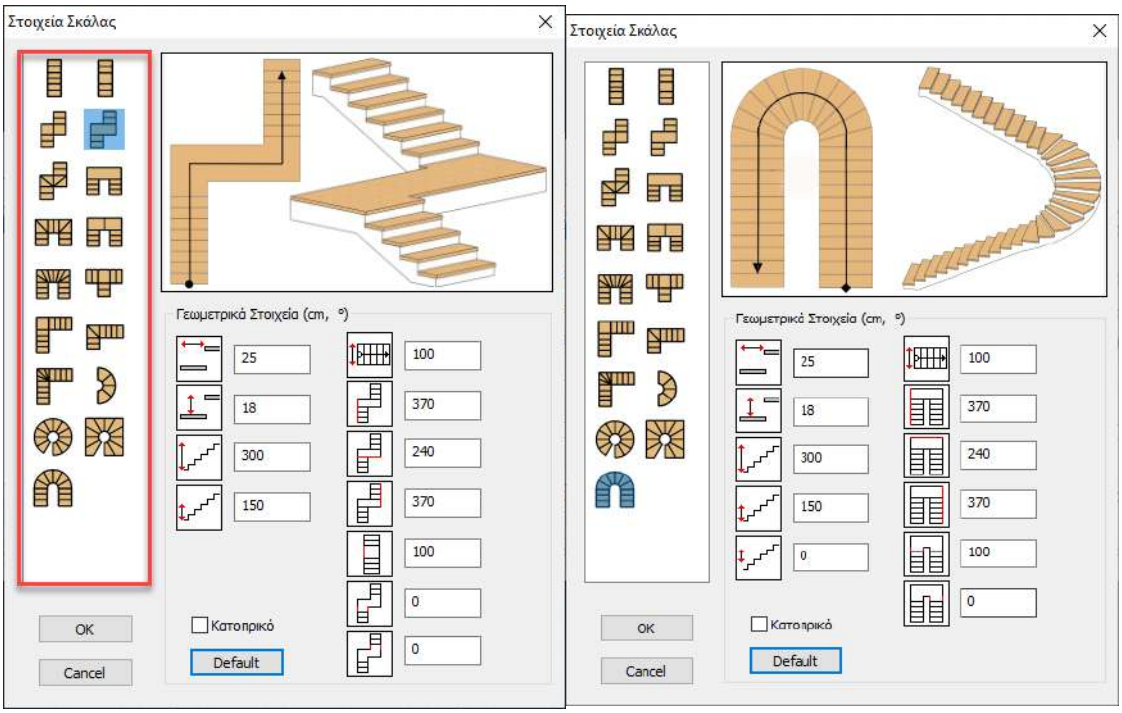
5.4 Stairs

This is the new command that allows modeling of stairs.
Selecting the "Stairs" command opens the window of the same name in which you define the characteristics of one or more stairs selected from a long list and create them.



The procedure for creating the ladder is as follows:

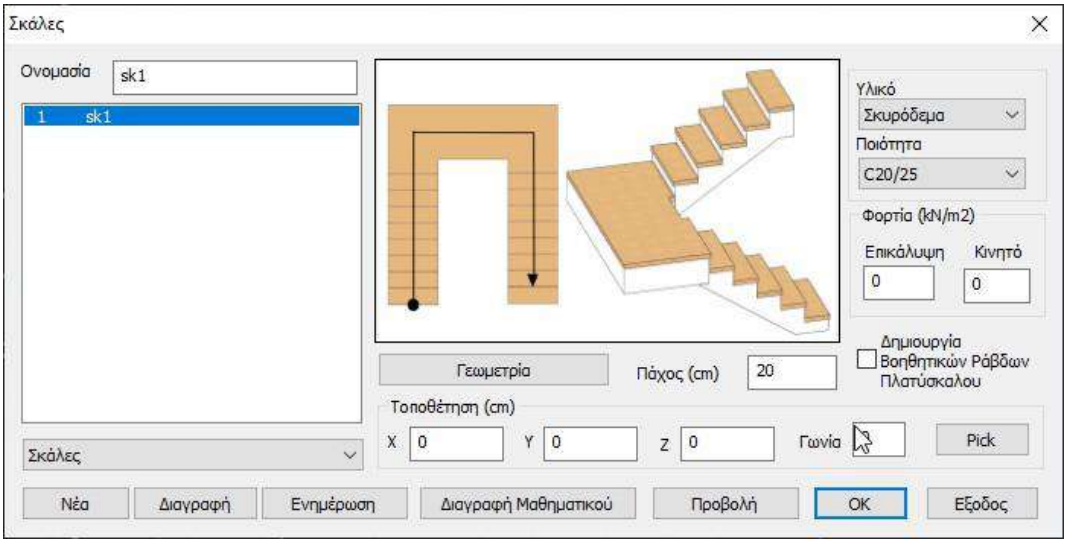
1. In the **Name** field, enter a name for the ladder you want to create and then select **New**
2. Select the **Geometry** command and a window opens with a list stairs to choose the one that suits your study. The staircase is selected by left-clicking within the list and at the same time the drawing and geometric data of the selected staircase is displayed in the right part of the window:

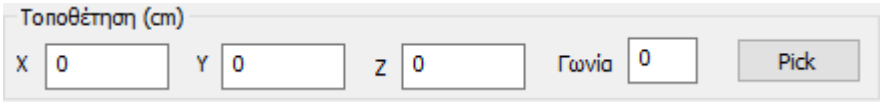


The icons on the side with a red indicator show the dimension that the user is asked to declare while there is also the option Default that automatically fills in the dimensions for a predefined ladder that can be modified at will.

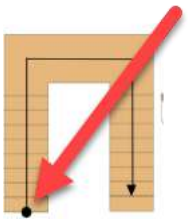
The black arrow on the drawing of each ladder indicates its direction and position of the Mathematical Model of the ladder after its calculation.

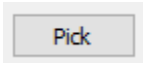
In cases where the ladder is mirrored with respect to the one shown, select the command Κατοπρικό. After you have finished with the geometric elements of the selected ladder, select OK to return to the first window:



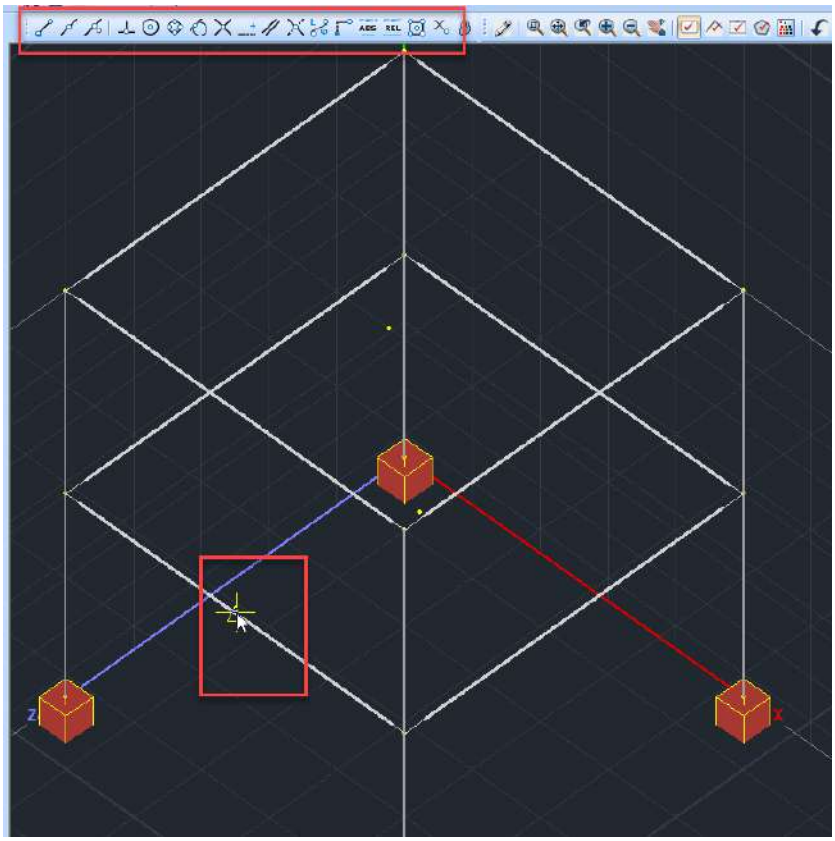


The Position field allows you to specify the insertion point of the ladder (the black dot of the drawing):

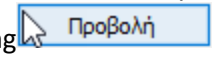


You can either write the coordinates of the point inside your model, or you can choose 

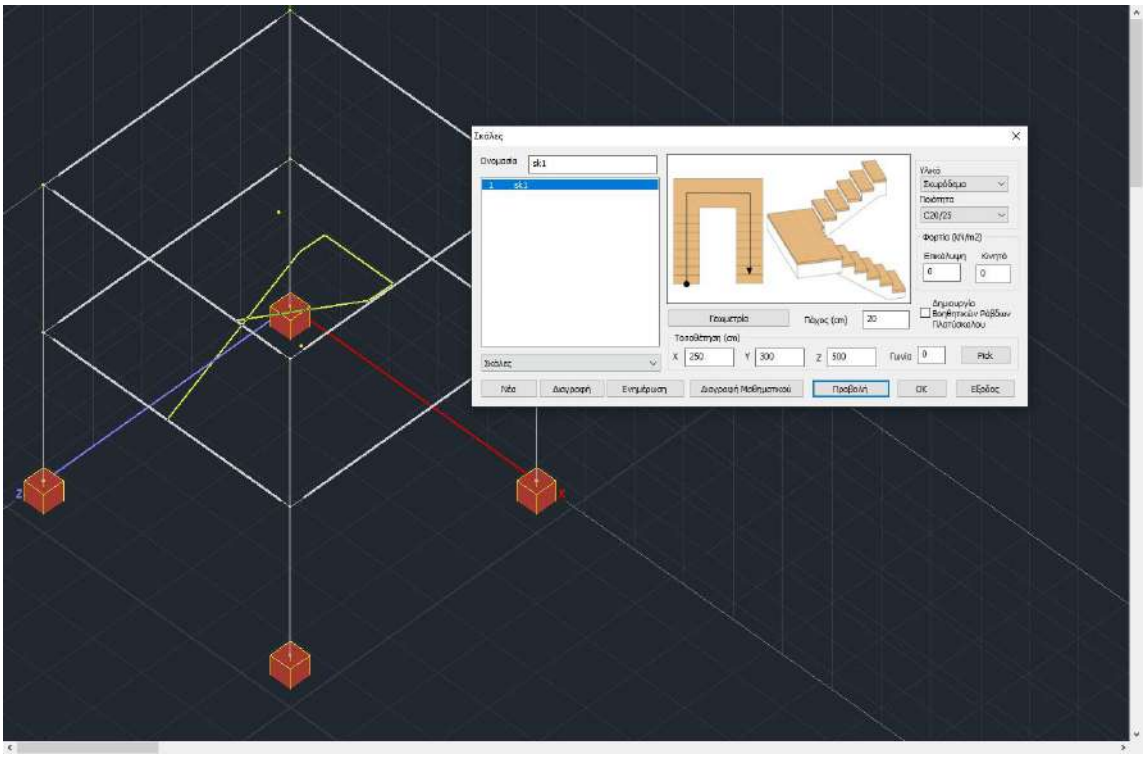
and show it using attractions:



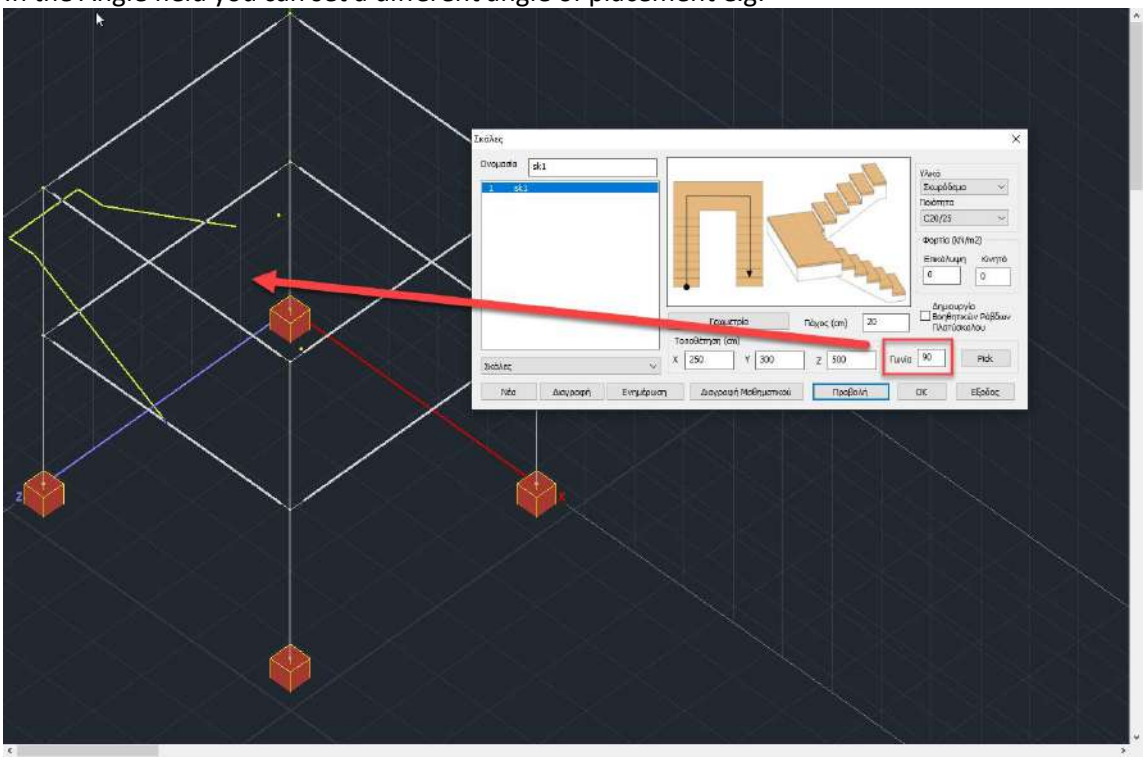
The window automatically reopens and the coordinates of the ladder insertion point are filled in.

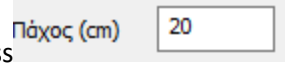
By selecting 

you can see the ascent line of the ladder you created:



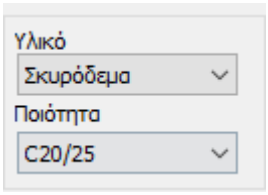
In the Angle field you can set a different angle of placement e.g:



Finally, set the thickness 

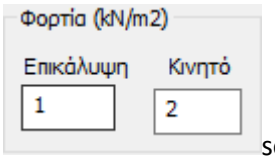
of the "mat" of the ladder.

In the field:

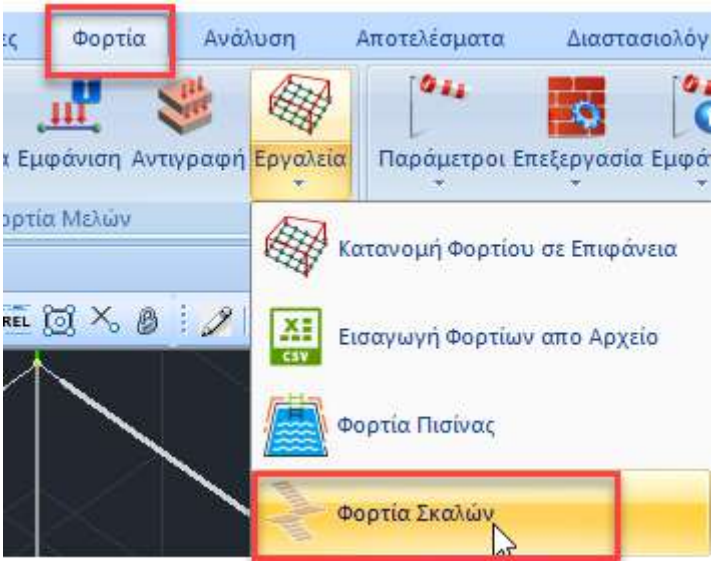


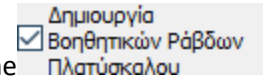
you specify the material and quality for the ladder.

In the Loads field:



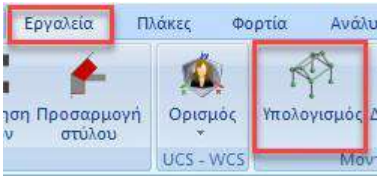
set the load in (kN/m2) for the ladder overlap, as well as the Mobile load to be taken into account then in the Loads:



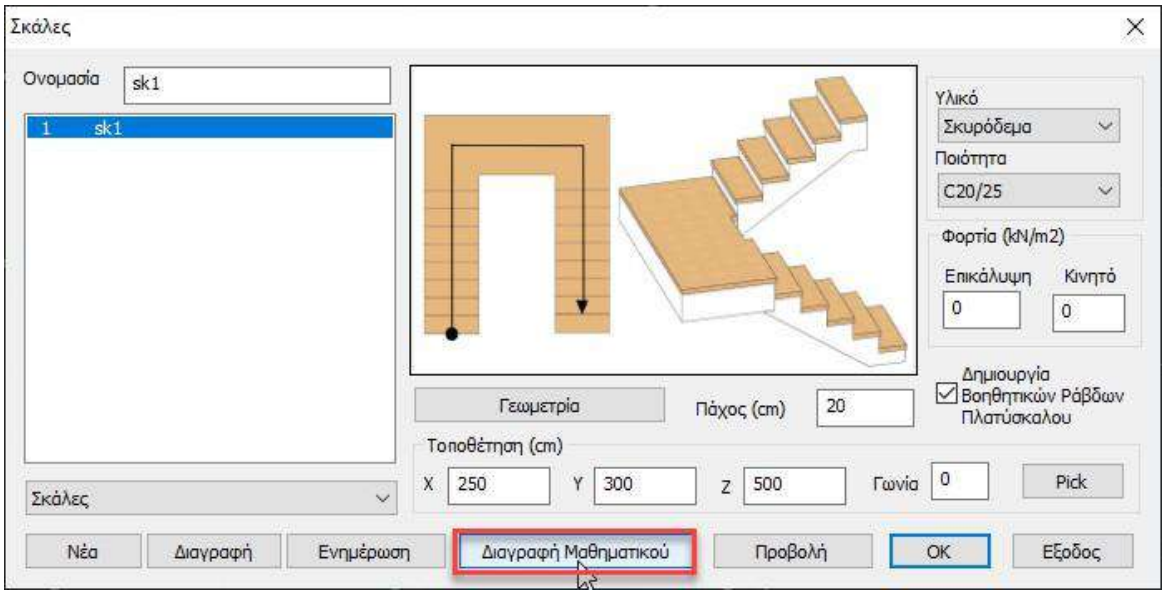
With the 

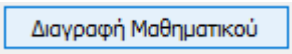
option, additional bars are created that serve to better connect the mathematical simulation of the ladder with the rest of the structure.

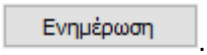
After you have completed all the data for your ladder, through the TAB Tools you can calculate the Mathematical Model



In case you wish to make a modification to the ladder you have created, go back to Modelling - Ladders :



Select the 

make the modifications you wish; and select 

.

select

6. Additional information

The "**Extras**" command group contains additional commands particularly useful in the field of modelling:

- **Cross-section identification**
- **Typical Constructions**
- **Model checks**
- **Information**

6.1 Cross-section identification

Importing floor plans into the SCADA Pro environment offers multiple possibilities. The user can enter the corresponding floor plan at each level and take advantage of the drawing's pull points to enter the data.
Select "File">"Import" and open the DXF/DWG file of your study.

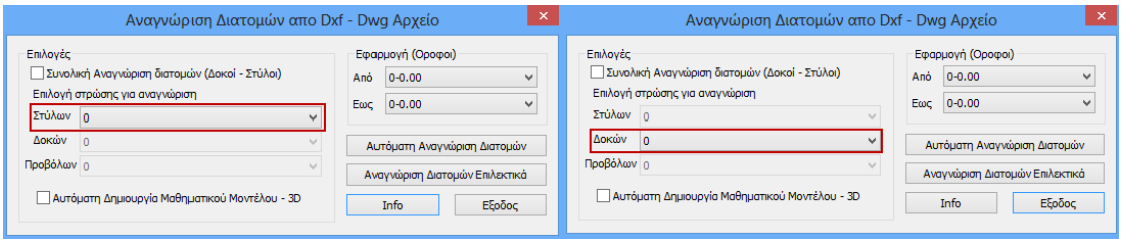
With Cross-sectional Recognition, SCADA Pro offers an additional unique feature that greatly simplifies and accelerates the modeling of your study.

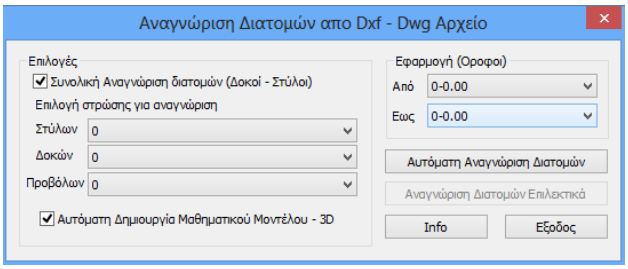
This is the automatic creation of data from DXF/DWG. OBSERVATION:
Conditions for using the command:

- Have the levels and levels created
- Have the floor plans (.dwg/.dwf files) imported at the corresponding levels

The **Cross-section Identification** command includes the list on the left.

Either of the three options opens the same dialog box, the only difference being that, in the "Columns" option, active are the Columns, while in the "Beams" option, active are the Beams.

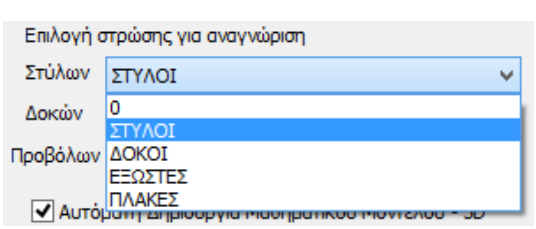


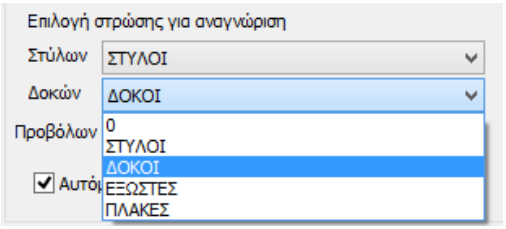


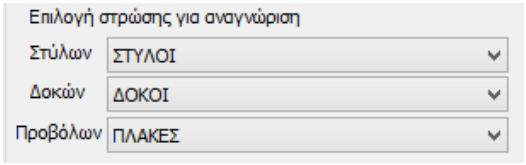
In addition: by activating the "Total Recognition", the Pillars and Beams and Propellers are activated for simultaneous recognition.
The list with the arrow next to "Select layer for recognition" - Columns, Beams and Slabs, includes all Layers of the .dwg auxiliary file.

OBSERVATION:
<u>The basic requirement for the correct operation of the identification automation is that both the columns, beams and slabs belong to a single separate layer of their own</u>.

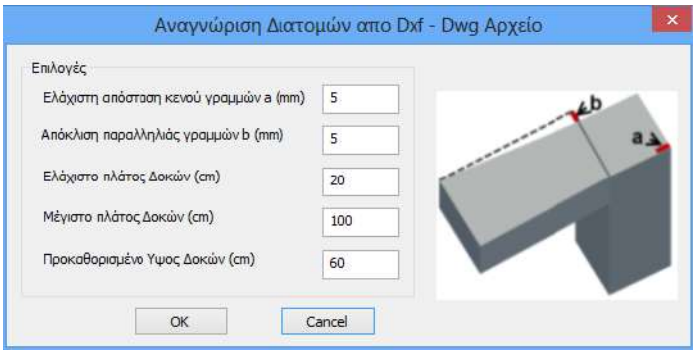
EXAMPLE:







Info : press the "Info" button to define some additional geometry elements that allow the correction of possible design deviations:

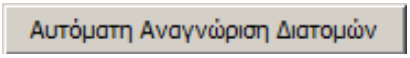


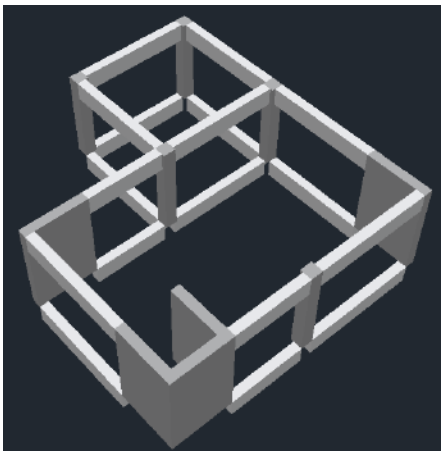
The first 2 correct possible design errors (gaps, parallelism deviation) (see drawing on the right)

The last 3 determine which parallels will define beams and how high they will be.

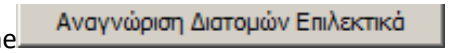
By activating the automatic creation of the mathematical model, the program not only recognizes and enters the physical cross-sections (physical model), but also calculates the inertial elements and creates the mathematical model directly.

OBSERVATION:
The basic requirement for the automatic recognition of plates and projections is that both the columns and beams have been selected for creation, and that the automatic creation of Mathe is activated. Model, so that the members that will surround the slabs exist.

Selecting the: 



you get the photorealistic representation of the model directly.

Selecting the 

- Cross-section identification Selectively> **Columns**

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

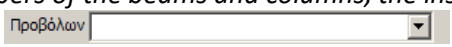
- Cross-section identification Selectively> **Beams - Foundation beams**

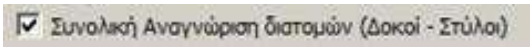
select, as before, the beams (anode or foundation respectively) one by one.

OBSERVATIONS:
To display the beam dialog box, to set the height of the beam, move the mouse to the inside of the beam outline and press SHIFT on the keyboard. Enter the geometric data and continue clicking on the beam.

The automatic insertion of the beams with Cross-section Recognition creates beams with rectangular cross-section and 60cm hanging. You can intervene from the start using SHIFT or after they have been placed via the Properties that open on the right of the screen each time you select an element.

In the DXF/DWG design file, make sure that the contours of columns and beams are closed and defined with a polyline, starting and ending with a vertex of the column or beam, or with individual lines, one for each side.

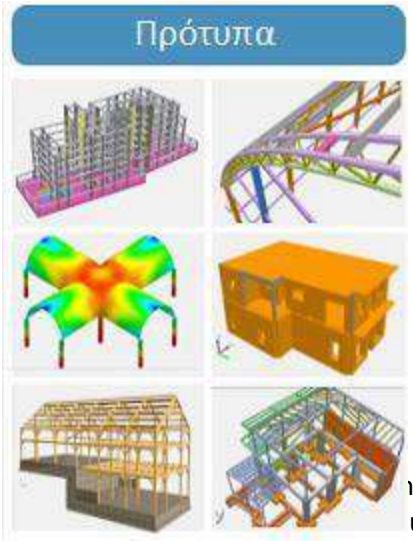
Without the members of the beams and columns, the insertion of the plates cannot work, so the field 

is only activated by activating of 

6.2 Typical constructions

The introduction to the standard constructions tool can be done in 2 ways:

1st way:
Left-click on one of the icons on the home screen to select the type of standard construction



concrete metal

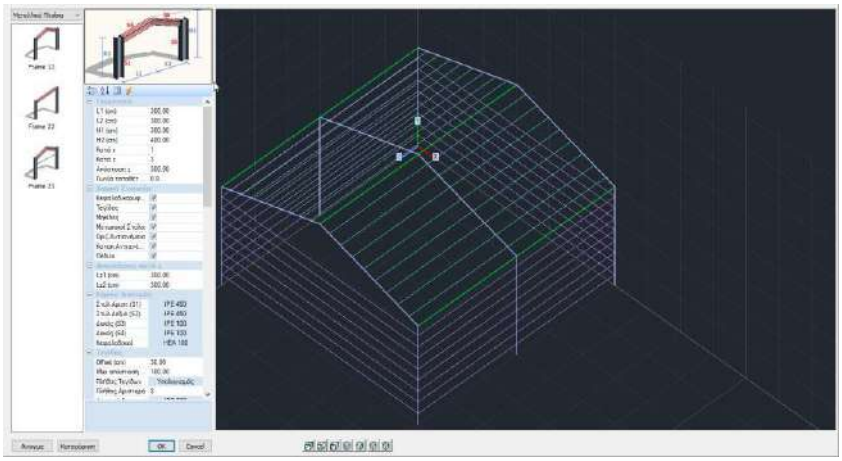
surface masonry

wooden
mixed construction

the dialog box that opens and the standard
utomatically opens.

2nd way:

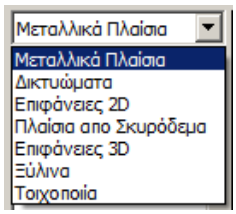
- select the command ADDITIONS > TYPICAL CONSTRUCTIONS and
- automatically opens the standard constructions dialog box.



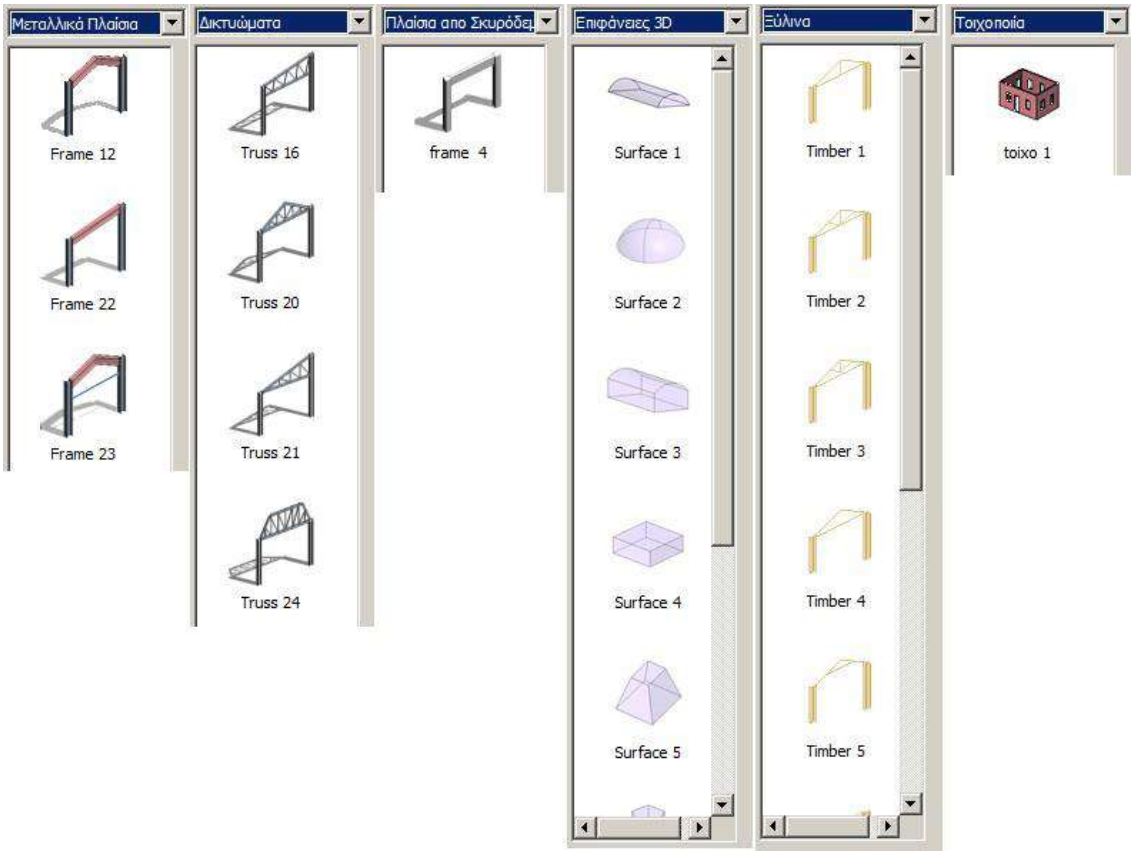
OBSERVATION:

If there are already elements in that file, then you also select the insertion point on the desktop to open the standard constructions window.

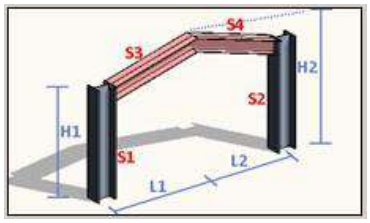
Select from the list of.



the type of construction and the corresponding



6.2.1 Metal frames

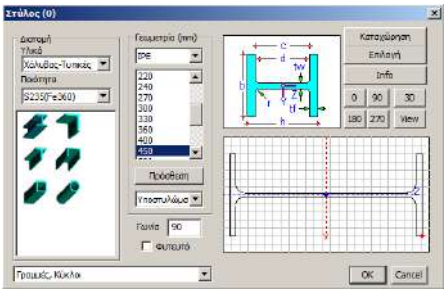


Select the "Geometry" based on the drawing and the x and z iterations.

Γεωμετρία	
L1 (cm)	300,00
L2 (cm)	300,00
H1 (cm)	300,00
H2 (cm)	400,00
Κατά x	1
Κατά z	3
Απόσταση z	300,00
Γωνία τοποθέτησι	0,00

The structural elements that will take part in the construction must have the corresponding checkbox.
For each structural element, select the corresponding cross-section.

Κύριες Διατομές	
Στύλ.Αριστ.(S1)	IPE 450 (0.0)
Στύλ.Δεξιά (S2)	IPE 450 (0.0)
Δοκός (S3)	IPE 330 (0.0)
Δοκός (S4)	IPE 330 (0.0)
Κεφαλοδοκοί	HEA 180 (0.0)



Click on the default cross-section and in the dialog box select your own cross-section.

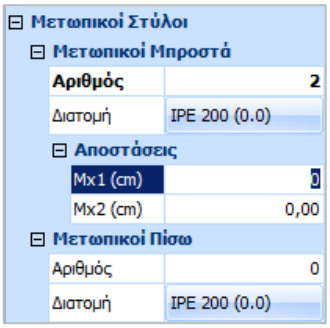
OBSERVATION:
Whenever you modify a default cross-section, in the dialog box specify the layer to which it will belong. It is important to have the correct correspondence of the elements in the layers so that you can take advantage of the program commands that act globally (for each layer) and save a lot of time.

Τεγίδες	
Offset (cm)	30,00
Max απόσταση (α	100,00
Πλήθος Τεγίδων	Υπολογισμός
Πλήθος Αριστερά	8
Διατομή Αριστερά	IPE 100 (0.0)
Πλήθος Δεξιά	8
Διατομή Δεξιά	IPE 100 (0.0)

In order to calculate the number of pegs, enter in cm: "Offset": the distance of the first peg from the headstock, "Max distance": the maximum distance between the pegs
and select "Calculate".
The program automatically calculates the number of stripes per stripe. Alternatively, enter the two numbers (left and right) directly.

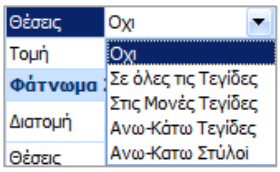
Offset (cm)	30,00
Max απόσταση (α	100,00
Πλήθος Μηκίδων	Υπολογισμός
Υπάρχουν Αριστε	☑ Ναι
Υπάρχουν Δεξιά	☑ Ναι
Υπάρχουν Μπροσ	☑ Ναι
Υπάρχουν Πίσω	☑ Ναι
Αριθμός	8
Διατομή Αριστερά	IPE 100 (0.0)
Διατομή Δεξιά	IPE 100 (0.0)
Διατομή Μπροστά	IPE 100 (0.0)
Διατομή Πίσω	IPE 100 (0.0)

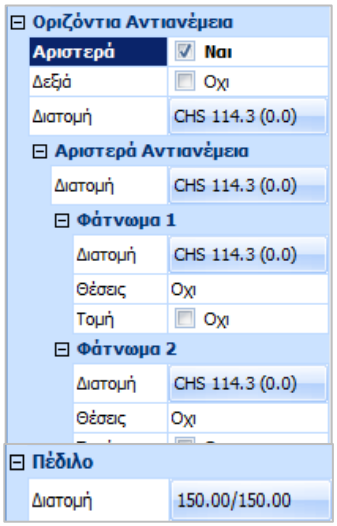
Calculate the number of μηκίδων (see previous) and uncheck the checkboxes of μηκίδων in the directions that do not exist.
It is possible to install thimbles of different cross-sections per direction. Just select the default cross-section and in the dialogue box set the new one.



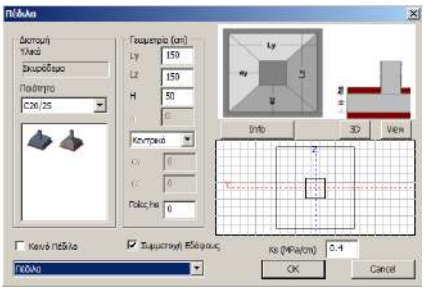
Enter the number of front posts (front and back) and select the cross-section.
For a number greater than 0, the "Distances" field opens where you set the distances between them in cm.

Activating the horizontal wind deflectors for the starboard and/or left, the box grows and for each facet, in addition to specifying the cross-section, the position and whether and how they intersect is requested
Similarly for the vertical headwinds.

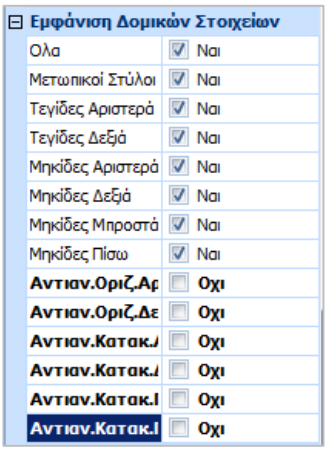


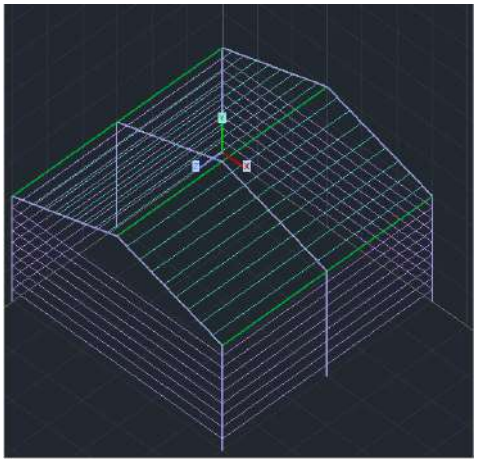


For the skis, click on the default cross-section to set the geometry, the terrain participation and the corresponding layer.

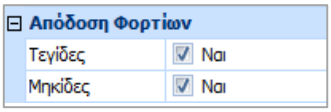


Choose which of the items to display on the dynamic screen and which not.





The dynamic screen shows the construction as it is being created. The bar at bottom allows you to select a view. Alternatively, by pressing the left mouse button and moving it, the figure rotates.



The "Load Performance" field refers to the wind and snow loads according to Eurocode 1 (Loads>>Wind-Snow Loads). When the tethers and the snow tethers are active, the program automatically allocates the wind and snow loads to them.

*You will find the "**Wind-Snow Loads**" automation in detail in the relevant chapter.*

"Save" to save the construction you set. You can create a folder where you can save the constructions you create through the standard constructions and build your own library of constructions that you can call upon in a future study.

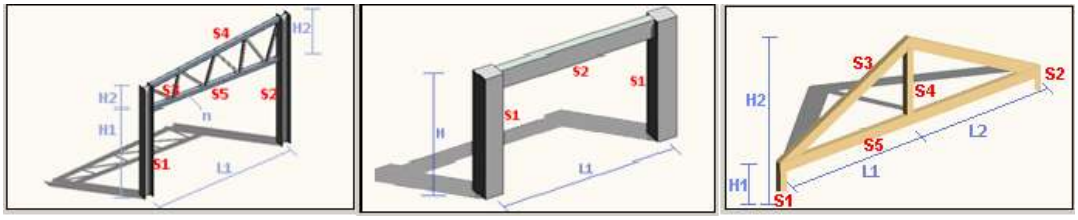
⚠ *It is now possible to preview the files you create and save in standard constructs.*

"OK" and the 3D photorealistic model of your study will appear in the SCADA Pro interface. Turn off the photorealism to display the physical and/or mathematical model and using the program's tools, make any modifications you wish.

⚠ *In the same study you can use more than one standard construction, of the same or different material, to create the most complex studies. Select the insertion point, set the "standard construction", save and "OK". Repeat the same procedure to insert a second standard construction on top of the first one.*

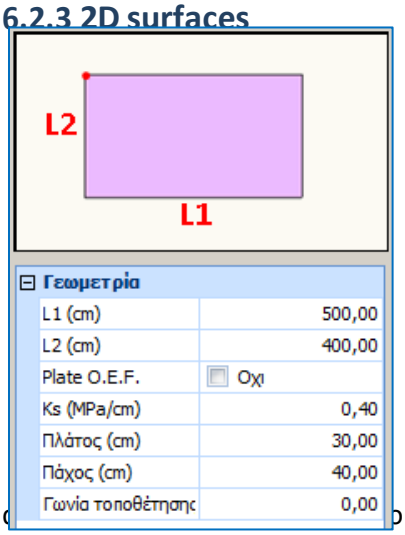
6.2.2 Mesh, concrete frames, wooden frames

Similarly you work by selecting any other type of linear construction from the list.



Select the base geometry of the design, member cross-sections, repeats and the frame is imported into the interface, referencing the default insertion point.

6.2.3 2D surfaces



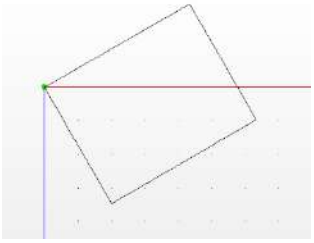
Select one of the proposed 2D surfaces and enter its geometric characteristics based on the design.

In case of paving, activate the checkbox

and enter the value of the spring constant Ks (MPa/cm).

The values "Width" and "Thickness" refer to the dimensions of the surface that will simulate the specific surface
(Note: the density is taken by default equal to 0.15)*

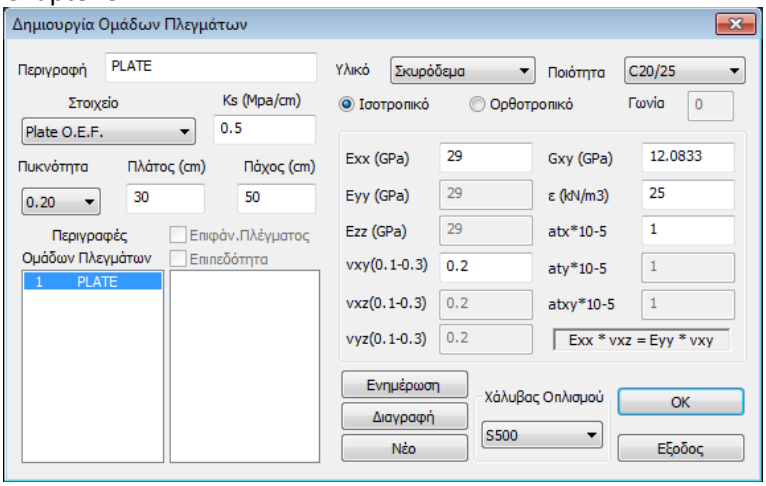
'Mounting angle' is the angle (in degrees) to be inserted in relation to total axes, in the XZ plane of the working surface.



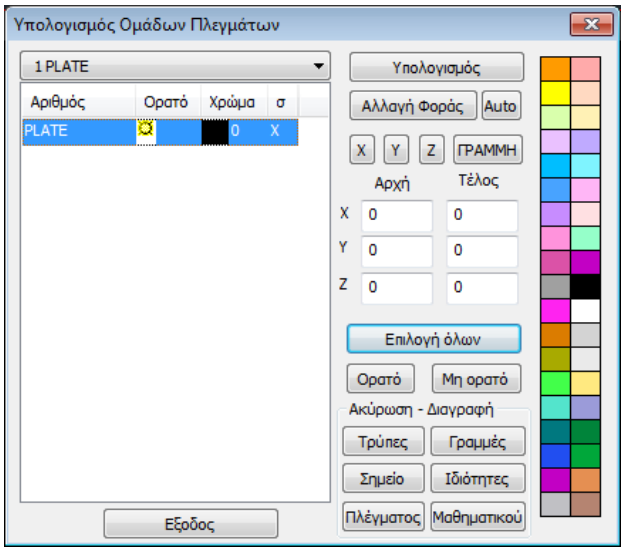
e.g. for an angle of 30° the surface will be tilted as in the figure:

"OK" and the SCADA Pro interface will display the outline of the surface as you have defined it.

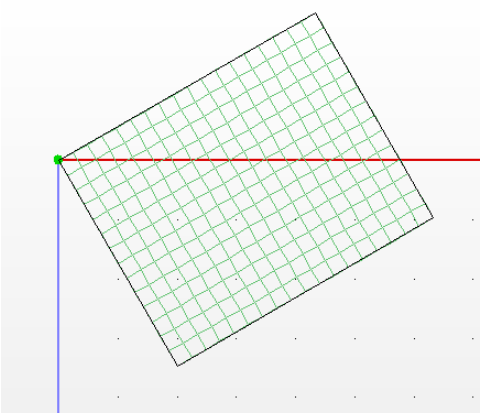
Within the "Modeling" field select the command "Surface 2D">>"Grid". In the dialog box within the list of groups, the mesh as defined in the standard constructions is displayed. Select it in case you want to make changes (e.g. to the density*) and "Update". The command is described in detail in Chapter 3.



If no modifications are needed, go directly to the command "Modeling>> "Surface 2D">>"Calculation", which opens the dialog box:

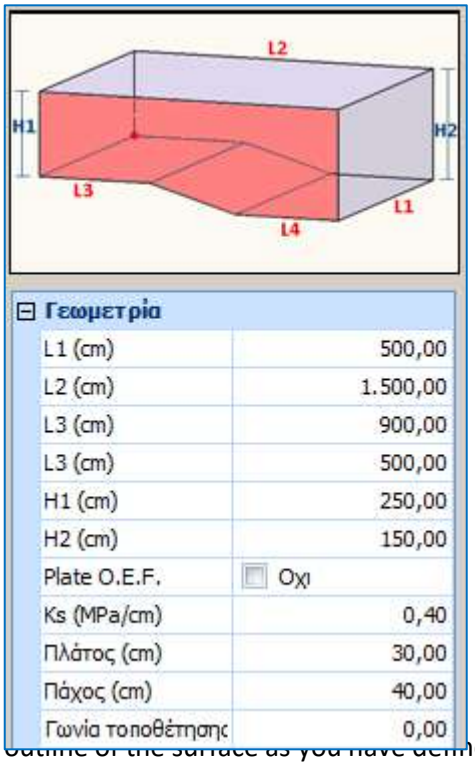


Select the mesh and then "Calculate" to calculate the "Meshing"



In this way the physical model of the surface is obtained and then proceed to the creation of the mathematical model with the command *"Tools">>"Model">>"Calculate",* which is described in detail in Chapter 4.

6.2.4 3D surfaces



Select one of the proposed 3D surfaces (e.g. the pool) and enter its geometric characteristics based on the design.

In case of paving, activate the checkbox

and enter the value of the spring constant Ks (MPa/cm).

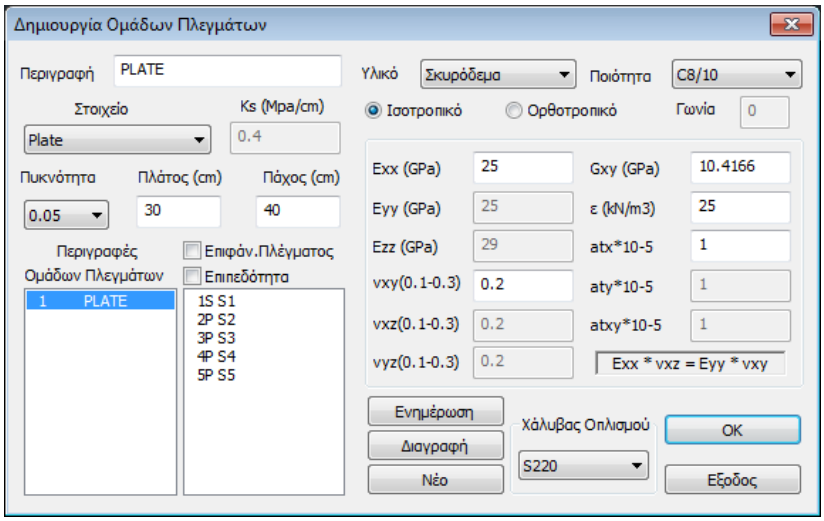
The values "Width" and "Thickness" refer to the dimensions of the surface that will simulate the specific surface
(Note: the density is taken by default equal to 0,15)*

'Mounting angle' is the angle (in degrees) at which the surface is to be inserted in relation to the total axes, in the XZ plane of the working surface.

e.g. for an angle of 30° the surface will be tilted as in the figure:

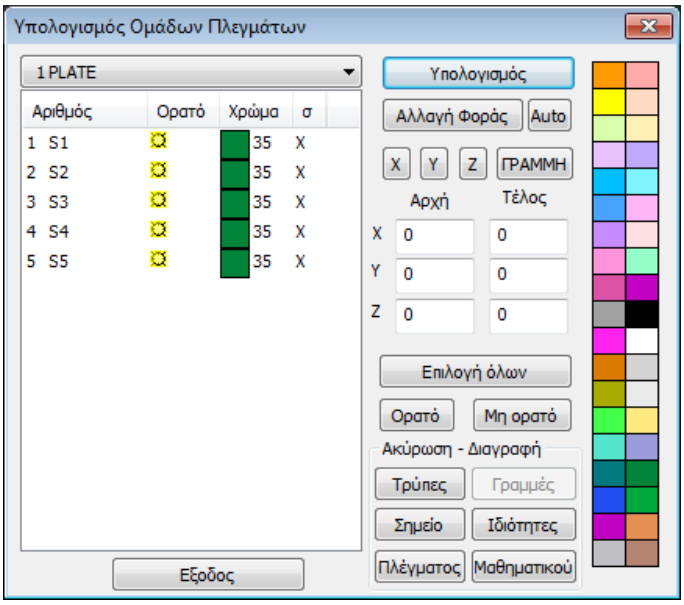
"OK" and the SCADA Pro interface will display the outline of the surface as you have defined it.

Within the "Modeling" field select the command *"Surface 3D">>"Grid"*. In the dialog box within the list of groups, the mesh as defined in the standard constructions is displayed, along with the subgroups. Select the mesh and subgroup in case you want to make changes and "*Update*". The command is described in detail in Chapter 2.

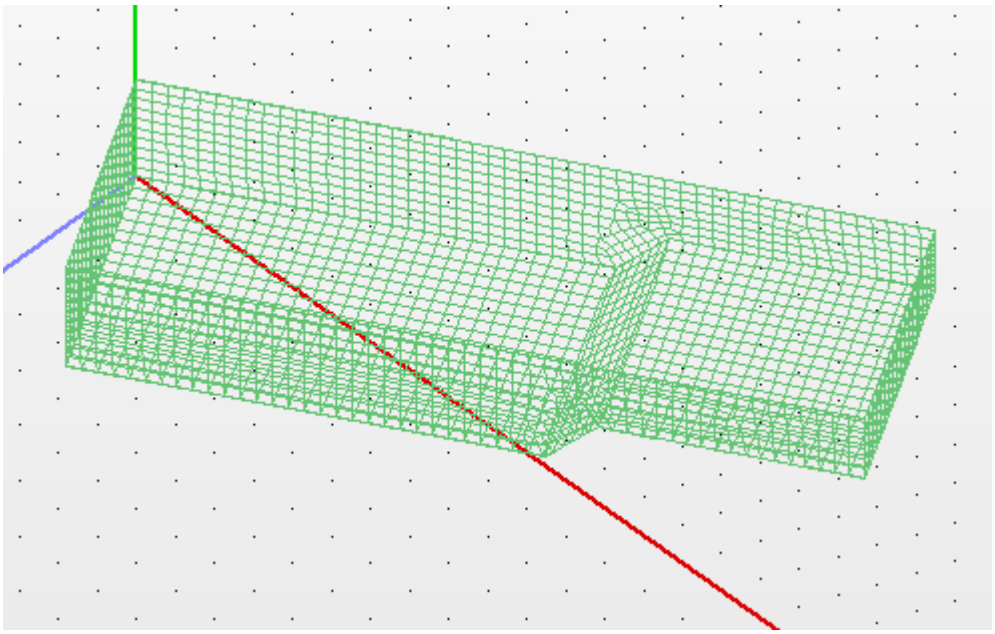


If no modifications are needed, go directly to the command

"Modeling>> "Surface 3D">>"Calculation", which opens the dialog box:



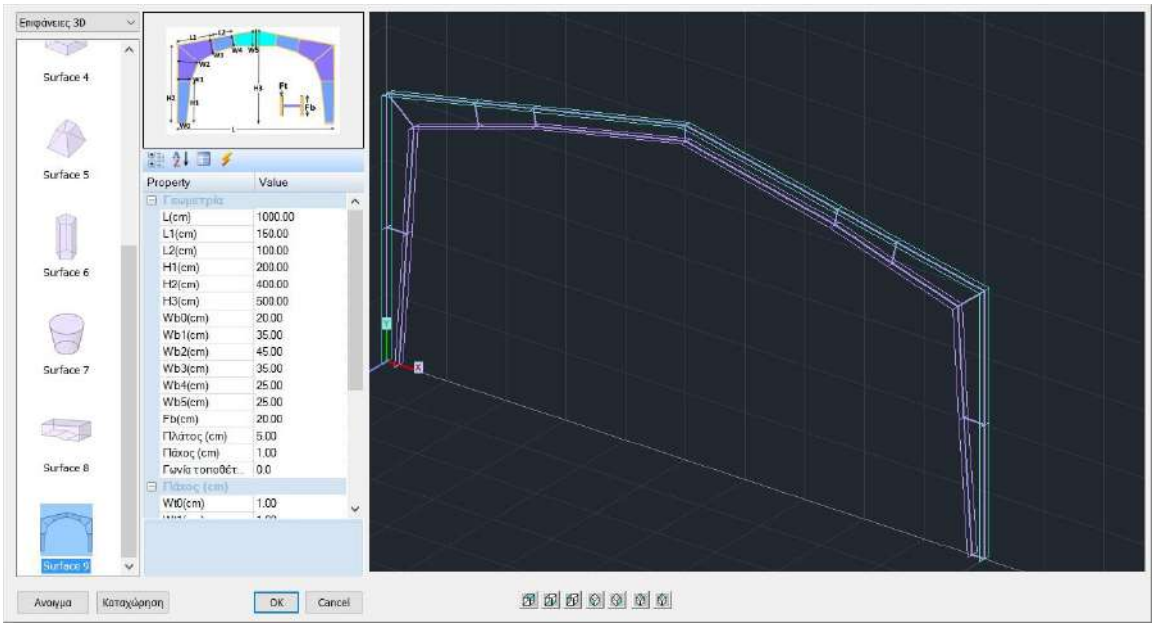
With "Calculate" you calculate the meshing.

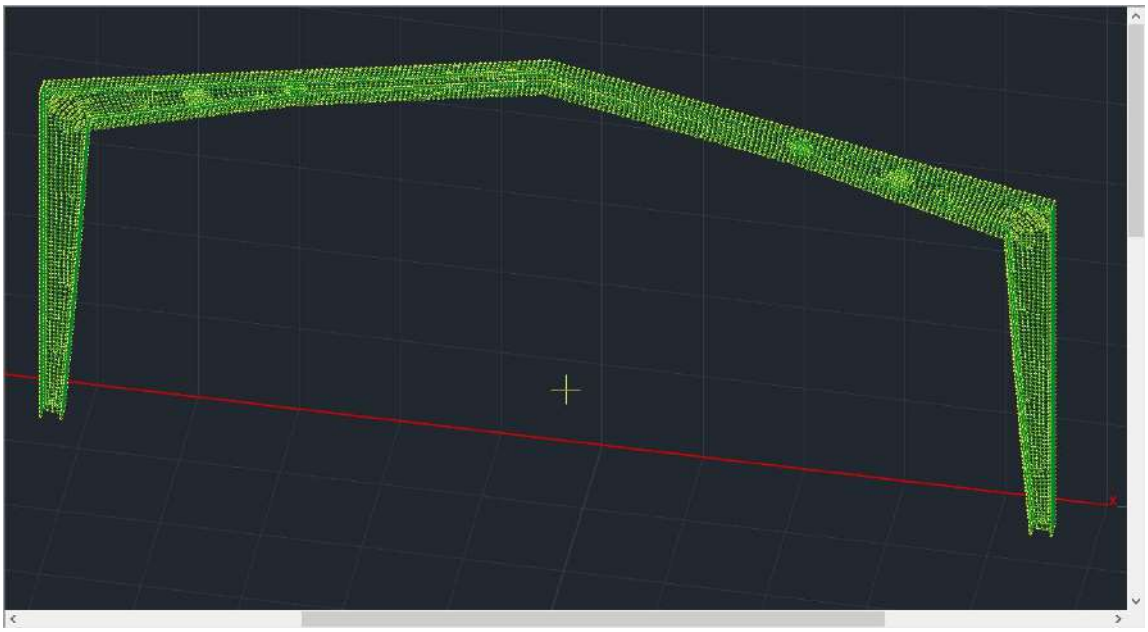


In this way the physical model of the pool is obtained and then proceed to the creation of the mathematical model with the command *"**Tools**">>"**Model**">> "**Calculation**",* which is described in detail in Chapter 4.

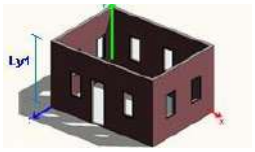
OBSERVATION:

There is also the possibility of automatic simulation of ***a typical metal frame of variable cross-section*** *with finite surface elements by defining the geometry and the corresponding thicknesses.*

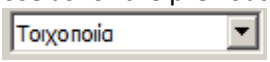




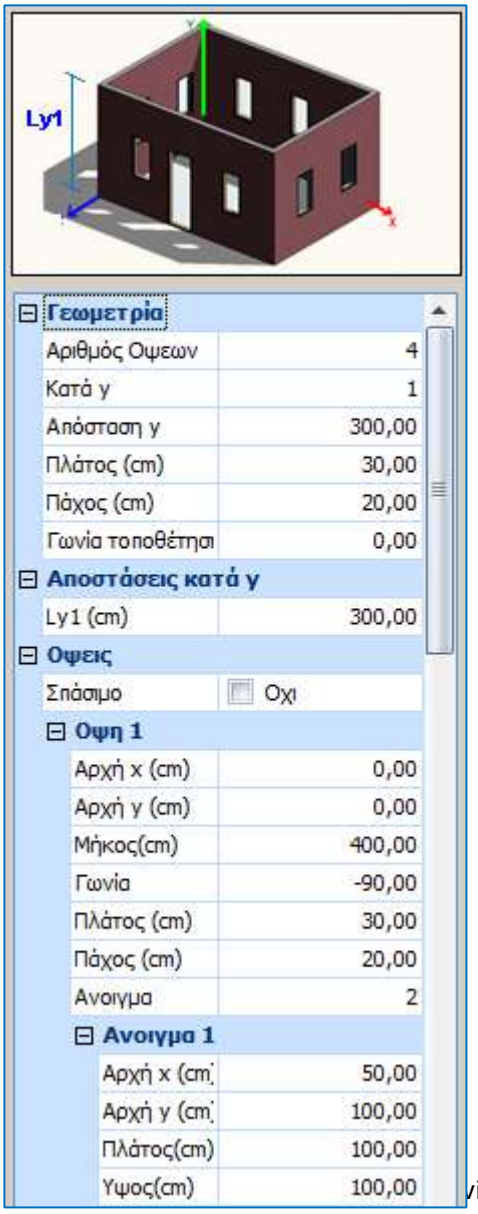
6.2.5 Masonry



With regard to masonry, the standard construction tool can be used in two ways to meet all requirements.

1st way : the classical way. We choose as for the previous constructions, an introduction point and in the context of 

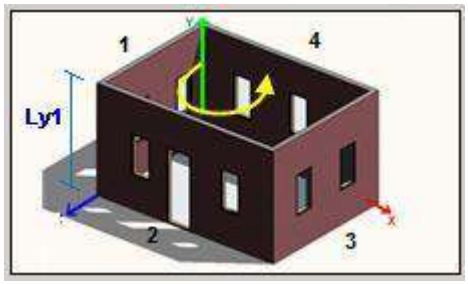
We choose the geometry, defining the number of faces, the number of y iterations (number of floors) and the distance y (height of floors). The width and thickness refer to the walls and the angle of insertion, the angle of insertion into the working surface in the XZ plane.



For more than one floor, you can set different floor heights in the "Distances by Y" field

"Breaking" faces is optional and what it does is to "break" each face into more than one surface, specifically in the middle of the holes, so that each face is simulated with continuous surfaces without holes. Otherwise, the simulation considers one surface for each face with its individual holes.

For each face you define: - its start coordinates and angle, in the XZ plane with respect to the local axes (as shown in the figure) and moving anti-clockwise - the width and thickness of the wall and - the number of openings.
Accordingly, you define the geometry and position of each opening.



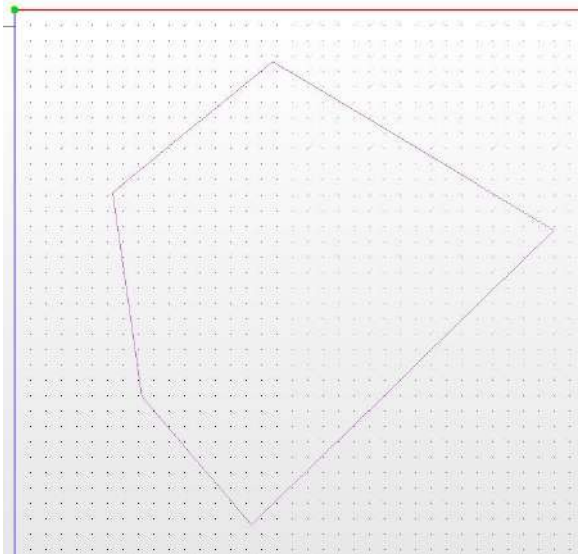
view and each opening, import the vector to the desktop by selecting OK.

Continue with the meshing process as described in the previous paragraph.

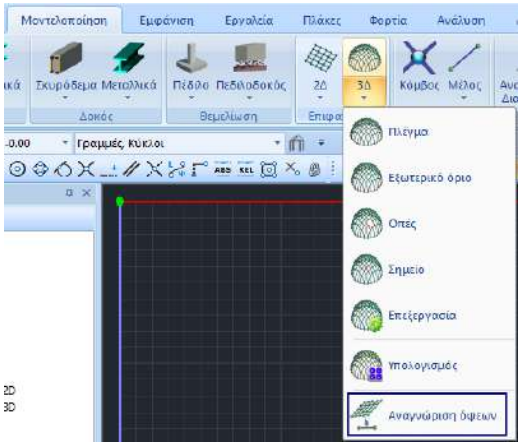
METHOD 2: SCADA Pro allows you to create any contour for the masonry and with the help of standard constructions to "build" your carrier easily and quickly.

The procedure is as follows:

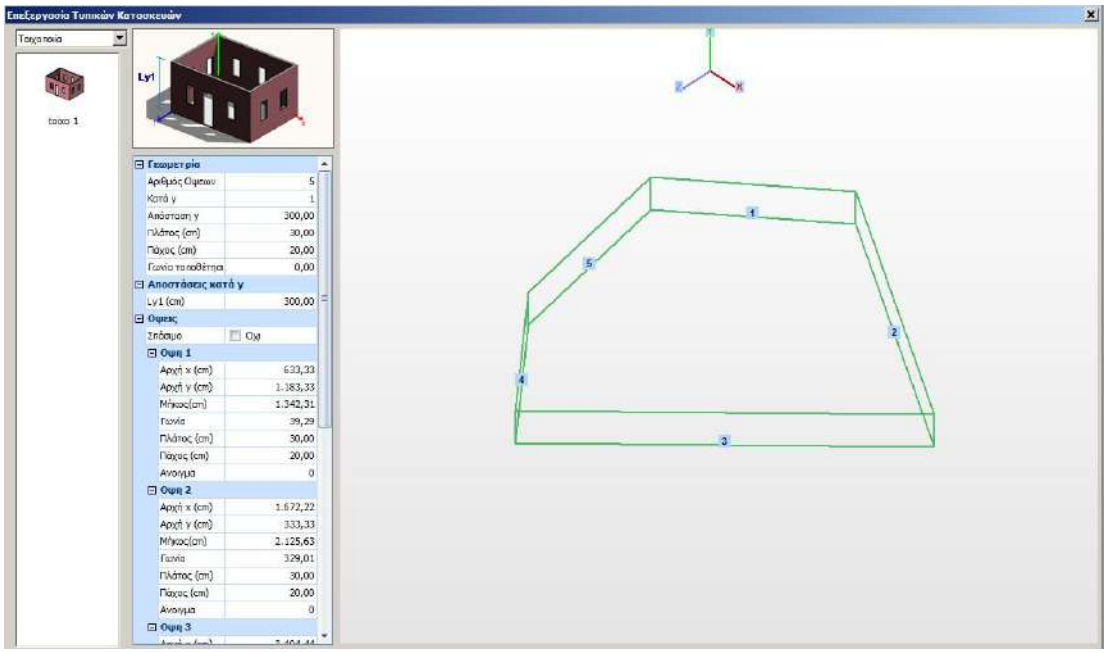
- Import a floor plan from an existing .dxf or .dwg file or using the commands within the "*Basic*" section draw a closed surface in the XZ plane of the desktop: "Draw">>"Line">>"Polyline" create surface right-click.



- Select the command in the *"Modeling"* section *>> "Surface 3D" >>* "***Face Recognition***"



and with Window  you select the whole floor plan. Right-click and the standard constructions box opens:



The program automatically recognizes the geometry of the floor plan. It suggests by default a height and creates the elevations in relation to the total axes.

- The user is asked to define the number of floors and the individual elevations, as well as the openings for each face, following the procedure of the 1st Way.

After completing the process for each view and each opening, import the vector to the desktop by selecting OK.
Continue with the meshing process as described in the previous paragraph.

OBSERVATION:
When <u>you have two or more floor plans from different dwg</u>, it is as follows:

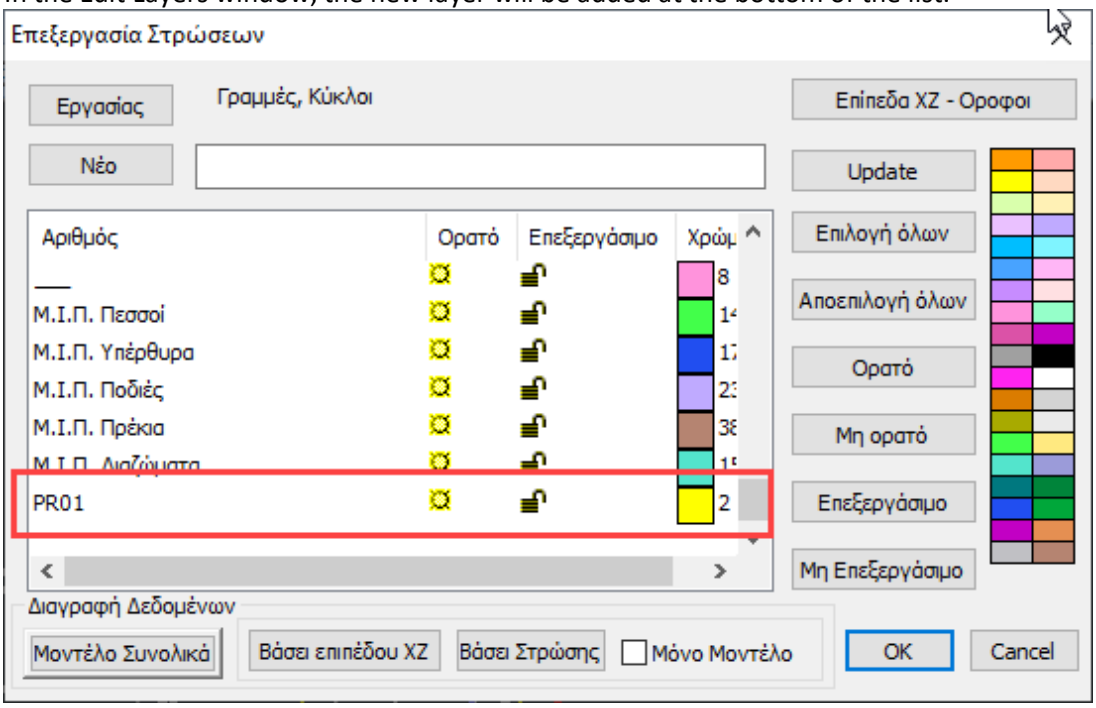
- You import the first dwg, do the usual face recognition and create the ground floor.
- Then you bring in the second dwg, do a face recognition and "glue" the first floor onto the ground floor.

You now have two main groups and lines that are identified and/or need to be broken. Same process for as many floors as I have.

OBSERVATION:

The Select layer field allows you to define a new layer to which these lines of the plan that were converted to SCADA Pro lines will belong. Select New to create the new layer with the name of the layer of the drawing or type a new name and then New.

In the Edit Layers window, the new layer will be added at the bottom of the list.



OBSERVATION:
This way you can separate the lines of the drawings of each level and select only the lines of the active level at a time, having made all other lines invisible and uneditable.

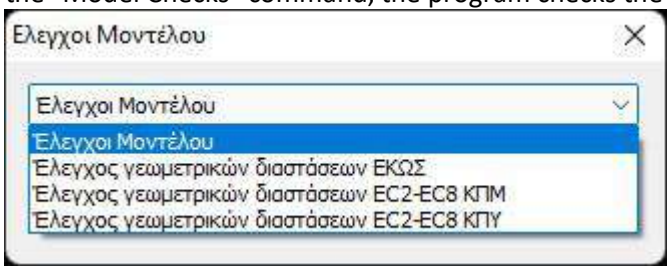
Finally, using the "Consolidate" command, you select all the main groups that have been created and create a new one that includes all the subgroups with their outlines now as they should be. If you wish, you can now delete the original groups and their outline lines. (see &4.2.1.2)

6.2.6 Swimming pools

In the new version of SCADA Pro a new integrated tool has been added for design, load calculation and automatic calculation combinations for in-ground pools, using 3D surface elements and standard structures.
A detailed description for the creation and calculation of in-ground pools can be found in the User Manual E.IN-GROUND POOLS.

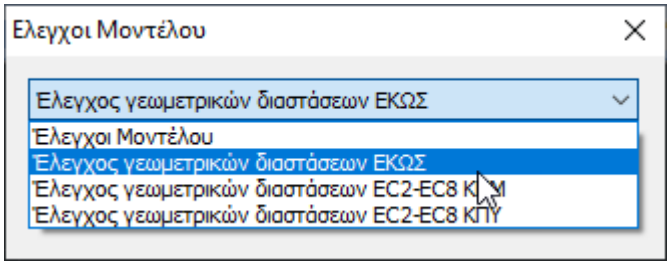
6.3 Model checks

After creating the physical and then the mathematical model of the study, by selecting the "Model Checks" command, the program checks the model for possible errors and warnings.



The screen displays a table with possible error messages related to the physical or mathematical model ("Err", number, message). Consult the messages and, where necessary, make the necessary modifications using the corresponding commands explained in detail in Chapter 3.

In addition, the new version of the program has incorporated checks concerning the dimensions of the cross-sections of the elements in accordance with the Eurocodes, as well as with the EAK and EKOS 2000.
The results of these checks are presented in the form of messages in the general checks of the program, after you select the desired regulation:



Error7001	Ο λόγος ανοίγματος Ι της δοκού 112 προς ύψος διατομής hw < 4 (l/hw=2.56).(Υψίκορμη)
Error7001	Ο λόγος ανοίγματος Ι της δοκού 112 προς ύψος διατομής hw < 4 (l/hw=2.56).(Υψίκορμη)
Error7001	Ο λόγος ανοίγματος Ι της δοκού 118 προς ύψος διατομής hw < 4 (l/hw=1.81).(Υψίκορμη)
Error7001	Ο λόγος ανοίγματος Ι της δοκού 118 προς ύψος διατομής hw < 4 (l/hw=1.81).(Υψίκορμη)
Error7001	Ο λόγος ανοίγματος Ι της δοκού 126 προς ύψος διατομής hw < 4 (l/hw=2.88).(Υψίκορμη)
Error7001	Ο λόγος ανοίγματος Ι της δοκού 126 προς ύψος διατομής hw < 4 (l/hw=2.88).(Υψίκορμη)
Error7008	Ο στύλος 59 έχει διάσταση < 25cm
Error7008	Ο στύλος 60 έχει διάσταση < 25cm

⚠ *"Err" is not always a sign of an error, it could just be a warning. The researcher should correct errors and heed the warnings.*

"RESULTS OF GENERAL AUDITS"

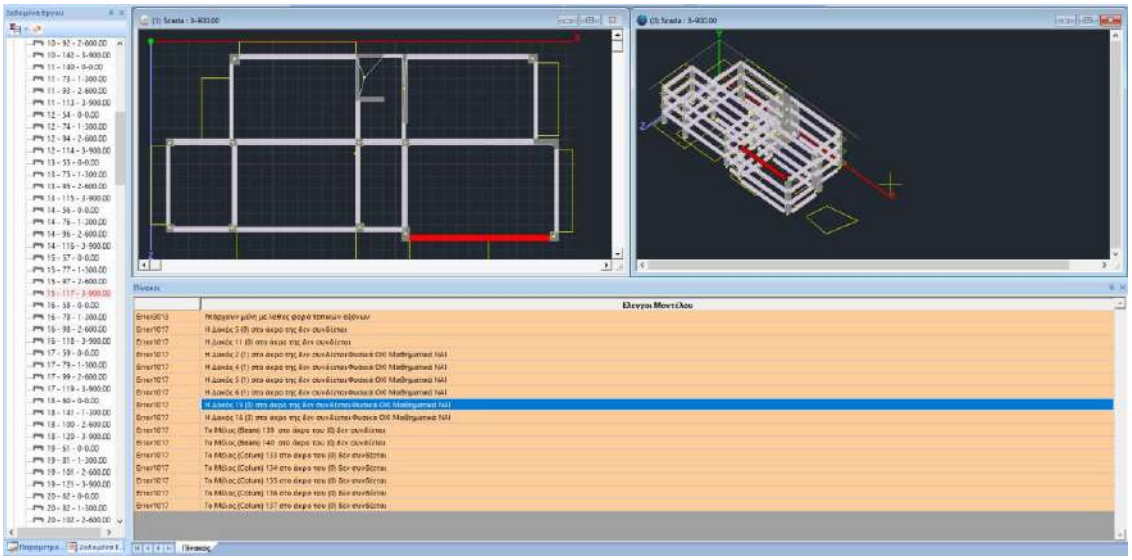
error1001 Beam %-d (%-d) does not have a mathematical
delegate\n Error1002 Pillar %-d (%-d) has no mathematical
representative\n
Error1003 Member %-d of Beam %d has a problem with the start-end nodes (zero)
(%-d-%-d)\n
Error1003 Member %-d of Beam %d (%-d) has identical start-end nodes (%-d)\n
Error1003 Member %-d of Column %d has a problem with start-end nodes
(zeroes) (%-d-%-d)\n
Error1004 Member %-d of Pillar %d (%-d) has identical start-end nodes (%-d)\n
Error1003 Member %-d of beam %d has short length = %-.2f \n
Error1003 Member %-d of the beam %d (%-d) has a short length = %-.2f
\n Error1004 Member %-d of Pillar %d has a short length = %-.2f\n
Error1004 Member %-d of Pillar %d (%-d) has a short length= %-.2f\n
Error1005 There are Members with the same name (%-d)\n
Error1006 Member (%-d) (%-d) is the same as Member (%-d) (%-d) (%-d)
(Syndication)\n Error1005 There are Members with the same name (%-d) \n
Error1006 Member (%-d)(%-d)(%-d) is the same as Member (%-d)(%-d)(%-d)
(Linkage)\n Error1005 There are Members with the same name (%-d) \n
Error1006 Member (%-d)(%-d)(%-d) is the same as Member (%-d)(%-d)(%-d)
(Linkage)\n Error1005 There are Members with the same name (%-d) \n
Error1006 Members (%-d)(%-d)(%-d) and (%-d)(%-d)(%-d) have the same start node
and the same end node.
Error1007 There are nodes with the same name (%-d)(%-d) \n
Error1007 Node %-d has the same coordinates as node %-d(%-d) [%-.3f cm]\n Error1008
Node (%-d)(%-d)(%-d) does not have a correct aperture node \n
Error1009 The start node of the beam %d is wrong\n Error1009
The end node of Beam %d is wrong\n Error1010
The start node of Pillar %d is wrong\n Error1011
The end node of Pillar %d is wrong\n Error1012 Node
%d of surface 3D %d is wrong\n
Error1013 The start node %-d of the Fieldbus %d (%-d) has the wrong degrees
of freedom\n
Error1014 The end node %-d of the Pedigree node %d (%-d) has the wrong
degrees of freedom\n
Error1015 The Member %-d of Beam %d (%-d) has an error %-
s\n Error1016 Member %-d of Pillar %d (%-d) is wrong %-s\n
Error1017 Beam %-d (%-d) at its end is not connected\n
Error1017 Beam %-d (%-d) at its end is not connected Physically NO Mathematically
YES\n Error1018 Beam %-d (%-d) has no connection to other members in the vector\n
Error1019 Column %-d (%-d) has no connection to other members in the vector\n
Error1020 In Field %-d, the Node of\n is not correct
error1021 In Field %-d the correspondence of the poles is not correct\n
error1022 The connection to Pillar %d (Node=%d)\n Error1301 Member %-d of Beam
%d is placed vertically (%-d-%-d)\n Error1049 On the surface %-d (level 0) there are surface
members which are not defined
as grids on elastic ground (paving)
error1301 The Member %-d of Pillar %d has been placed vertically (%-d-%-d)\n

Error1023 Pillar %-d does not belong to a floor (y=%-.3f)\n
Error1023 Beam %-d does not belong to a floor (y=%-.3f,%-
.3f)\n
error1024 The material of member %-d has an elasticity modulus value lower than the corresponding shear modulus
Error1009 Possible error in the dependencies of the degrees of freedom of Node %-d (y=%-.3f)\n
Error1017 Beam Member (Beam) %-d at its end (%-d) is not
connected\n Error1017 The Member (Colum) %-d at the end of (%-d) is not
connected\n
Error1007 Truss Member %-d(%-d) is connected to a node belonging to a septum\n
Error1678 The %-d post is placed in the wrong direction\n
Error3013 There are members with wrong local axis direction\n
Error1008 Node (%-d)(%-d)(%-d) is not connected to another element of the
construct\n Error4008 Member %-s of beam %-s has a zero value in some dimension\n
Error4008 Member %-s of Column %-s has a value of zero in some dimension\n
Error4114 Node %-s has a spring constant value of 0

warning3003 Check the surface element %-d\n
warning4001 Member (%-d) has release of all start-end moments\n Warning4002 Check
the degrees of freedom of the members that exist in the node (%-
(d)(%-d)\n
Warning4038 Node %-d has the wrong numbering\n
Warning4038 Member %-d (Beam) is misnumbered\n
Warning4038 Member %-d (Pillars) is wrongly numbered\n
Warning4039 Member %-d (Pillars) is not vertical\n

6.3.1 Model controls with direct display of the referenced item on the screen

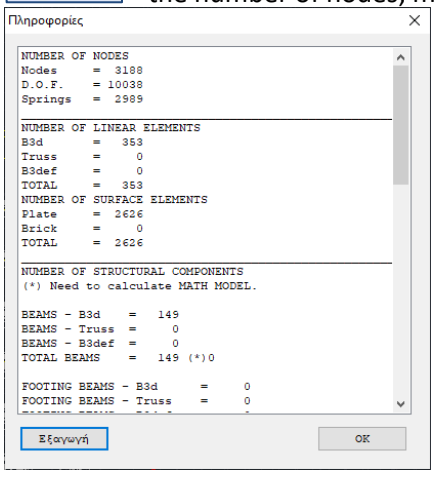
In the new version of SCADA Pro you now have the possibility to directly manage the errors and warnings that are displayed to you. More specifically, by clicking on the respective error message, then the reported element (member or node, or surface element) is displayed on the vector in red, as well as in the tree on the left. In this way you can easily and immediately process and correct all error messages.



For further clarification on the explanation of errors and errors in the warnings you can refer to the user manual of the WARNINGS & WARNINGS ERRORS.

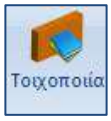
6.4 Information

Command for a comprehensive display of information concerning the active study: the number of nodes, members, structural elements, as well as volume, weight, etc.



NEW! Plus given η possibility export of the information.

7. Libraries



The "Libraries" command group includes the libraries for:
-**Masonry** and
-**Concrete sections**.

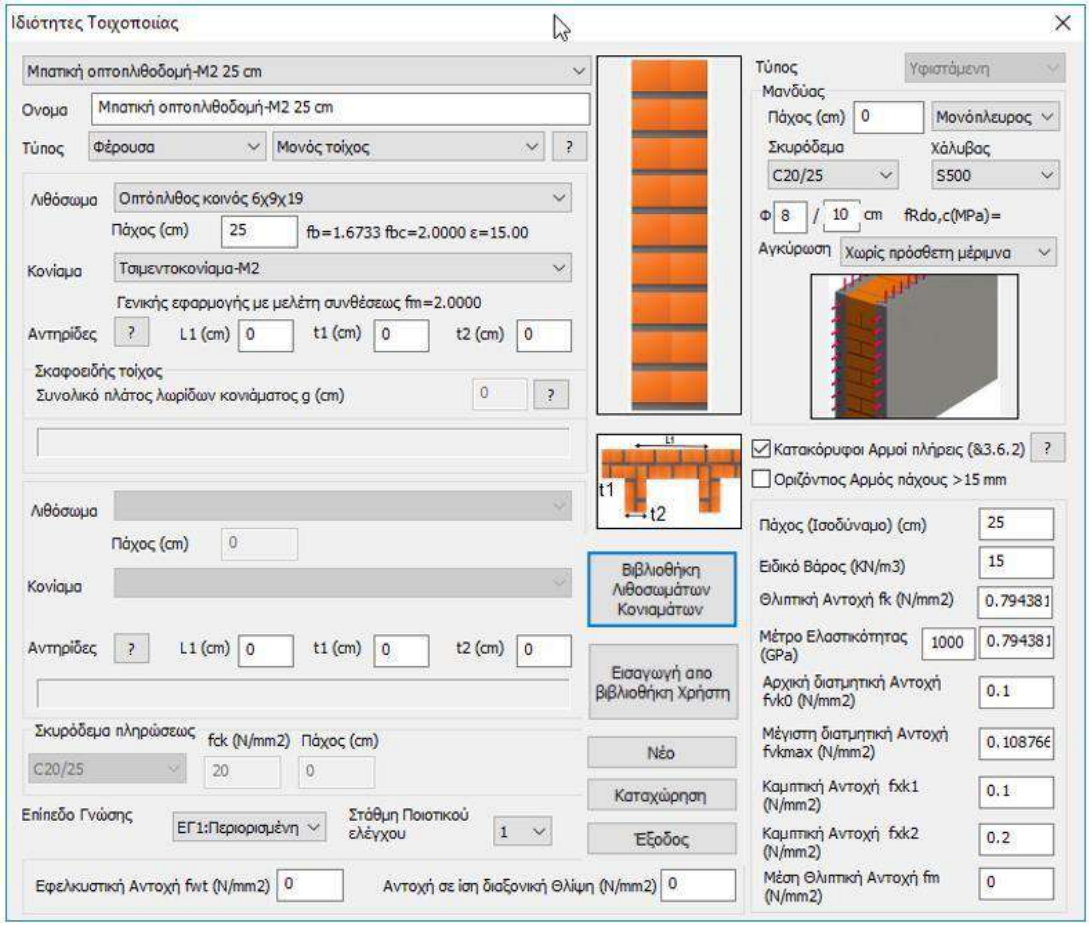
Libraries can be enriched by the user. The entries of data within them are not only for the active study, but also for each subsequent study.

7.1 Masonry

For structures made of **WOODEN WALLS**, except for ***Standard Masonry Structures*** and

the command ***Recognize Views*** that serve in the vector setup, with the command
you define the properties of the masonry, which you register with a name create your own library.

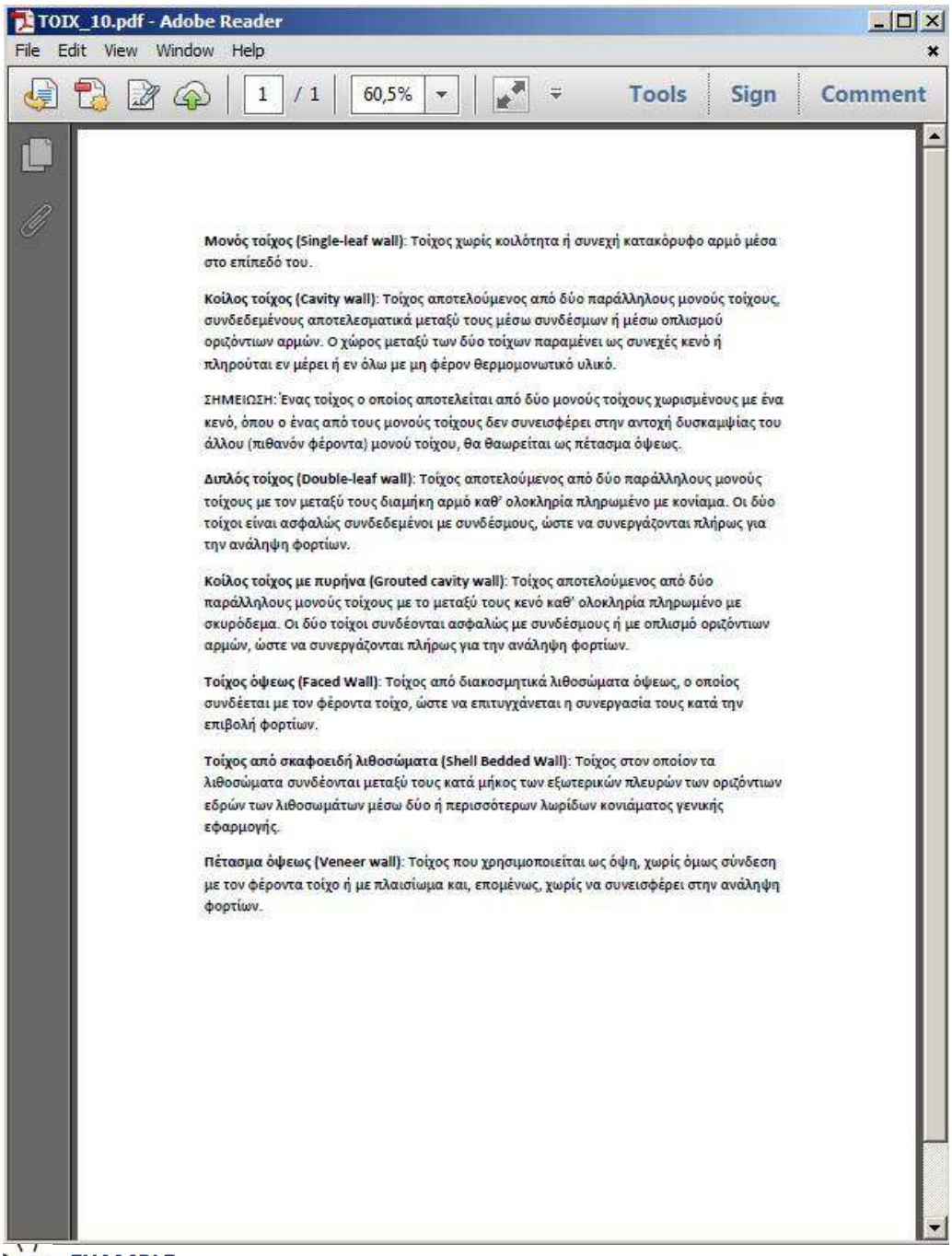
Selecting the command opens the dialog box:



Where, you either select one of the registered masonry units or create a new one by typing a name, selecting the *TYPE* and setting the corresponding properties for the **Stone**, **Mortar**, **Abutments**, **Concrete Filler** and **Sheathing**. You also define from the corresponding option whether the masonry is load-bearing or masonry infill.

⚠ *Depending on the option of the masonry TYPE, in the window dialog box some fields are activated or deactivated.*

The definitions of the different Types are displayed by selecting ? *on the right.*

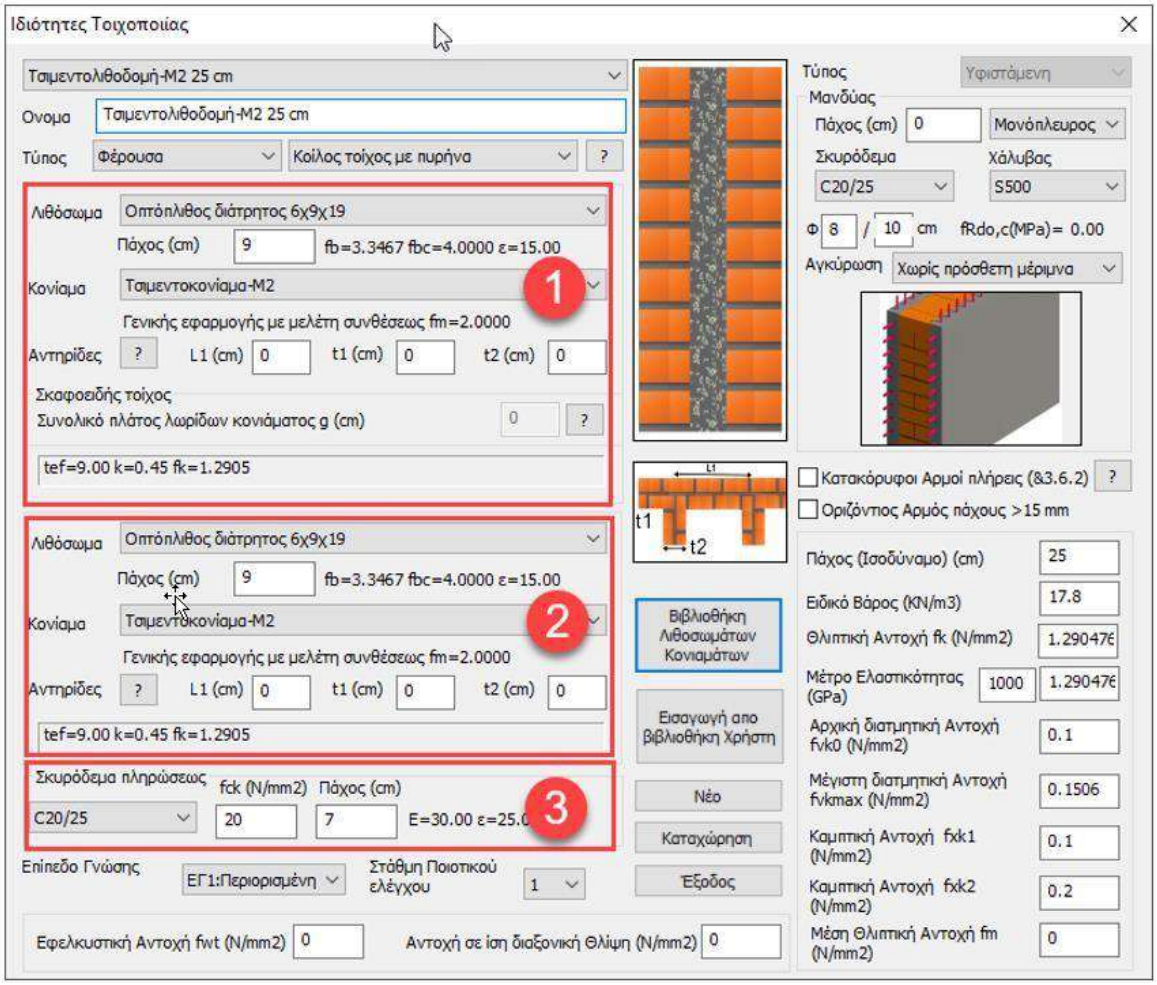


EXAMPLE:

Name: wall1
Type: Hollow wall with core

Κοίλος τοίχος με πυρήνα (Grouted cavity wall): Τοίχος αποτελούμενος από δύο παράλληλους μονούς τοίχους με το μεταξύ τους κενό καθ' ολοκληρία πληρωμένο με σκυρόδεμα. Οι δύο τοίχοι συνδέονται ασφαλώς με συνδέσμους ή με οπλισμό οριζόντιων αρμών, ώστε να συνεργάζονται πλήρως για την ανάληψη φορτίων.

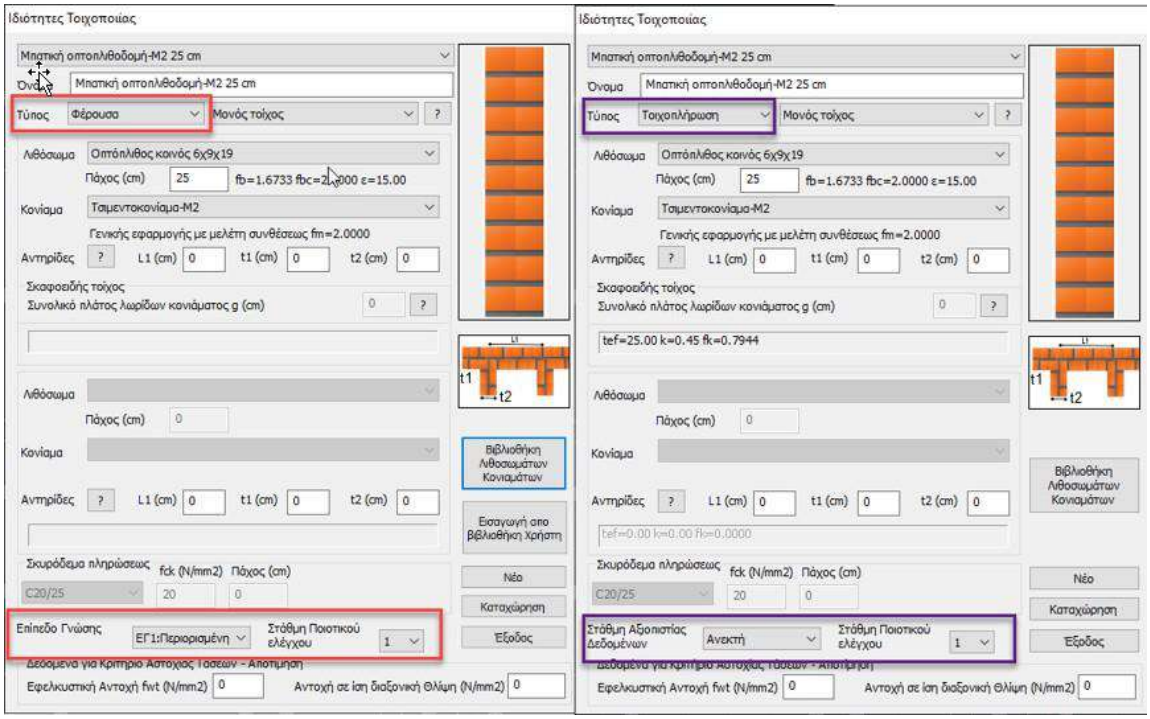
All fields in the window are active, since this type requires the definition of 2 single walls and the concrete infill.



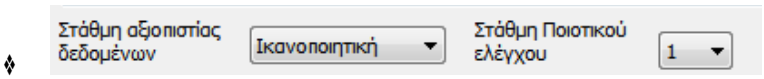
In the fields *wall1* & *wall2* you set for the

- **stonework**: the type and thickness
- **mortars**: the kind

and these options automatically update the corresponding coefficients
fb=3.3467 fbc=4.0000 ε=15.00



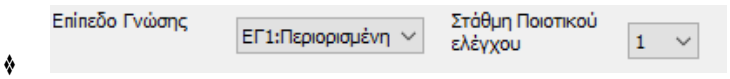
In the fields:



you specify

- the data confidence level according to the EIA if it is an existing masonry infill; and
- the quality control level if it is a new load-bearing masonry or an added masonry infill.

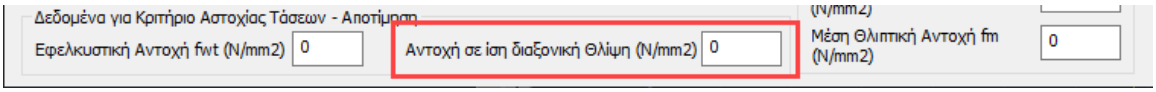
This option is replaced when you choose *load-bearing masonry*, with:



Knowledge Level (EC8-3 §3.3):
CI1: Limited -> CFEG1= 1.35 CI2: Normal -> CFEG2= 1.20
CI3: Full -> CFEG3= 1.00

Quality control level (EC6-1 - National Appendix): Safety factor gM

		1	2	3
	Τοιχοποιία από:			
Α	Λιθοσώματα Κατηγορίας Ι, κονίαμα με μελέτη συνθέσεως	1,7	2,0	2,2
Β	Λιθοσώματα Κατηγορίας Ι, προδιαγεγραμμένο κονίαμα	2,0	2,2	2,5
Γ	Λιθοσώματα Κατηγορίας ΙΙ, οποιοδήποτε κονίαμα	2,2	2,5	2,7



At the bottom of the window you will find, , the tensile strength fwt, the equivalent biaxial compressive strength and the mean compressive strength fm.

OBSERVATIONS:

- They relate to studies for the ***assessment of*** load-bearing masonry and the user has fill in the fields manually.
- For the *Average Compressive Strength* even when it remains 0, the program automatically calculates based on the compressive strength fk.

Η σχέση που συνδέει τη μέση θλιπτική αντοχή f_m με τη χαρακτηριστική θλιπτική αντοχή f_k, λαμβάνεται από τον ΚΑΝ.ΕΠΕ. (Παράρτημα 4.1 (§2.β) ή κεφάλαιο 7 (§7.4.1.ζ.2)) όπου εκεί χρησιμοποιείται για τις τοιχοπληρώσεις. Έτσι ισχύει ότι:

$f_m = \min(1.5 \cdot f_k, f_k + 0.50\ (\text{MPa}))$, (ΚΑΝ.ΕΠΕ. - Παράρτημα 4.1 (§2.β))

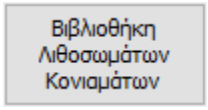
όπου:

f_m = μέση θλιπτική αντοχή,

f_k = χαρακτηριστική θλιπτική αντοχή.

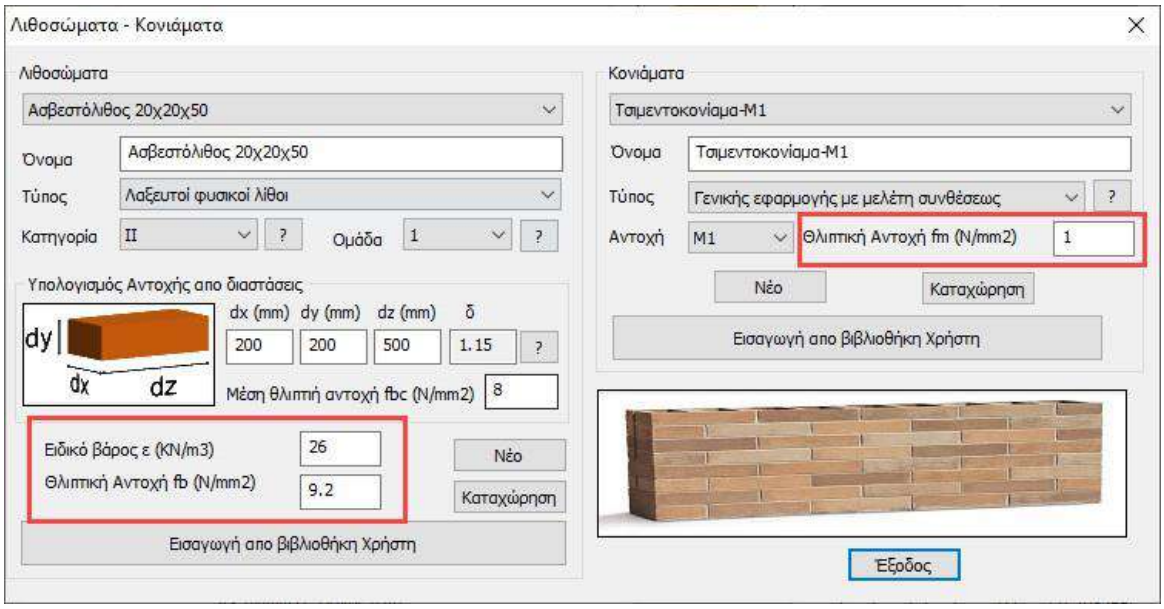
Στο Scada Pro η f_m μπορεί να δίνεται είτε ως τιμή από το χρήστη, είτε να υπολογίζεται αφού αυτός επιλέξει συνδυασμό λιθοσώματος και κονιάματος.

- The *Equal Biaxial Compression Strength* parameter is only necessary if the masonry is checked by ***a stress criterion***.



In the **Stone and Mortar Library** you will find ready-made stone, mortar and masonry typologies.

The user has the possibility to enter other mortars and mortars, simply by typing the name and specifying the type and group, for the compressive strength (which is automatically updated) and selecting "New".
You can also change the type and group of an existing stone or mortar and update it by clicking on "Entry".
In "Masonry" select from the stone and mortar lists, and create a new masonry type by clicking on "New". The specific weight and strength are calculated automatically.



The designer, in cases of assessment, may specify the values for the compressive strengths of the stones **(fb**) and mortars **(fm)** obtained from laboratory tests. On the basis of these values, the characteristic compressive strength of the masonry **(fk)** is obtained (EC6-1-1, §3.6.1.2).

$$f_k = K f_b^{0.7} f_m^{0.3}$$

$$f_k = K f_b^{0.85}$$

$$f_k = K f_b^{0.7}$$

- **Antiderides**

The choice of using **Braces**, affects the stiffness of the wall and therefore the active thickness:

? *To find out about the relevant article of the Regulation*.

toix_11.pdf - Adobe Reader

File Edit View Window Help

1 / 2 60,5% Tools Sign Comment

5.5.1.3 Ενεργό πάχος τοίχων

(1) Το ενεργό πάχος, t_{ef}, ενός μονού τοίχου, ενός διπλού τοίχου, ενός πετάσματος όψεως, ενός τοίχου από σκαφοειδή λιθοσώματα, και ενός κοίλου τοίχου με πυρήνα, όπως αυτοί ορίσθηκαν στην παράγραφο 1.5.10, θα πρέπει να λαμβάνεται ίσο με το πραγματικό πάχος του τοίχου.

(2) Το ενεργό πάχος ενός τοίχου, του οποίου η δυσκαμψία αυξάνεται μέσω αντηρίδων θα πρέπει να λαμβάνεται από την εξίσωση (5.10):

$$t_{ef}=\rho_t t \qquad (5.10)$$

όπου:

t_{ef} είναι το ενεργό πάχος

ρ_t είναι συντελεστής λαμβανόμενος από τον Πίνακα 5.1

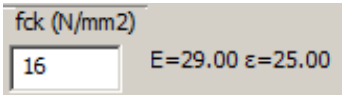
t είναι το πάχος του τοίχου

Πίνακας 5.1 – Συντελεστής δυσκαμψίας, ρ_t, για τοίχους με αντηρίδες

Λόγος της αξονικής απόστασεως των αντηρίδων προς το πλάτος των πεσσών	Λόγος του πάχους της αντηρίδας προς το πραγματικό πάχος του τοίχου με τον οποίο συνδέεται		
	1	2	3
6	1,0	1,4	2,0
10	1,0	1,2	1,4
20	1,0	1,0	1,0
Επιτρέπεται γραμμική παρεμβολή μεταξύ των τιμών του Πίνακα 5.1			

- **Concrete Filling**

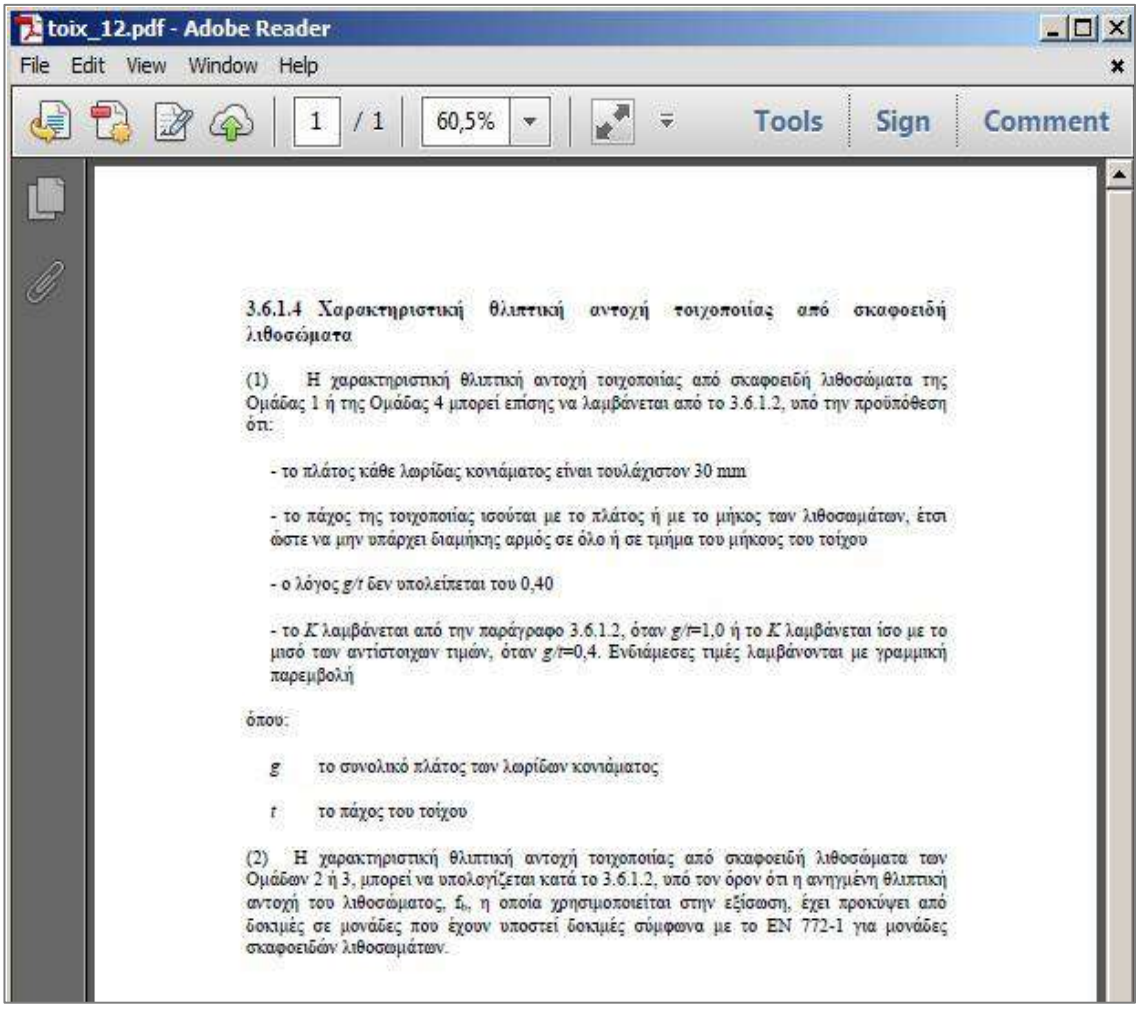
In the **Fill Concrete** field, select the quality and set the thickness.

The are automatically updated 

- **Scaphoid Lithosomes**

The choice of using **scaphoidal lithics**, affects the characteristic compressive strength of the Masonry.

To find out about the relevant article of the Regulation.



3.6.1.4 Χαρακτηριστική θλιπτική αντοχή τοιχοποιίας από σκαφοειδή λιθοσώματα

(1) Η χαρακτηριστική θλιπτική αντοχή τοιχοποιίας από σκαφοειδή λιθοσώματα της Ομάδας 1 ή της Ομάδας 4 μπορεί επίσης να λαμβάνεται από το 3.6.1.2, υπό την προϋπόθεση ότι:

- το πλάτος κάθε λωρίδας κονιάματος είναι τουλάχιστον 30 mm

- το πάχος της τοιχοποιίας ισούται με το πλάτος ή με το μήκος των λιθοσωμάτων, έτσι ώστε να μην υπάρχει διαμήκης αρμός σε όλο ή σε τμήμα του μήκους του τοίχου

- ο λόγος g/t δεν υπολείπεται του 0,40

- το K λαμβάνεται από την παράγραφο 3.6.1.2, όταν g/t=1,0 ή το K λαμβάνεται ίσο με το μισό των αντίστοιχων τιμών, όταν g/t=0,4. Ενδιάμεσες τιμές λαμβάνονται με γραμμική παρεμβολή

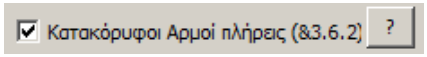
όπου:

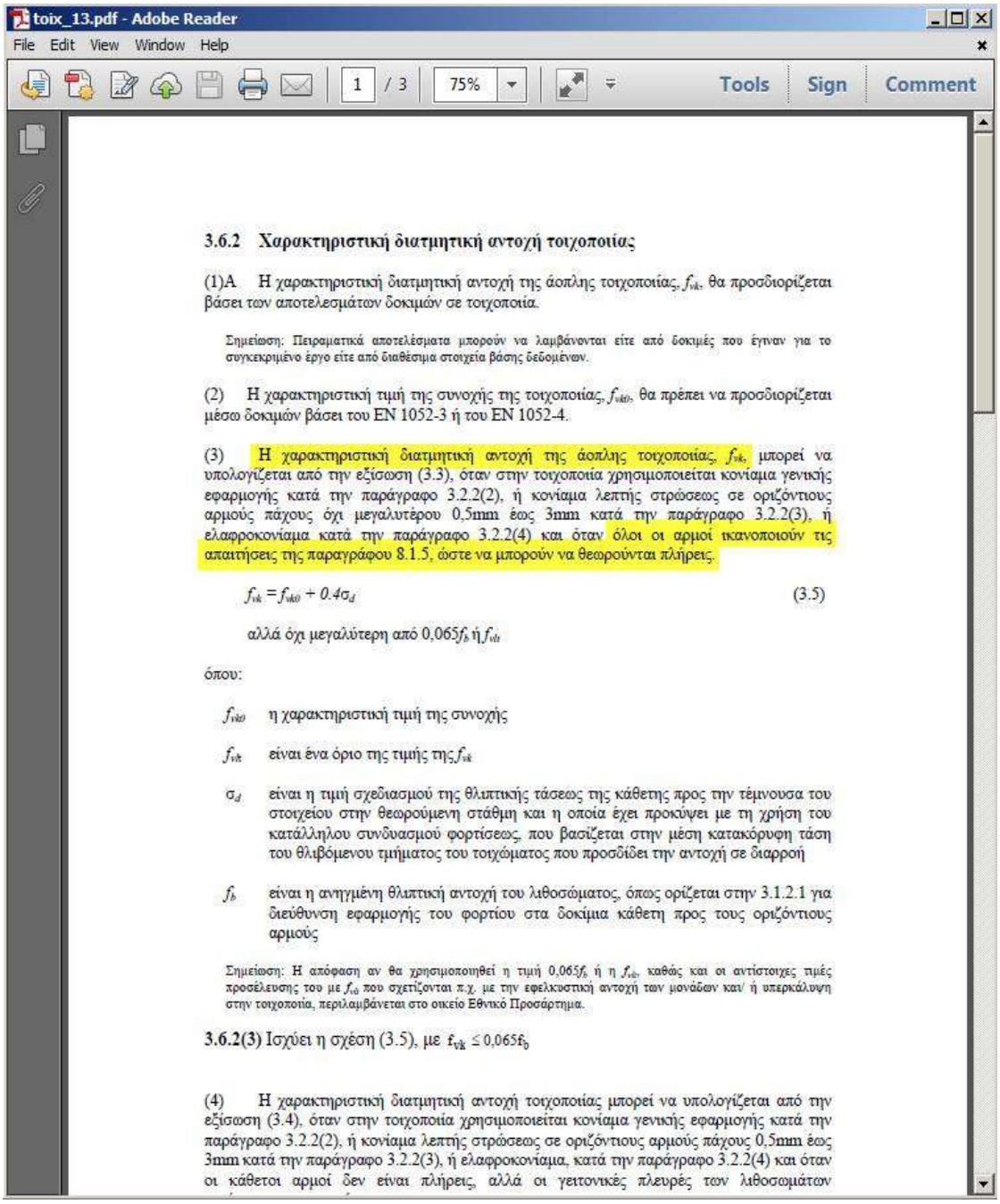
g το συνολικό πλάτος των λωρίδων κονιάματος

t το πάχος του τοίχου

(2) Η χαρακτηριστική θλιπτική αντοχή τοιχοποιίας από σκαφοειδή λιθοσώματα των Ομάδων 2 ή 3, μπορεί να υπολογίζεται κατά το 3.6.1.2, υπό τον όρον ότι η ανηγμένη θλιπτική αντοχή του λιθοσώματος, f_b, η οποία χρησιμοποιείται στην εξίσωση, έχει προκύψει από δοκιμές σε μονάδες που έχουν υποστεί δοκιμές σύμφωνα με το ΕΝ 772-1 για μονάδες σκαφοειδών λιθοσωμάτων.

The calculation of the characteristic shear strength of masonry based on formula (3.5) requires the joints to meet the requirements that make them considered complete.
In this case, check the corresponding checkbox to use (3.5) for the calculation.





3.6.2 Χαρακτηριστική διατμητική αντοχή τοιχοποιίας

(1)A Η χαρακτηριστική διατμητική αντοχή της άοπλης τοιχοποιίας, f_{vk}, θα προσδιορίζεται βάσει των αποτελεσμάτων δοκιμών σε τοιχοποιία.

Σημείωση: Πειραματικά αποτελέσματα μπορούν να λαμβάνονται είτε από δοκιμές που έγιναν για το συγκεκριμένο έργο είτε από διαθέσιμα στοιχεία βάσης δεδομένων.

(2) Η χαρακτηριστική τιμή της συνοχής της τοιχοποιίας, f_{vk0}, θα πρέπει να προσδιορίζεται μέσω δοκιμών βάσει του EN 1052-3 ή του EN 1052-4.

(3) Η χαρακτηριστική διατμητική αντοχή της άοπλης τοιχοποιίας, f_{vk}, μπορεί να υπολογίζεται από την εξίσωση (3.3), όταν στην τοιχοποιία χρησιμοποιείται κονίαμα γενικής εφαρμογής κατά την παράγραφο 3.2.2(2), ή κονίαμα λεπτής στρώσεως σε οριζόντιους αρμούς πάχους όχι μεγαλυτέρου 0,5mm έως 3mm κατά την παράγραφο 3.2.2(3), ή ελαφροκονίαμα κατά την παράγραφο 3.2.2(4) και όταν όλοι οι αρμοί ικανοποιούν τις απαιτήσεις της παραγράφου 8.1.5, ώστε να μπορούν να θεωρούνται πλήρεις.

$$f_{vk} = f_{vk0} + 0.4\sigma_d \qquad (3.5)$$

αλλά όχι μεγαλύτερη από $0{,}065f_b$ ή f_{vlt}

όπου:

f_{vk0} η χαρακτηριστική τιμή της συνοχής

f_{vlt} είναι ένα όριο της τιμής της f_{vk}

σ_d είναι η τιμή σχεδιασμού της θλιπτικής τάσεως της κάθετης προς την τέμνουσα του στοιχείου στην θεωρούμενη στάθμη και η οποία έχει προκύψει με τη χρήση του κατάλληλου συνδυασμού φορτίσεως, που βασίζεται στην μέση κατακόρυφη τάση του θλιβόμενου τμήματος του τοιχώματος που προσδίδει την αντοχή σε διαρροή

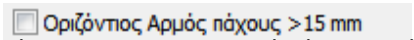
f_b είναι η ανηγμένη θλιπτική αντοχή του λιθοσώματος, όπως ορίζεται στην 3.1.2.1 για διεύθυνση εφαρμογής του φορτίου στα δοκίμια κάθετη προς τους οριζόντιους αρμούς

Σημείωση: Η απόφαση αν θα χρησιμοποιηθεί η τιμή $0{,}065f_b$ ή η f_{vlt}, καθώς και οι αντίστοιχες τιμές προσέλευσης του με f_{v0} που σχετίζονται π.χ. με την εφελκυστική αντοχή των μονάδων και/ ή υπερκάλυψη στην τοιχοποιία, περιλαμβάνεται στο οικείο Εθνικό Προσάρτημα.

3.6.2(3) Ισχύει η σχέση (3.5), με $f_{vk} \leq 0{,}065f_b$

(4) Η χαρακτηριστική διατμητική αντοχή τοιχοποιίας μπορεί να υπολογίζεται από την εξίσωση (3.4), όταν στην τοιχοποιία χρησιμοποιείται κονίαμα γενικής εφαρμογής κατά την παράγραφο 3.2.2(2), ή κονίαμα λεπτής στρώσεως σε οριζόντιους αρμούς πάχους 0,5mm έως 3mm κατά την παράγραφο 3.2.2(3), ή ελαφροκονίαμα, κατά την παράγραφο 3.2.2(4) και όταν οι κάθετοι αρμοί δεν είναι πλήρεις, αλλά οι γειτονικές πλευρές των λιθοσωμάτων

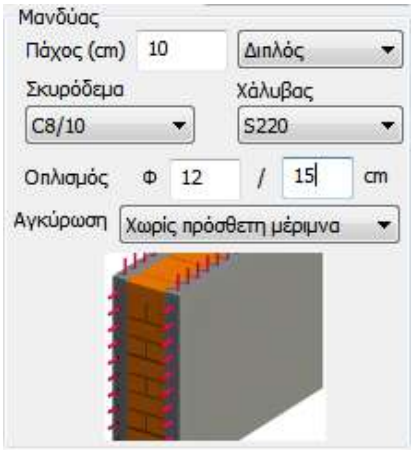
- **Cloak**



This option is activated when you have joints larger than 15 mm and is used in the case of wall fillings.

In the case where the use of **sheathing** in masonry is required, its geometric characteristics, the quality of materials and reinforcement are defined.

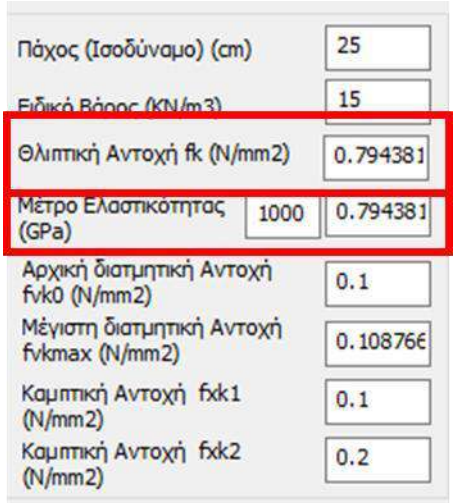
You also define how the reinforcement is anchored. This option is only taken into account in wall fillings according to the CEE.



- **Equivalent Wall**

The overall results for the masonry are calculated by the program, based on the data considered, and transferred to the global table.

- If the user knows the values for the equivalent wall, he can enter them directly.



<u>Typical masonry strength fk:</u>

Apart from the characteristic compressive strength of masonry resulting from EC6-1-1, the designer can calculate and enter it as a final value, based on another scientifically accepted method (Tasios
- Chronopoulos (1986) O.Brocker, Tasios (1985))

In addition, in the case of reinforcement with **Deep Mortar** or **Enemeta**, you define here the compressive strength of the reinforced masonry according to the respective formulas:

$$f_{wc} = \frac{1}{\gamma_{Rd}} \cdot \zeta \cdot f_{wc,o}$$ (Deep Harmony)

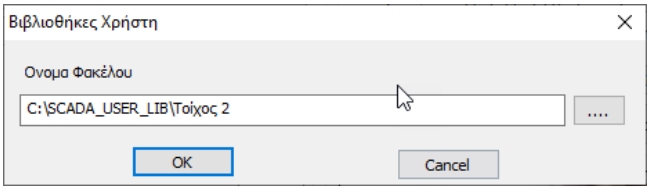
$$f_{wc,i} = f_{wc,0}\left(1 + \frac{V_i}{V_w}\frac{f_{c,in}}{f_{wc,0}}\right)$$ (Enmeta)

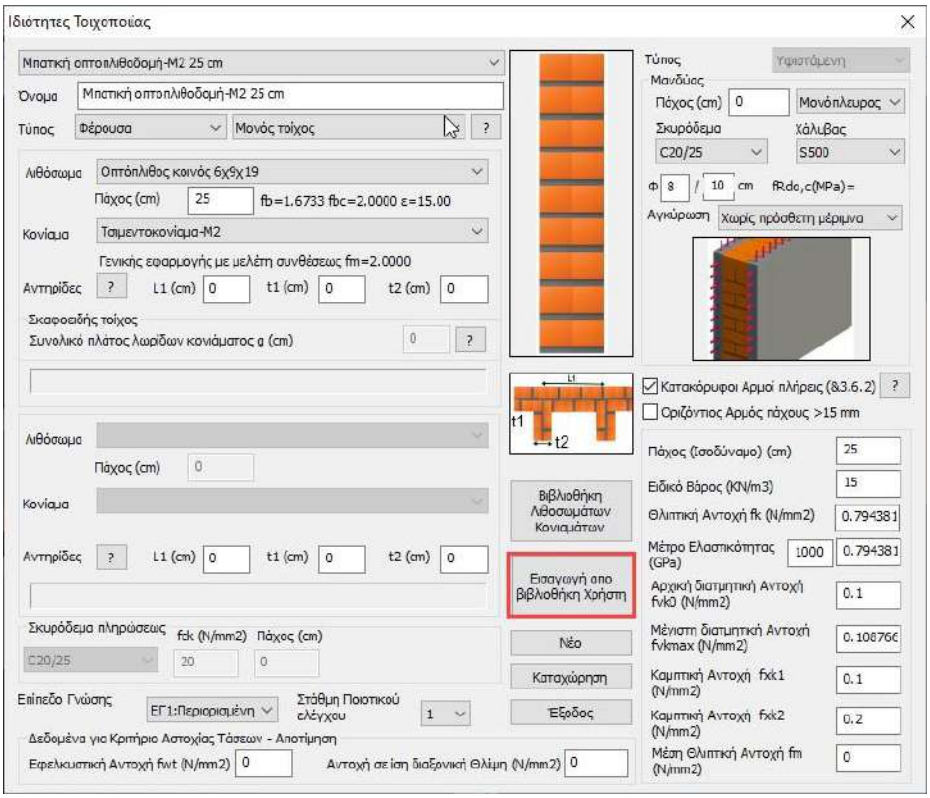
Elasticity Measure E:

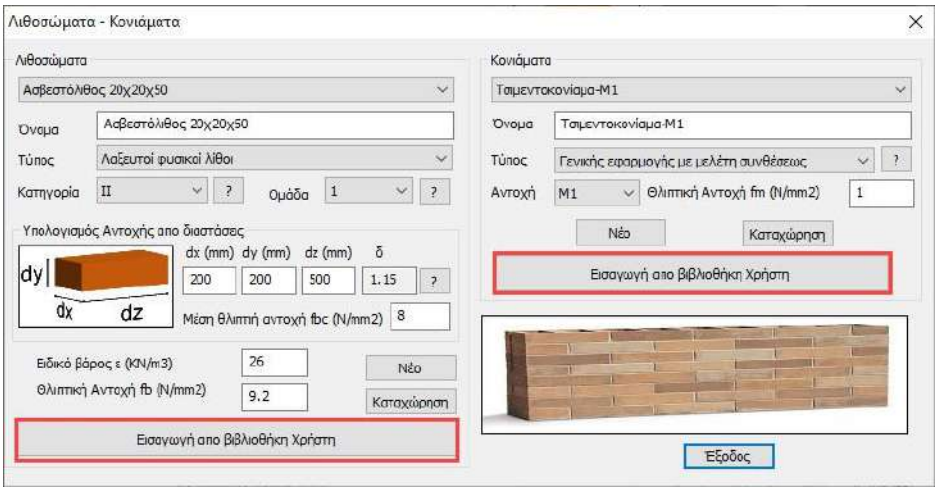
In the literature there is a wide variation in the values given for the Measure Elasticity. Generally acceptable values are: $E = (400 \div 1000) f_{wc}$ where the scholar can give the one he considers most reliable.

NEW!!!!

With the new Libraries command found in Add-ons, the user is now able to create their own material libraries. The libraries are listed in a separate folder and can be used in different studies. Each new project now has access to this library. So when within Modeling you select Masonry, in the dialog box there is now the option: Εισαγωγή απο βιβλιοθήκη Χρήστη, both in Masonry and Βιβλιοθήκη Λιθοσωμάτων Κονιαμάτων which allows you to access the library and select your saved materials:

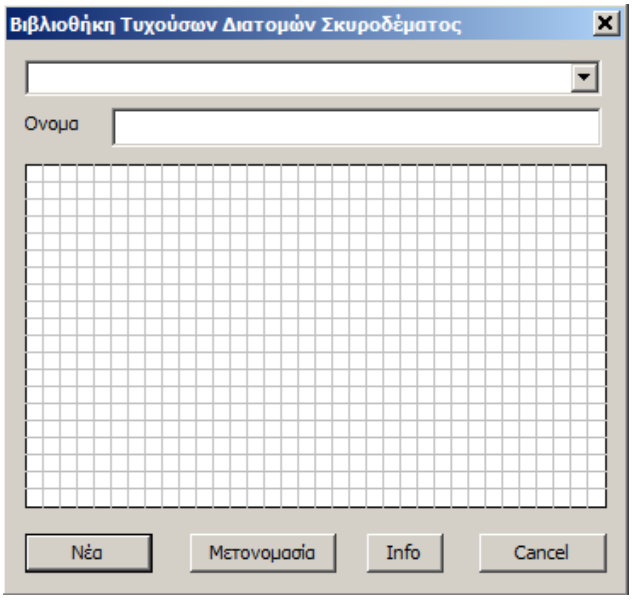






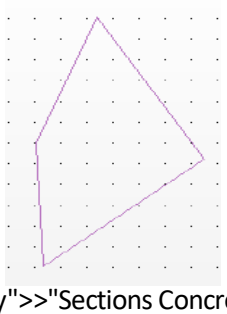
7.2 Concrete cross-sections

You can create any random pole cross-section by simply defining its outline. Its centre of gravity and all its inertial elements are automatically calculated using the boundary element method. The cross-section is automatically saved in your own library.
The first time you select the **Concrete Sections Library**, the blank window appears:



To import your own cross sections into the library, and to be able to call them each time, the procedure is as follows:

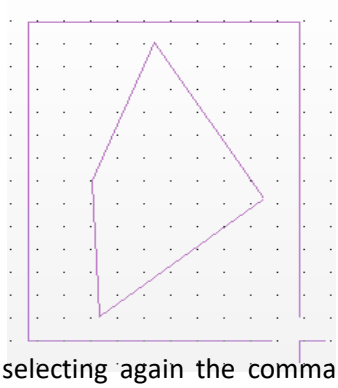
- From the "Basic" section a command is selected to draw the closed contour of the random section or alternatively import a dwg or dxf file with the shape of the random section.

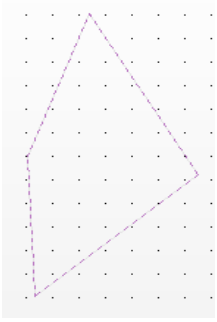


- Select the command "Library">>"Sections Concrete" and in the "Library">>"Sections Concrete" and in the

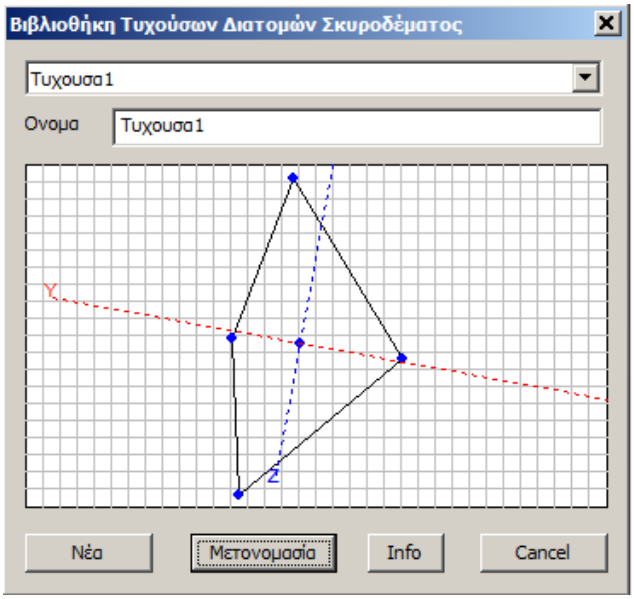
dialog box type a name (at least 3 characters) and "New".

- You activate the "windowed selection  . You select the shape by left clicking and dragging the window so that you get the whole shape. Left click again and the shape becomes dotted



and  right click to finish

- By selecting again the command "**Library**">>"**Concrete Sections**" in the window the cross-section is displayed with selectable insertion points and local axes. By changing the name in the corresponding field and selecting Rename you can rename the cross-section.



NOTE: All random cross-sections you have created are registered and you can find them at any time in the list

Select  to read all the geometric and inertial characteristics of the random cross-section.

Info

Περιγραφή	Τιμή
Επιφάνεια A (m2)	39.466
Καθαρή Επιφάνεια Ak (m2)	39.466
Στρεπτική Ροπή Αδράνειας Ix (dm4)	710862...
Καμπτική Ροπή Αδράνειας Iy (dm4)	243877...
Καμπτική Ροπή Αδράνειας Iz (dm4)	775533...
Επιφάνεια Διάτμησης Asy (m2)	32.888
Επιφάνεια Διάτμησης Asz (m2)	32.888
Γωνία Beta b	-10.417
Μέτρο Ελαστικότητας (GPa)	25.000
Μέτρο Διάτμησης G (GPa)	10.417
Ειδικό Βάρος ε (kN/m3)	25.000
Συντελεστής Θερμικής Διαστολής at*10^-5	1.000

Εξοδος

To see how to insert a random cross-section substructure into the model open the "Substructures" command and find it in the list of cross-sections.