SCADA Pro™ 18
Structural Analysis & Design

Example 6
Masonry Structure Analysis and Design

ACE HELLAS
INTEGRATED SOLUTIONS

Contents

• OVERVIEW

SCADA Pro new version is a result of more than 40 years of research and development while containing all the innovative capabilities and top-notch tools for the construction business.
SCADA Pro utilizes a compact and fully adequate platform for constructing new buildings (analysis and design) or existing ones (check, assessment, and retrofitting).
The software employs the Finite Element Method, combining line and plane finite elements in a smooth way. For design purposes, the user is offered all the Eurocodes as well as all the relevant Greek regulations (N.E.A.K, N.K.O.S., E.K.O.S. 2000, E.A.K. 2000, E.A.K. 2003, Old Antiseismic, Method of permissible stresses, KAN.EPE).
There are numerous possibilities offered for the modeling of various kind of structures. Structures made of reinforced concrete, steel, timber, masonry, or composite structures are now fully feasible.
Several smart operations add on to the practicality and usability of the software. The user can produce the model of a structure no matter how complicated it is, work at ease with the 3D model, process through the steps of analysis and design in a convenient way, up to the conclusion of what initially may seem the most demanding project.
SCADA Pro is presented to you as a powerful tool to meet the highest needs of modern civil engineering!

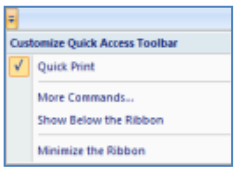
• INTRODUCTION

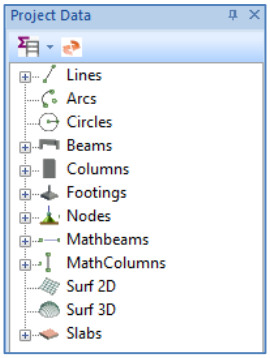
The current manual comes as an aid for a new user of SCADA Pro, making the interface of the software as familiar as possible. It consists of several chapters, where one after the other, describes the consecutive steps of a simple example of a loadbearing masonry project.
The most useful information is presented, in regards to the best possible understanding of the software commands and logic, as well as the process that has to be followed.

• THE NEW INTERFACE

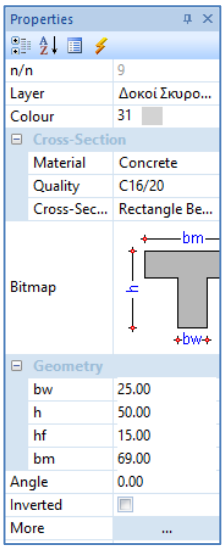
The new interface of the SCADA Pro software is based on the RIBBON structure, thus, the several commands and tools are reached neatly. The main idea of the RIBBON structure is the grouping of commands that have small differences and work in the same context, in a prominent position different to each group. This converts the use of a command, from a tedious searching procedure through menus and toolbars, into an easy to remember the chain of two or three clicks of the mouse button.

The user can collect his/her most popular commands into a new group, for an even faster access. This group remains as it is for future analyses after the program ends. Different commands can be added to it or removed from it, and its placing in the workspace may be altered through the “Customize Quick Access Toolbar” utility.





Apart from the RIBBON structure, all the entities that a structure consists of are presented in a tree structure, at the left side of the SCADA Pro main window, either for the whole structure or at each level of the structure. This categorization enhances the use of each entity. When the tree structure is choosing an entity, it is highlighted at the graphical interface and the level of the structure that contains this entity is isolated. At the same time, at the right side of the window, the entity's properties appear. The user can check or modify any of these properties at once. Conversely, the entity can also be chosen at the graphical interface, and automatically it is presented, at the left side in the tree structure and at the right side with its properties. The right-click mouse button can be very helpful here, since several commands and features, distinct for each entity, can be activated with it.

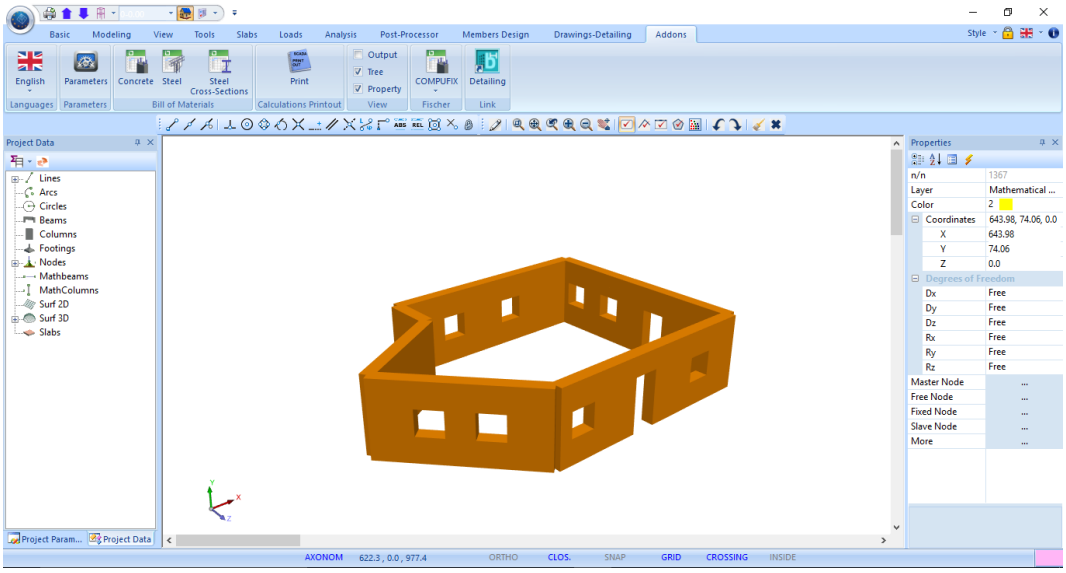


The "Properties" list that shows up at the right side of the window, not only shows all the properties of the entity shown but can be used for any quick and easy changes, the user wants to make, too.

1. GENERAL DESCRIPTION

1.1 Geometry

The considered single floor masonry structure consists of 6 views with openings and raft foundation.



1.2 Materials

All walls are of single-leaf type with dimensional natural stone units 20x20x25 and M5 mortar named, “Wall M5 0.50”. For the raft, concrete C20/25 and Reinforcing Steel B500C was used.

1.3 Regulations

Eurocode 8 (EC8, EN1998) for seismic loads.
Eurocode 2 (EC2, EN1992) for the design of the concrete elements.

1.4 Load and Analysis assumptions

Dynamic Spectrum Analysis with pairs of torsional moment along the same direction.
The loads by the method above are:
(1) G (dead)
(2) Q (live)
(3) EX (node loads, seismic forces along XI axes, derived from dynamic analysis).
(4) EZ (node loads, seismic forces along ZII axes, derived from dynamic analysis).
(5) Erx ±(node torsional moments, derived from node seismic forces along XI axes, offset by the accidental eccentricity ±2eτzi).

(6)Erz±(node torsional moments, derived from node seismic forces along ZII XI axes, offset by the accidental eccentricity ±2eτxi.
(7)EY (seismic vertical component –seismic force along y direction- derived from dynamic analysis).

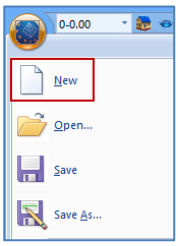
1.5 Notes

All the commands that were used in this example, as well as the rest of the commands, are explained in detail in the manual that accompanies the program.

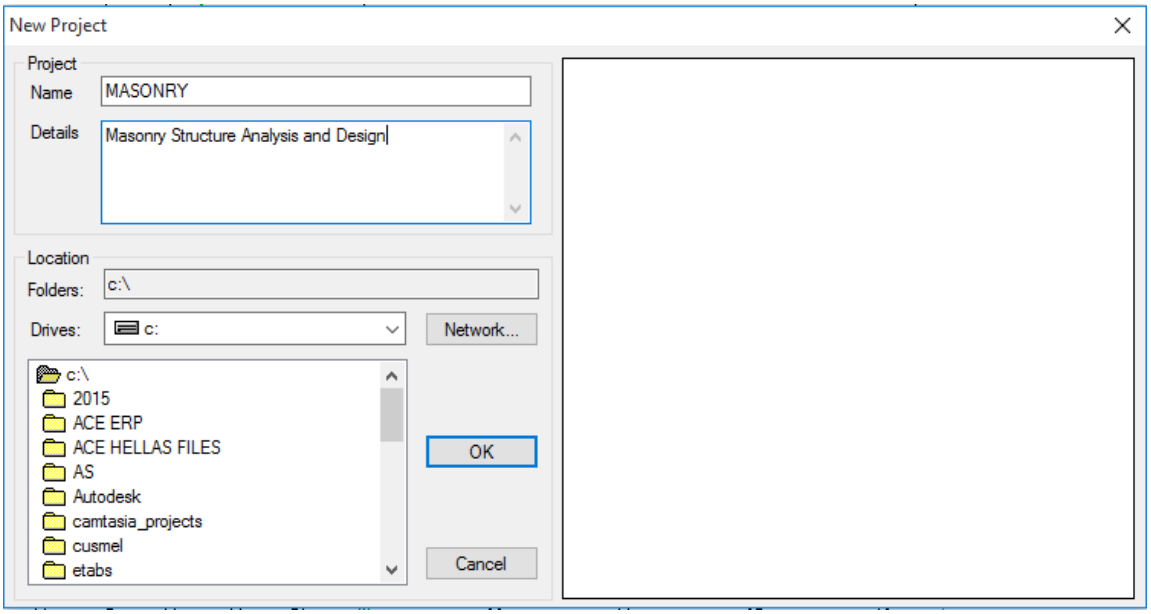
2. DATA INPUT - MODELING

SCADA Pro is enriched with a masonry library while the user can automatically create the masonry model using only the centered outline of the structure and modify each side through the Templates editor.

⚠ The **Templates** command can be used in two ways so that it fulfills every modeling demands.

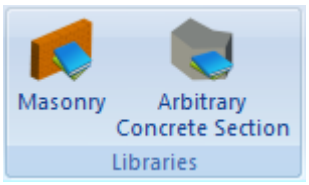
Use the  button located at the initialization window, or select ”New” 

from the menu, to create a new file. In the dialog box that appears define the data of the new project.

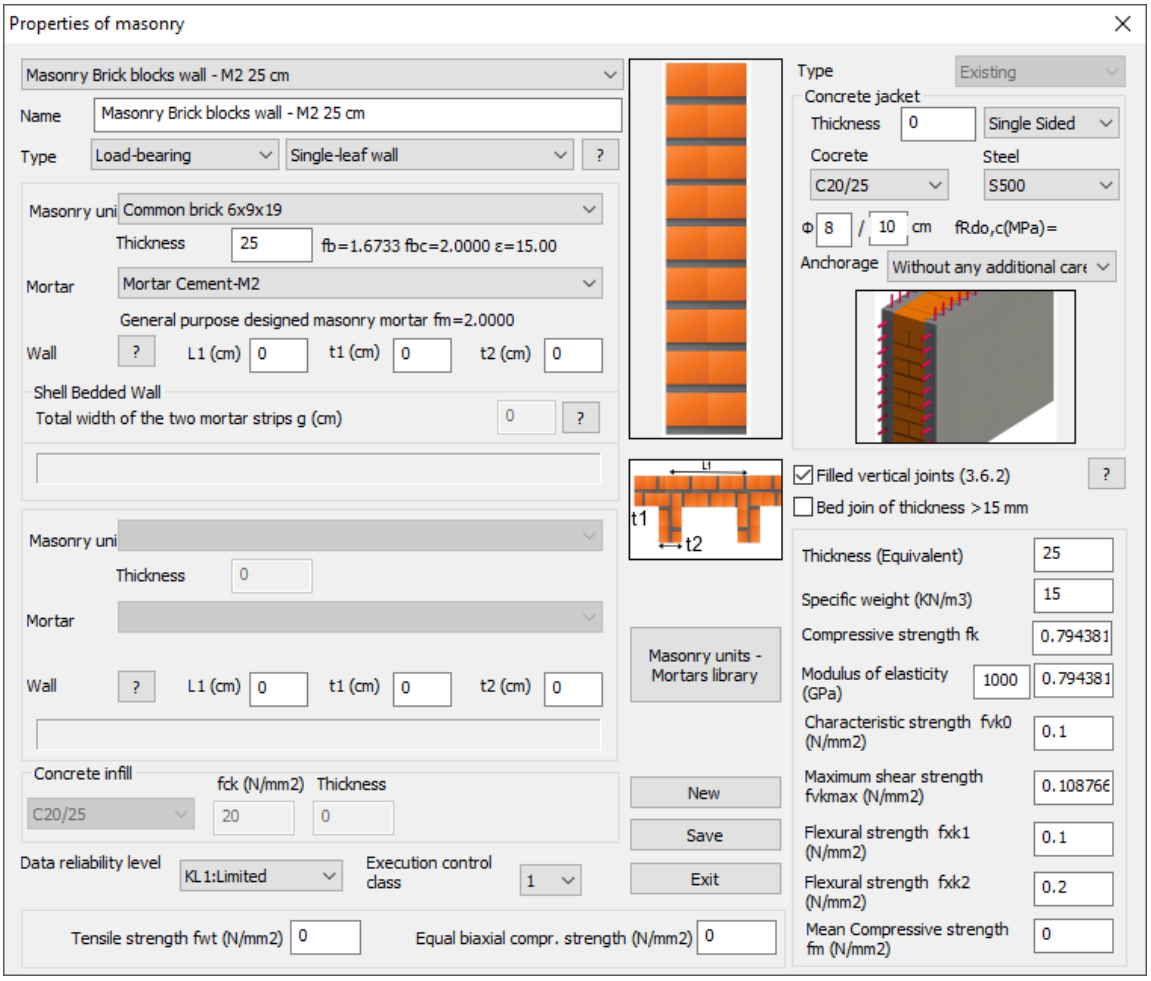


⚠ The name of the file can contain up to 8 characters of the Latin alphabet without any symbols (/, -, _) nor spaces. You can add a description or add some information related to the structure, in the “Info” field.

2.1 Masonry library – wall definition:



Inside the "Modeling" unit, in "Libraries" group, the "Masonry" command, opens the respective library:



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

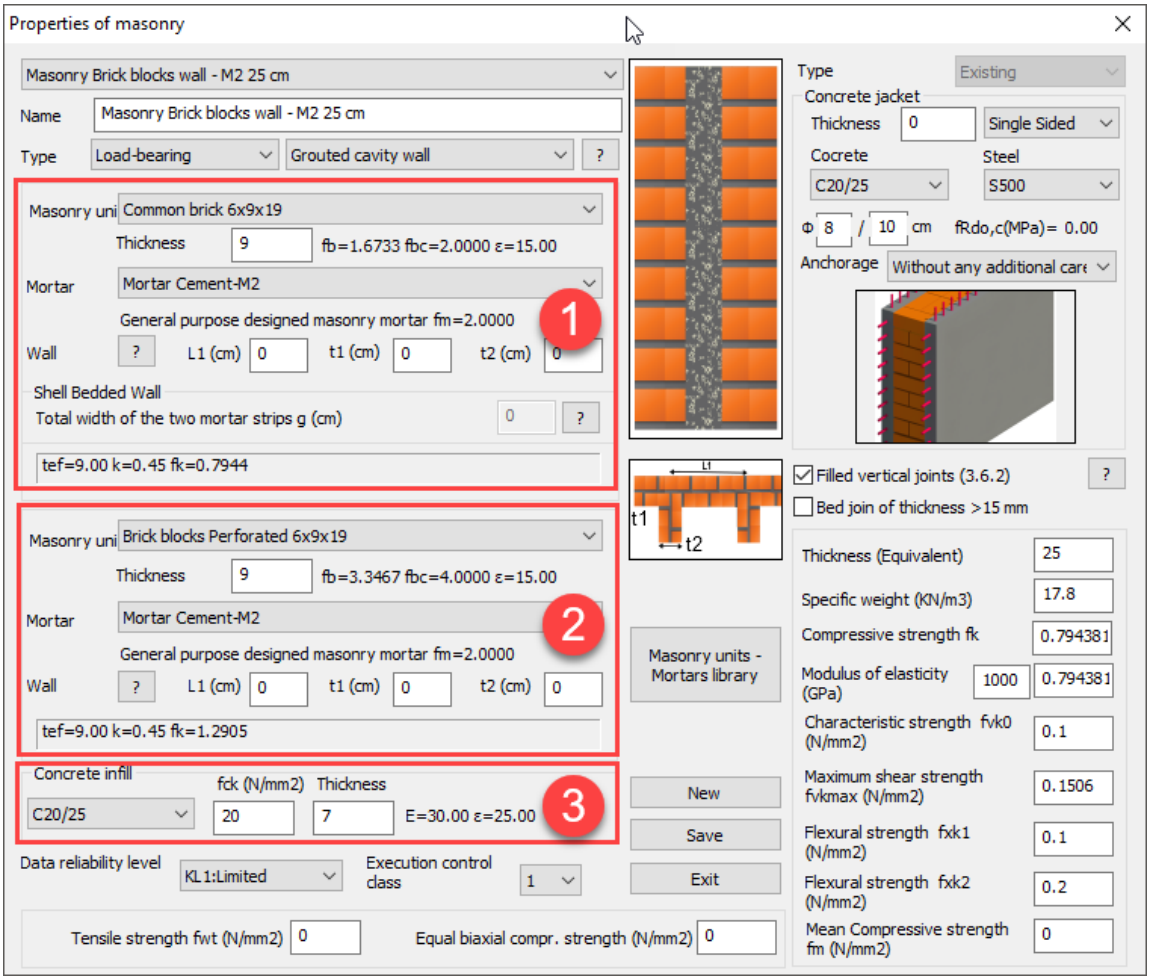
⚠ *Depending on the selected TYPE of masonry, in the dialog box, some fields are enabled or disabled.*

⚠ The definition for each type is displayed by clicking the ? button on the right.

EXAMPLE

Name: Wall 1
Type: Grouted Cavity Wall

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



In *Wall1* & *Wall2* define

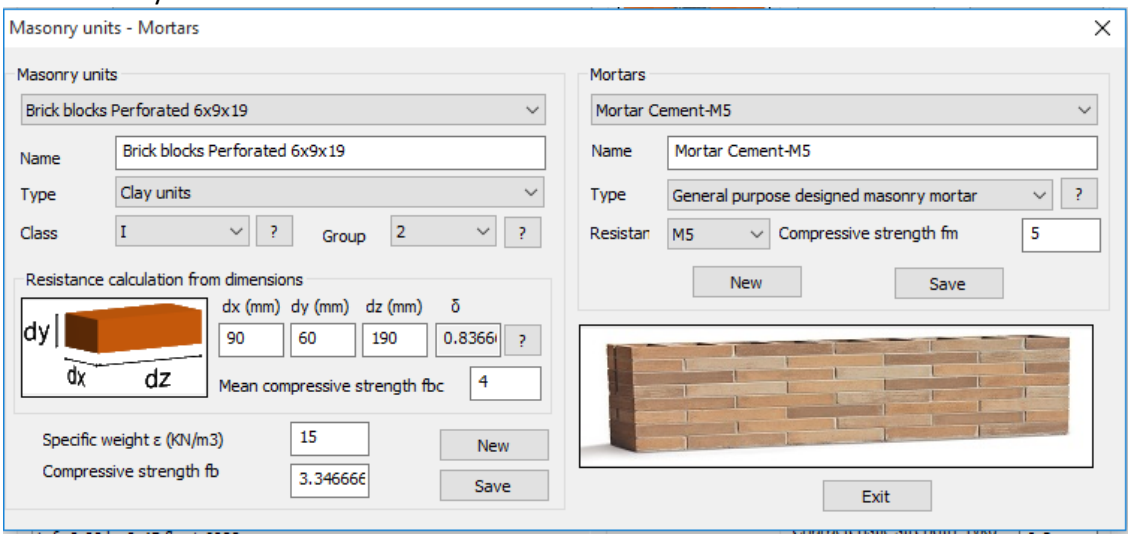
units: the type and thickness
Mortars: the type and the corresponding factors are updated automatically.

fb=3.3467 fbc=4.0000 ε=15.00

Masonry Units - Mortars Libray

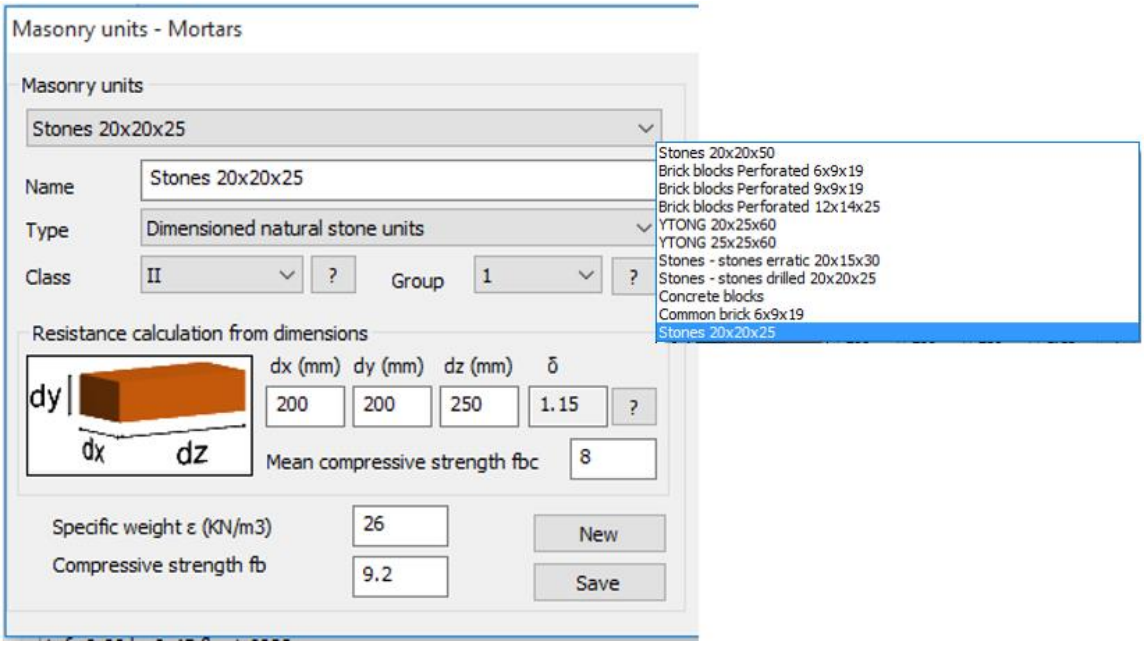
In the command "***Masonry Units – Mortars Library***" you will find standard typologies of clay bricks, mortar and masonry. You can enter other bricks and mortar, by simply typing the name and specifying the class and group, for the compressive strength (which is updated automatically). Then select the button "New".

You can also change the class and group of an existing masonry or mortar and update it by clicking "Submit".
In the field "Masonry Units", select from the drop-down lists the type of bricks and mortar, and create a new type of masonry by clicking "New". The weight and strength are calculated automatically.



For this example we chose:

1.1.1 Masonry units



Name: Dimensional natural stone units 20x20x25 (type in)
Type: Dimensional natural stone units (select from list)
Category: II, Group: 1 (select from list)

⚠ For further information regarding the Category and Group of your selection click the ? button on the right.

? **Masonry units may be Category I of II**

- category I

  Units with a declared compressive strength with a probability of failure to reach it not exceeding 5%. This may be determined via the mean or characteristic value

- category II

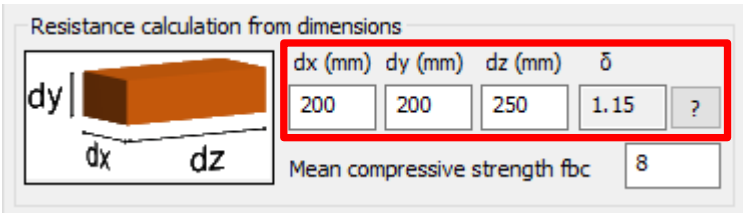
  not intended to comply with the level of confidence of Category 1 units (lower confidence level than for I)

	Materials and limits for Masonry Units							
	Group 1 (all materials)		Group 2		Group 3		Group 4	
		Units	Vertical holes				Horizontal holes	
Volume of all holes (%of the gross volume)	≤25	clay	>25;≤55		≥25;≤70		>25;≤70	
		calcium silicate	>25;≤55		not used		not used	
		concrete [b]	>25;≤60		>25;≤70		>25;≤50	
Volume of any hole (% of the gross volume)	≤ 12,5	clay	each of multiple holes ≤ 2 gripholes up to a total of 12,5		each of multiple holes ≤ 2 gripholes up to a total of 12,5		each of multiple holes ≤30	
		calcium silicate	each of multiple holes ≤ 15 gripholes up to a total of 30		not used		not used	
		concrete[b]	each of multiple holes ≤ 30 gripholes up to a total of 30		each of multiple holes ≤ 30 gripholes up to a total of 30		each of multiple holes ≤25	
Declared values of thickness of webs and shells (mm)	No requirement		web	shell	web	shell	web	shell
		clay	≥5	≥8	≥3	≥6	≥5	≥6
		calcium silicate	≥5	≥ 10	not used		not used	
		concrete [b]	≥ 15	≥ 18	≥ 15	≥ 15	≥20	≥20
Declared value of combined thickness [a] of webs and shells (% of the overall width)	No requirement	clay	≥ 16		≥ 12		≥ 12	
		calcium silicate	≥20		not used		not used	
		concrete [b]	≥ 18		≥ 15		≥45	

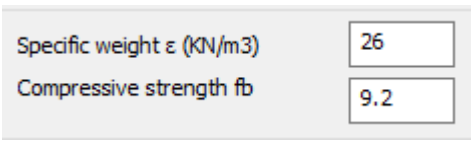
a. The combined thickness is the thickness of the webs and shells, measured horizontally in the relevant direction. The check is to be seen as a qualification test and need only be repeated in the case of principal changes to the design dimensions of units.

b. In the case of conical holes, or cellular holes, use the mean value of the thickness of the webs and the shells.

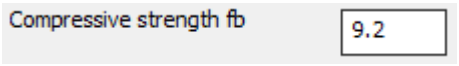
For the Strength Calculation from Dimensions, type the dimensions of the masonry unit and the reduction factor δ, is automatically filled.



Type the “Compressive Strength” fbc, which is the average value of experiments regarding the compressive strength of the masonry units and the “Specific Weight ε”.



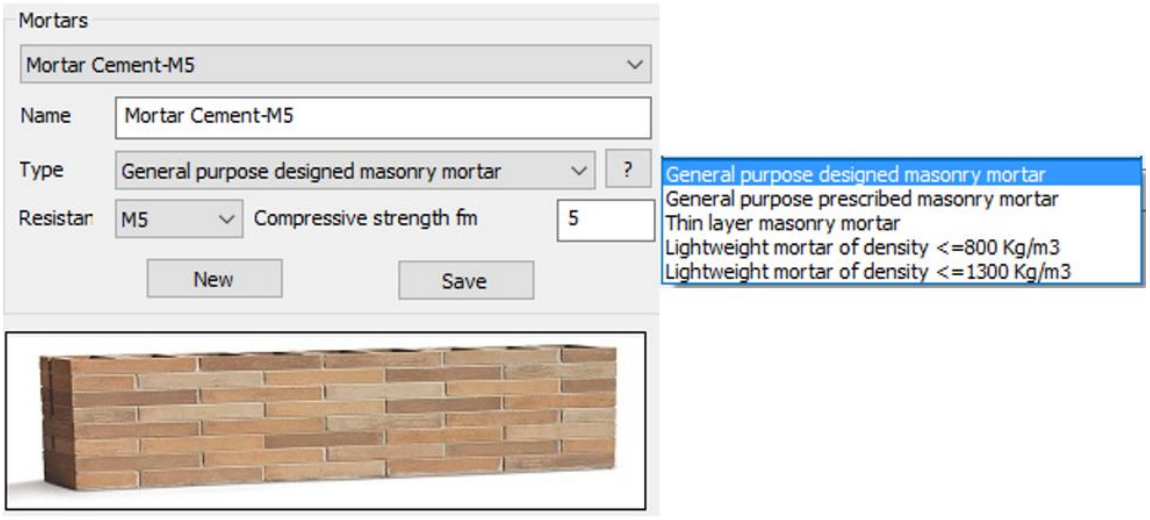
The program automatically calculates the "Compressive Strength" fb.



Select  to store in the masonry library this masonry unit.

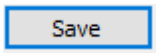
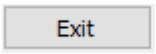
⚠ Every time that you save a masonry unit this is stored permanently and is available for the current and any future project as well.

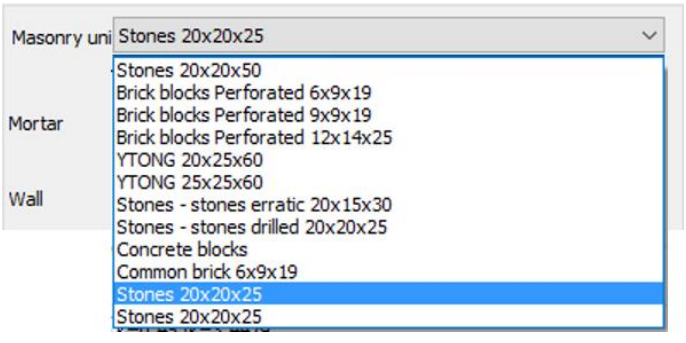
1.1.2 Mortar

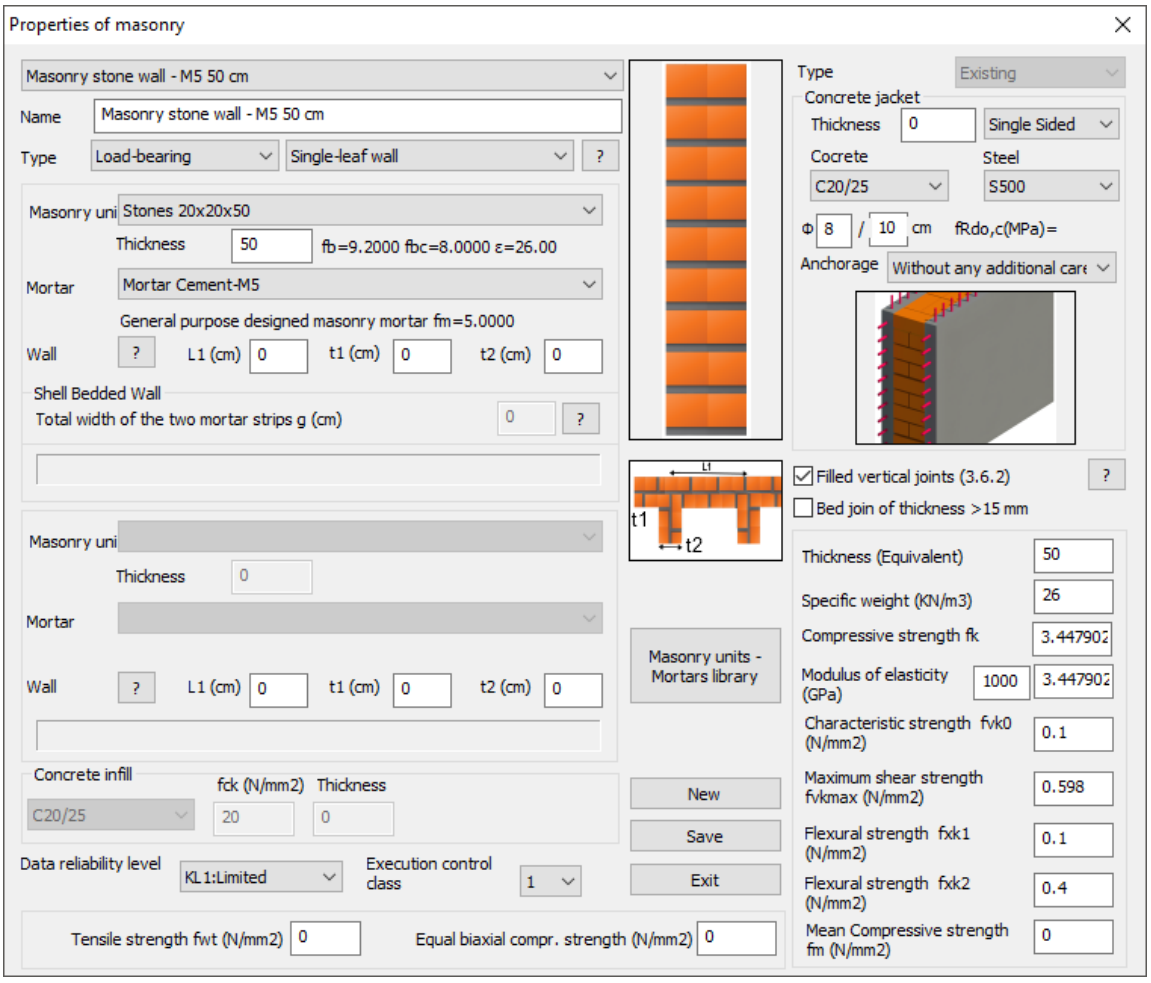


Name: Mortar-M5(select from list)
Type: General Purpose Mortar (select from list)
Strength: M5 (select from list)

The compressive strength Fm is automatically filled in by the program

Select  and  to return to the masonry library, where you can select the new masonry unit, which is now located in the list.





Name: Wall M5 0.50 (type)
Type: Single-leaf (select from list)

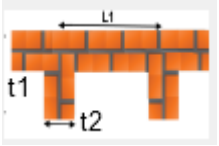
Masonry unit: Dimensional natural stone units (previously defined) and
Width: 50 cm (type)

⚠ The total masonry results are calculated by the program based on the input data and they are transferred to the summary table on the right. fb=9.2000 fbc=8.0000 ε=26.00

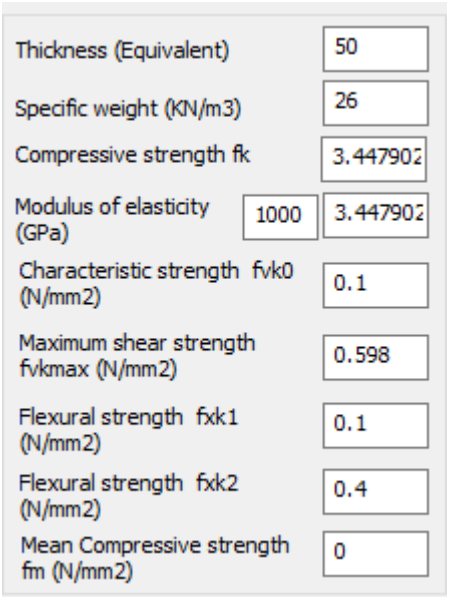
Mortar: Mortar-M5

⚠ Prescribed Masonry Unit fm is automatically updated.
General purpose designed masonry mortar fm=5.0000

For this example, all the masonry data are defined. Click Save to update the library and add the defined masonry to the list.

In case you had selected **Cavity Wall**, the second field regarding the masonry units and mortar for the second part of the wall will be enabled for editing as you did for the first wall. For **Shell Bedded Wall**, the field regarding the total width of the two mortar strips g will be enabled (see 3.6.1.4 for the calculation of the Specific Strength ). For struts, type the dimensions according to the image 

to calculate the active thickness according to equation 5.10 (see. 5.5.1.3)



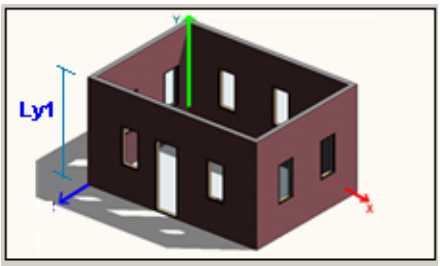
The total masonry results are calculated by the program based on the input data and they are transferred to the summary table. If the user knows the values of the equivalent wall, these can be defined manually.

2.2 Modeling:

2.2.1 Templates:

1st MODE: The Templates tool, includes a standard masonry structure, which can be modified accordingly, so that it can match the demands of a simple project.

Select the insertion point and choose from the drop-down list “Masonry” 



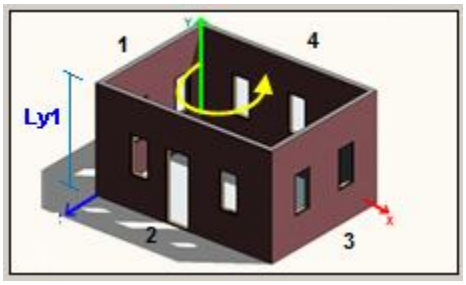
Geometry	
Number of front views	4
Along y	1
Distance y	300,00
Witdh (cm)	30,00
Thickness (cm)	20,00
Position Angle	0,00
Distance along y	
Ly1 (cm)	300,00
Front Views	
Break	No
Front View 1	
Start x (cm)	0,00
Start y (cm)	0,00
Length(cm)	400,00
Angle	-90,00
Witdh (cm)	30,00
Thickness (cm)	20,00
Opening	2
Opening 1	
Start x (cm)	50,00
Start y (cm)	100,00
Width(cm)	100,00
Height(cm)	100,00
Opening 2	
Start x (cm)	250,00
Start y (cm)	100,00
Width(cm)	100,00

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

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

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

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



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

2.2.2 Front View Identification:

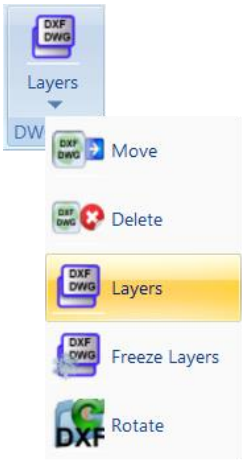
2nd MODE: SCADA Pro gives you the possibility to create a masonry structure on any external boundary, by using the tool “Templates”, quickly and easily.

The process is the following:

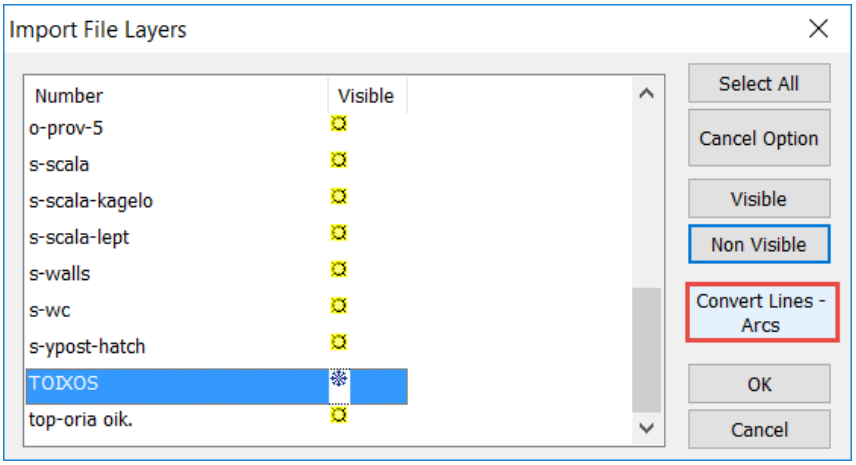
1. Enter a plan view in DXF or DWG file format by using the 

command

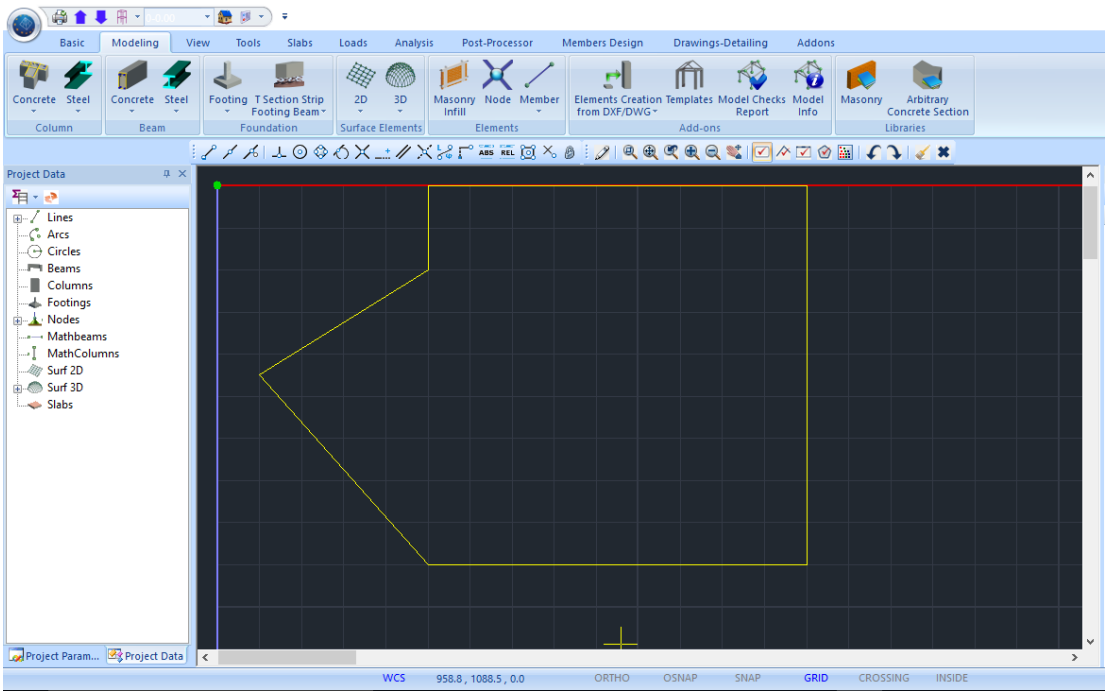
2. Use the command “Layers” to open the list of the design layers.



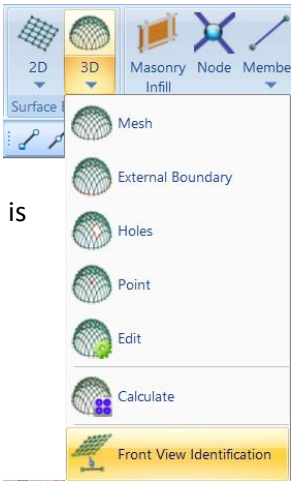
3. Select from the list the layer containing the walls and click on “Convert Lines, Arcs”.



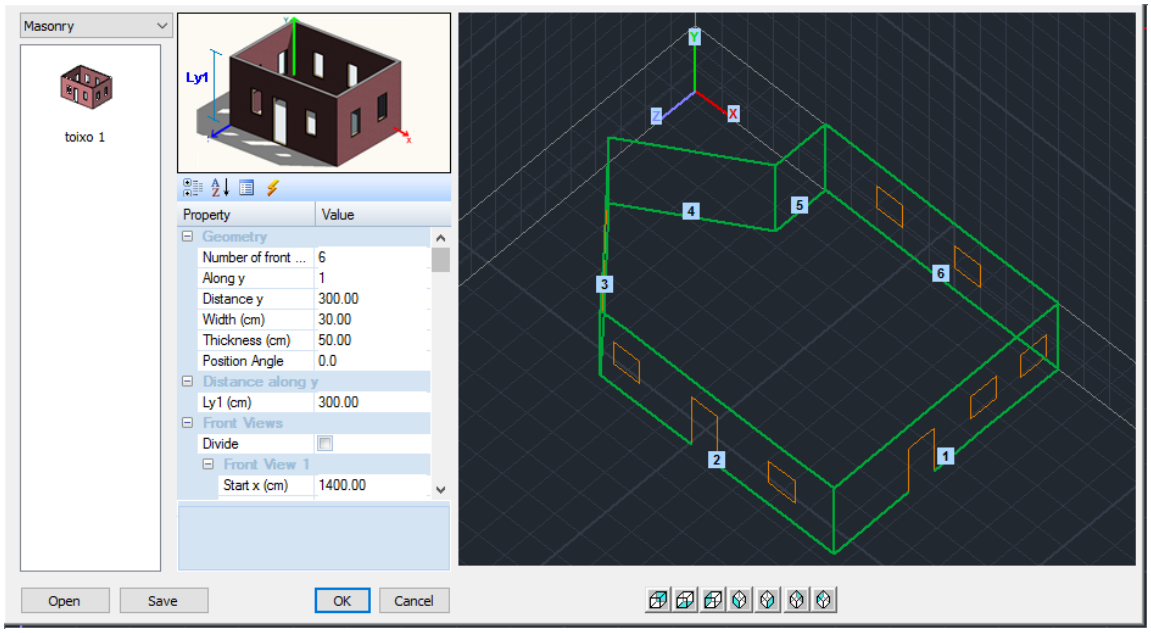
⚠ In case that you do not have a .dxf or .dwg file, you can design the plan level directly to the XZ level of the SCADA environment.



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

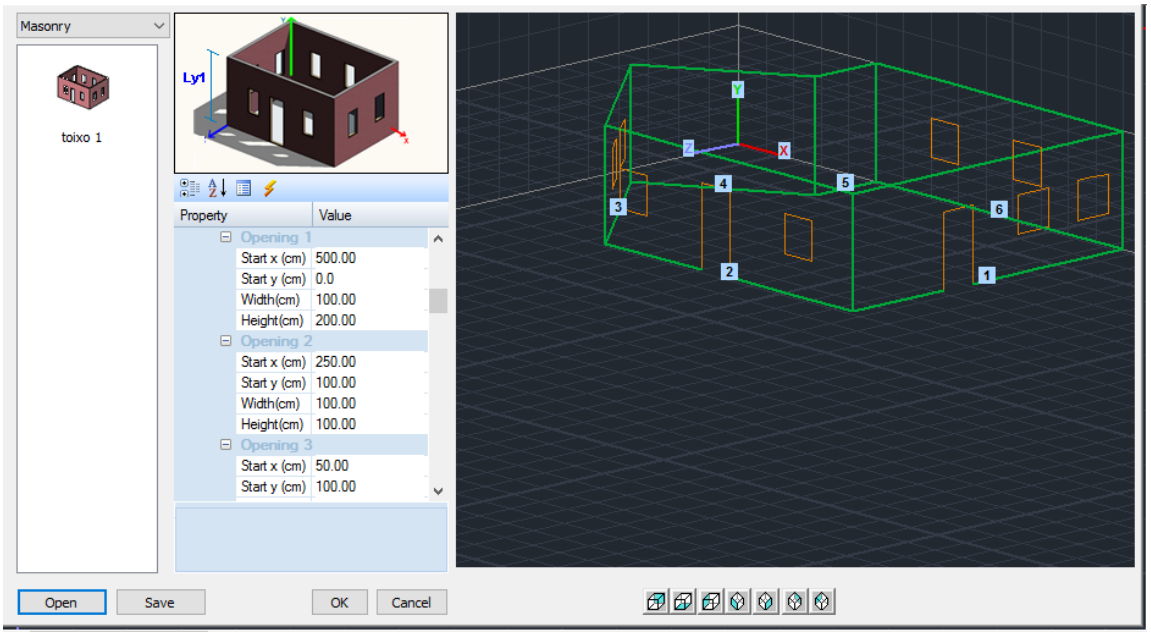


Then use the selection command "Window"  to select the total plan view. Right click and the masonry templates dialog box is displayed:



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

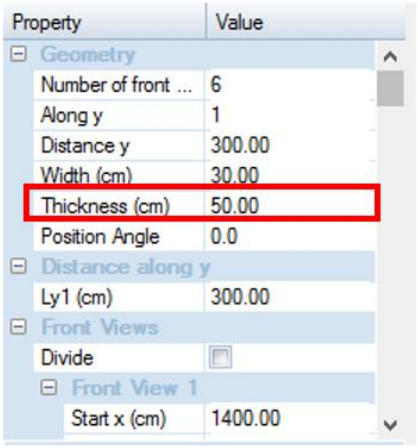
5. The user has to define the number of the floors and the corresponding heights, as well as the openings on each view by following the 1st MODE procedure.

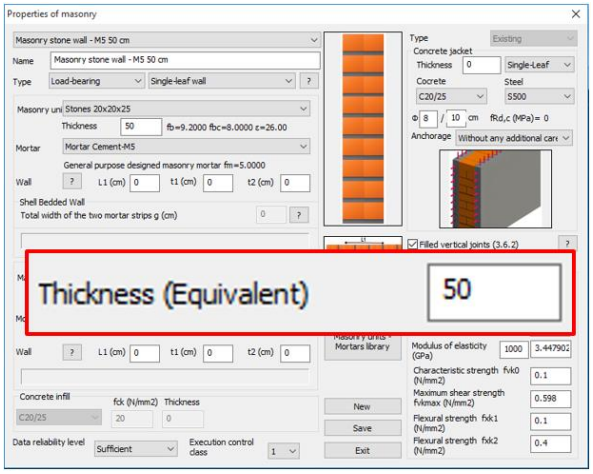


Since you have completed the process for each side and each opening, insert the project on the desktop by selecting the button “OK”.

⚠ You can save the formed model as an .stp file, by clicking the Save button, creating in this way your very own template library. Click Open to call a saved file and load the model at any point.

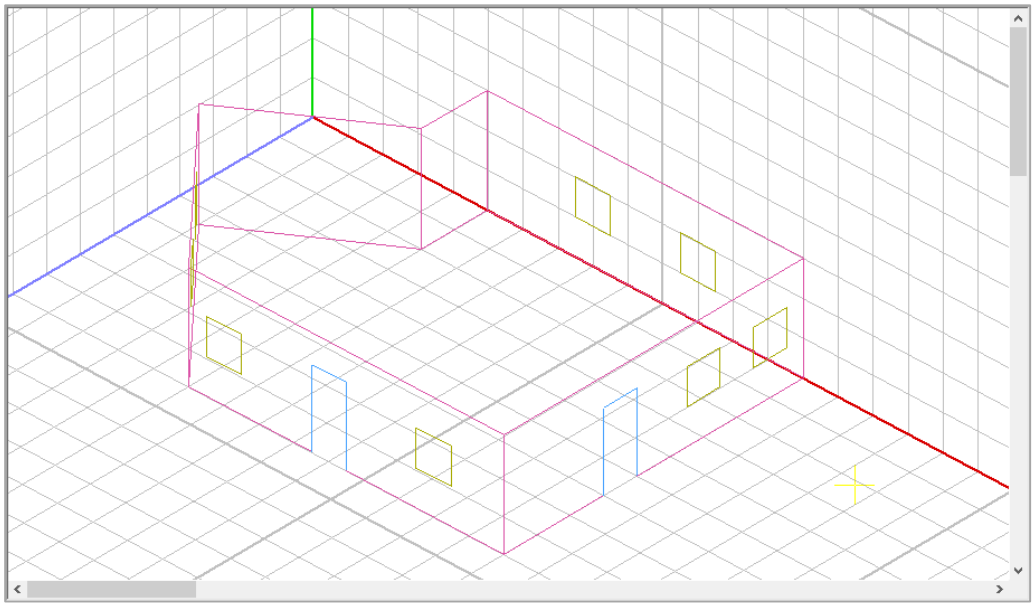
⚠ WARNING: Make sure that the Equivalent Thickness of the wall defined to the library has the same value as the Thickness defined in the Templates.





⚠ WARNING: In the templates field you can define a single value of thickness for all walls. To edit the thickness of some walls, you open the “Plate Elements Creation” form and you modify the values respectively.

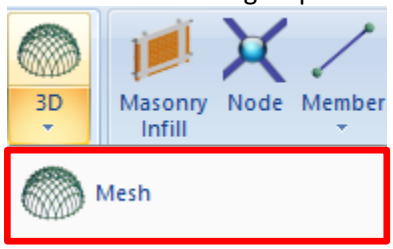
As soon as you have completed the process for each side and each opening, insert the project on the desktop by selecting the button “OK”.



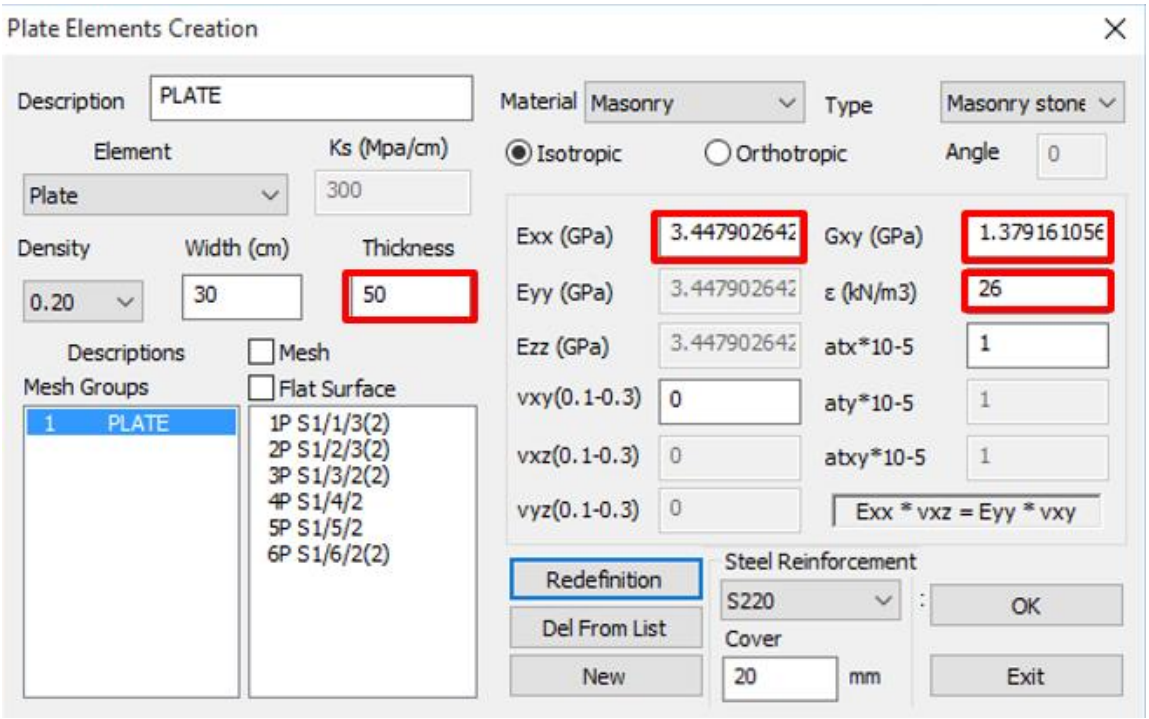
Inside SCADA environment, you can see the outlines for each view and its openings in 3D presentation.

2.3 Mesh Group Definition:

As soon as the model is imported in SCADA environment, select the 3D "Mesh" command inside "Surface Elements" group.

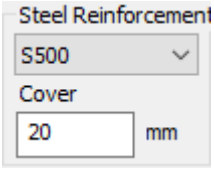


In the dialog window that opens, the Mesh Groups list, contains the 1 PLATE mesh, with its corresponding surfaces (one for each view). By selecting the 1 PLATE the fields regarding the Density, Width, Thickness etc (previously defined at Templates) values, are automatically filled in.



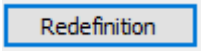
In the type, select from the list the previously defined wall from the library, and the respective fields Exx, Gxy and special weight ε are automatically updated.

Steel reinforcement and Cover



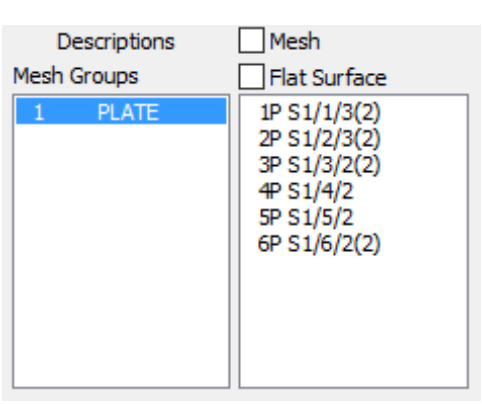
It is the field where you select the quality of the **steel reinforcement** for the surface finite elements and the mm of the **cover**:

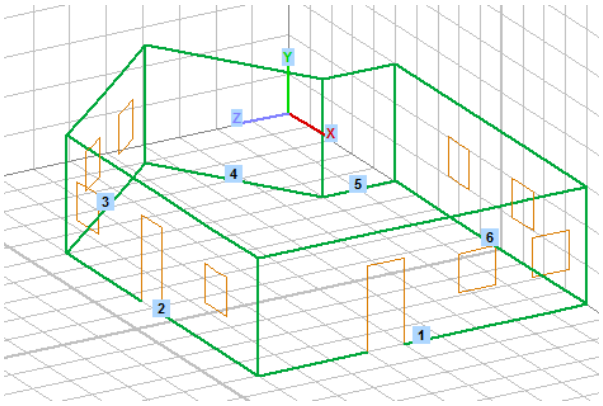
Now it's also possible to define different cover for each mesh of the Mesh Group.

Click the 

button to update the mesh and store any modifications.

2.3.1 Mesh sub-Group Definition:





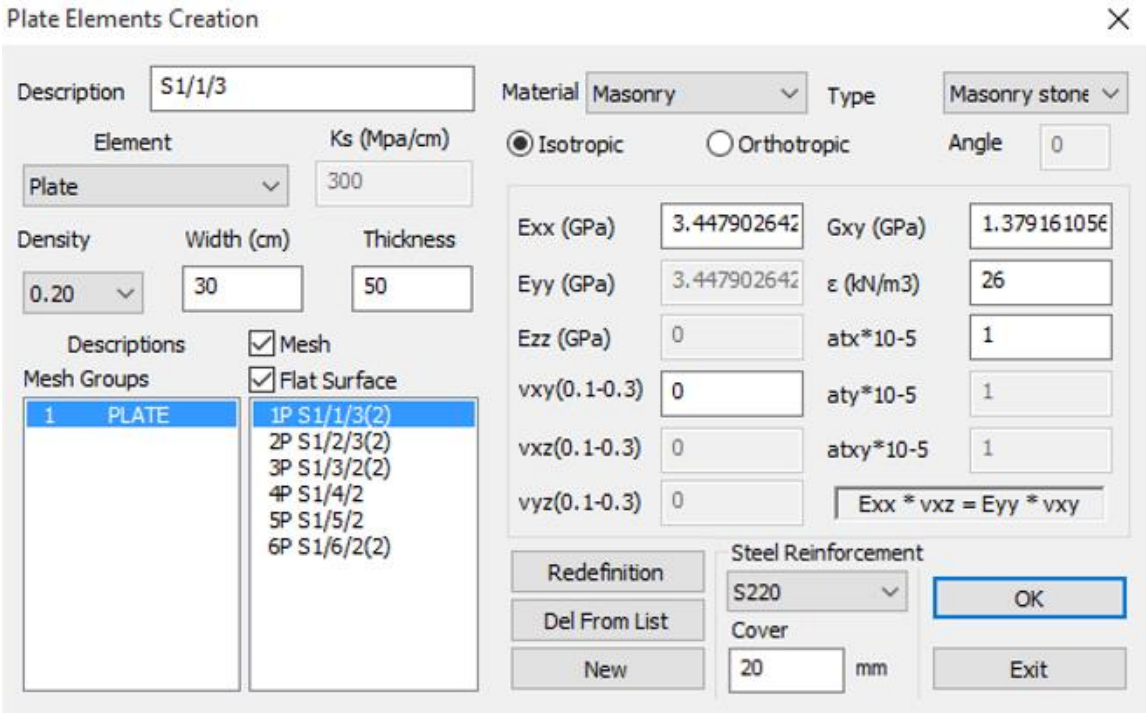
The derived from templates mesh model comes along with the Mesh group (1 PLATE) and a surface for each view.

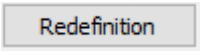
In the Surface name 1P S1/1/3(2) :

- The first number is the number of the view,
- The P letter stands for flatness
- The number inside the parenthesis, defines the number of holes in the respective view.

Activate the 

and select a surface. The fields are updated accordingly by the defined values of the selected surface,

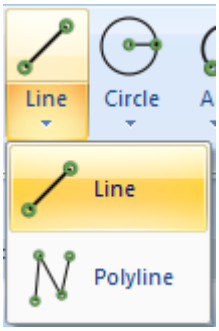


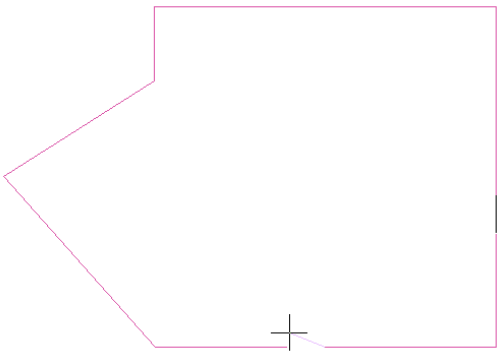
Enabling in this way the modification of any parameter (name, density, width, thickness type etc.) regarding the selected surface. Finally, click 

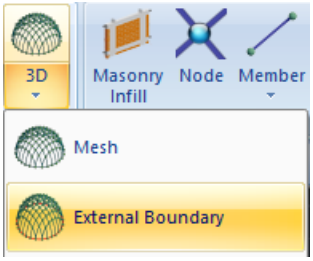
to apply the modifications.

2.3.2 Raft and mesh areas external boundary definition:

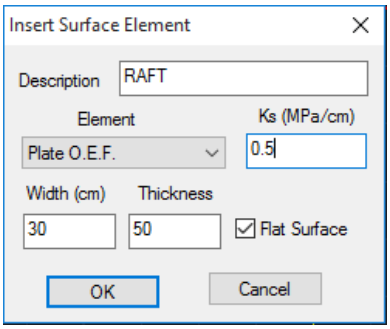
From the command group "Basic" select “Line” to draw the closed contour of the arbitrary cross section. Use snap tools for help.







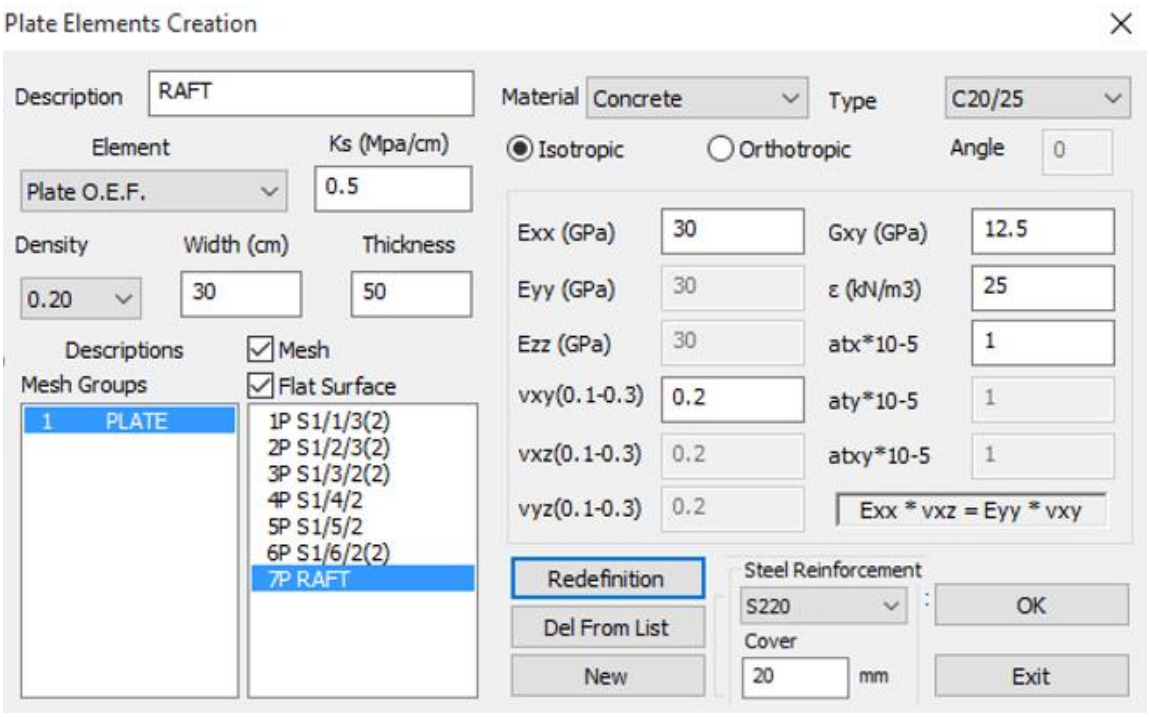
Then select “3D”>> “External Boundary” and left click to select the lines of the first boundary and right click to complete. The characteristics of the first mesh subgroup are displayed in the dialog box. The active “Flat Surface” command means that the surface belongs to the level.



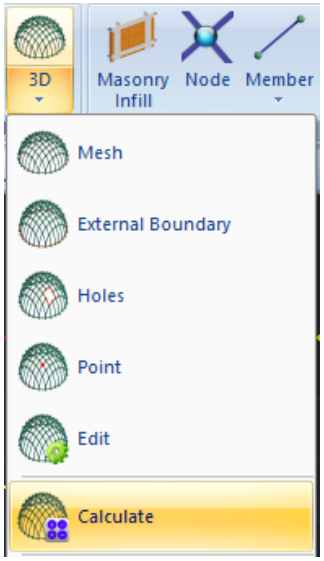
Define the parameters of the raft surface:
-type a name in the Description (RAFT)
-select “Plate O.E.F” from the list (plate on elastic foundation)
-type a value for the spring constant Ks (Ks=0.5 Mpa/cm)
-define, Width and Thickness (30, 50)
-click OK.

Return to 

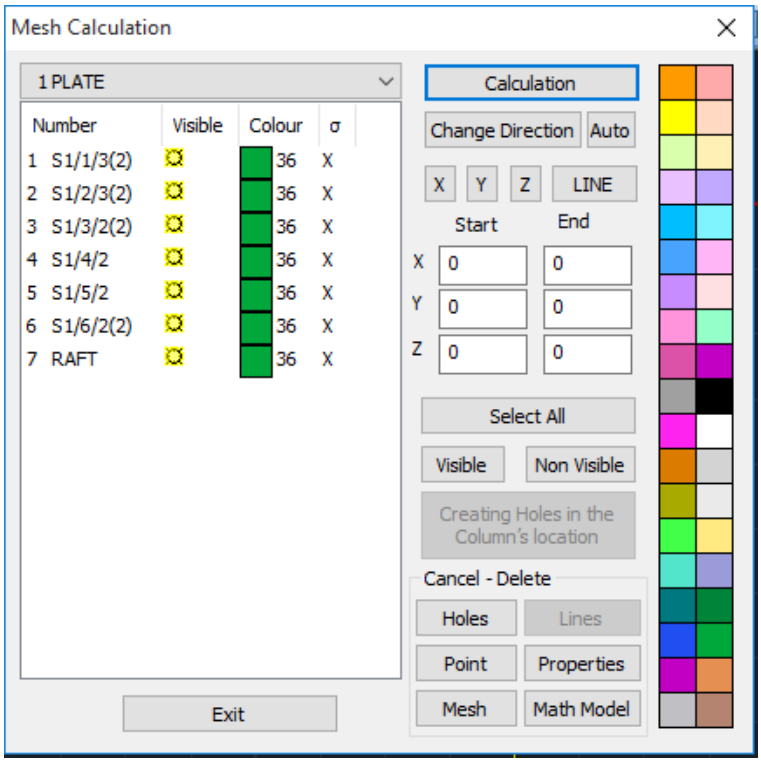
command to see the surfaces of the “RAFT” mesh group.



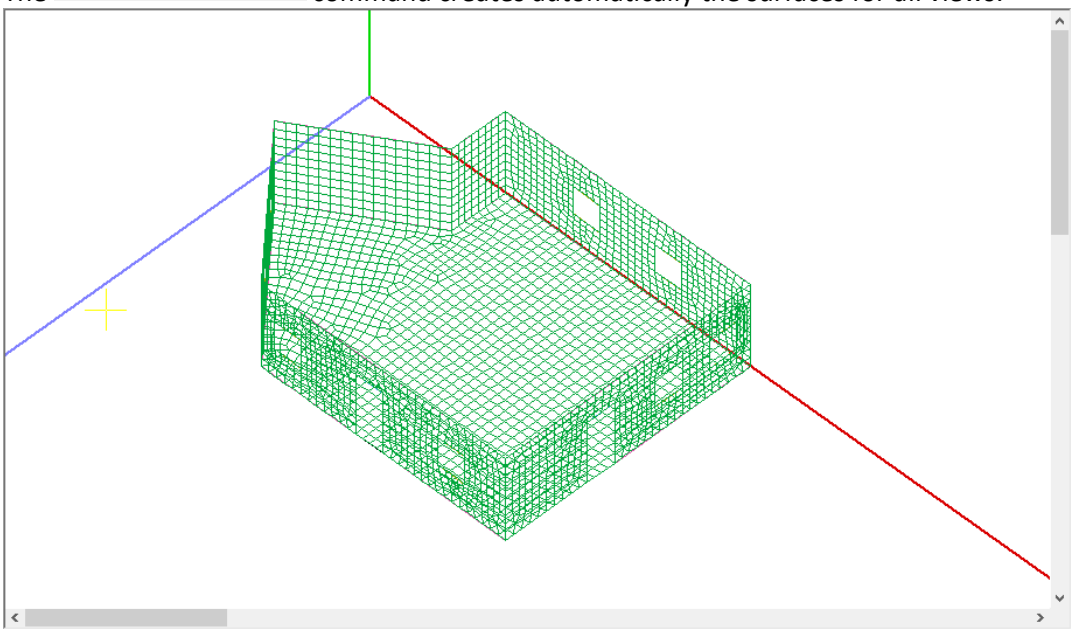
2.3.3 Surface Calculation:



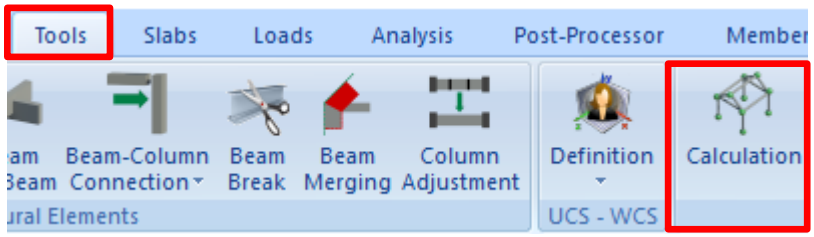
Select the Calculation command. In the dialog box that opens, the mesh list contains the 1PLATE group and its respective surfaces.



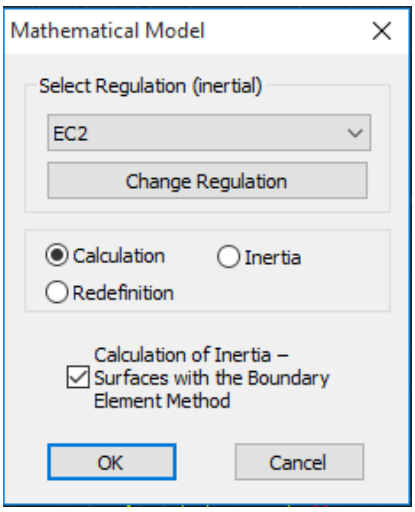
The Calculation command creates automatically the surfaces for all views.

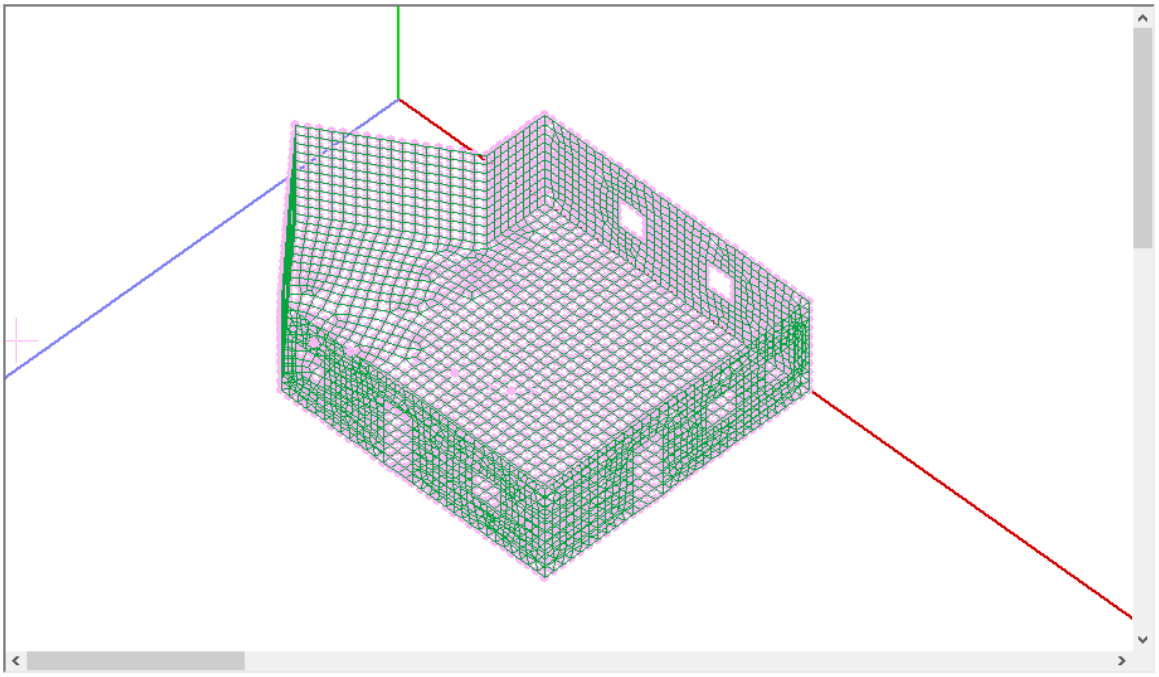


2.4 Mathematical Model calculation:



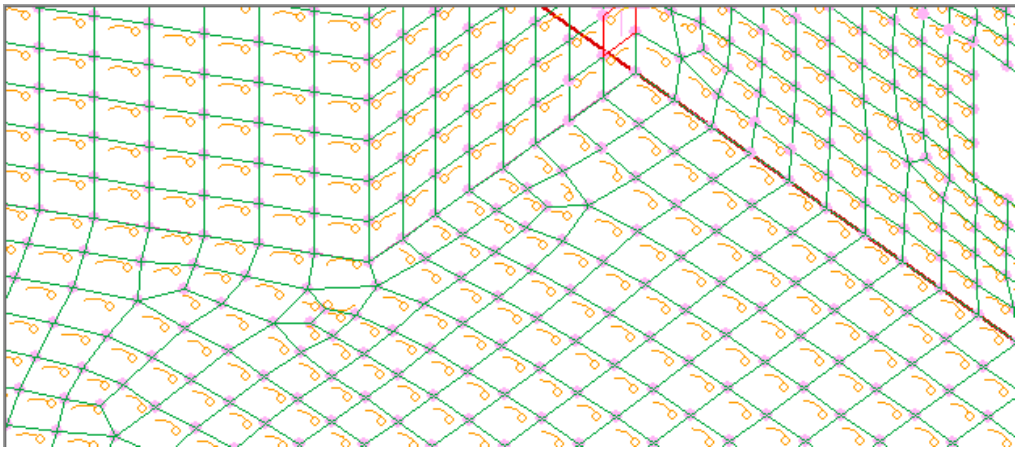
To create the mathematical model of the structure, from “Tools” unit select “Calculation” and click OK on the dialog window that opens:



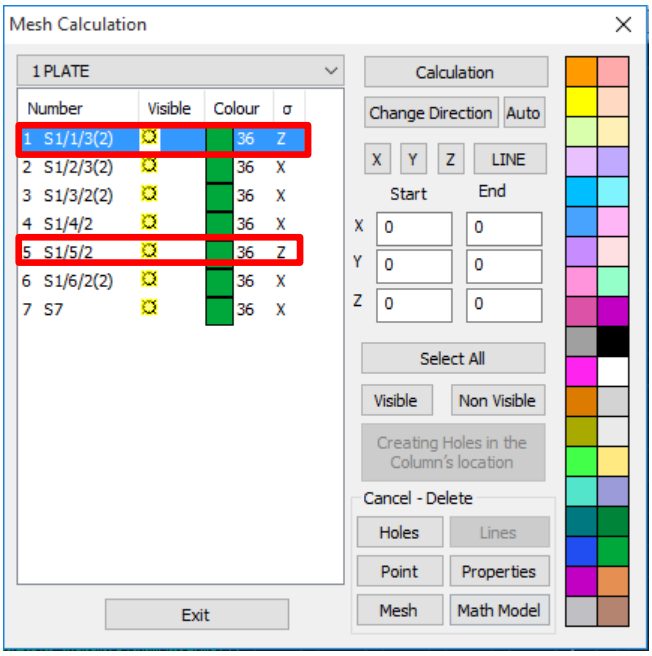


As soon as the mathematical model is created, the local axes and their direction (in respect with the global axes) must be redefined.

1. Inside the View unit, in Switches command, activate the Local Axes option.
2. Return to the "3D Mesh >> Calculation" command, and in the dialog form, select all the surfaces through the Select All command and click Auto to adjust local axes of all the surface finite elements of the plate to have the same direction.

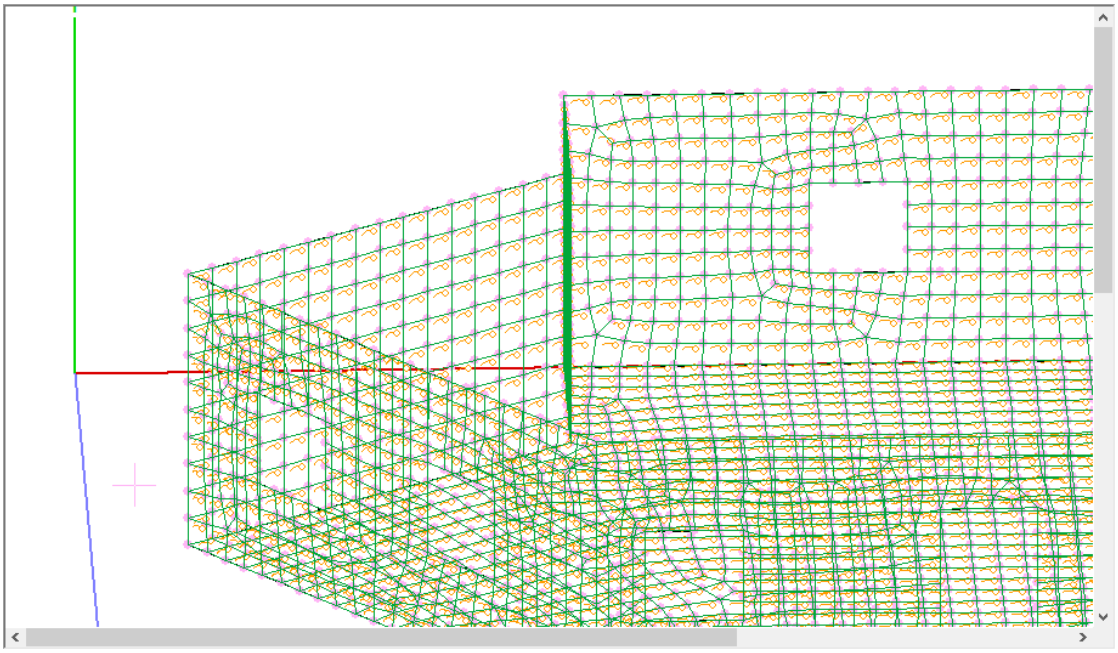


3. Finally, for views of which the local axis is <u>parallel to the global axes X or Z</u>, select them and click X or Z respectively, to define the main direction of the steel reinforcement (direction X or Z). For surfaces that run along X direction (vertical to Z axis) click X, while for surfaces that run along Z direction (vertical to X axis) click Z.

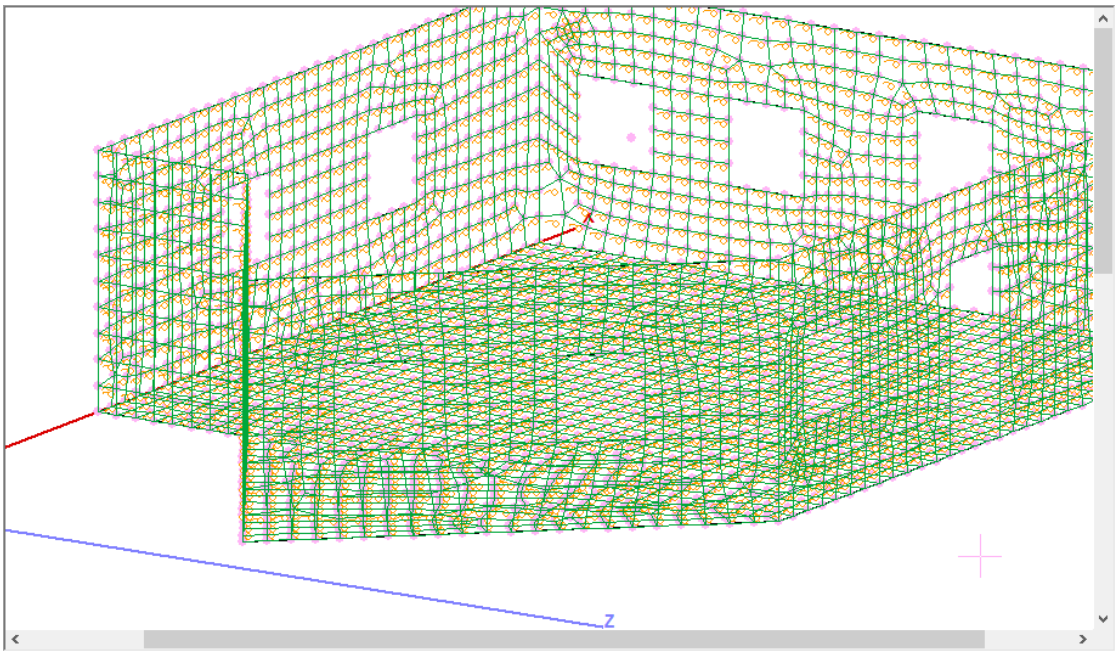


⚠ For views that are not parallel or perpendicular to the global axes, the main reinforcement direction is automatically defined.

In this example, for views 2,3,4,6,7 the x local axis is parallel to global X



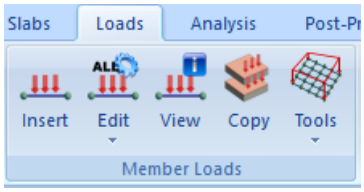
While for views 1 and five the x local axis is parallel to global X.



4. Click Exit to apply the modifications and close the window.

3. LOADS DEFINITION

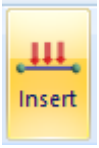
3.1 Manually imported:



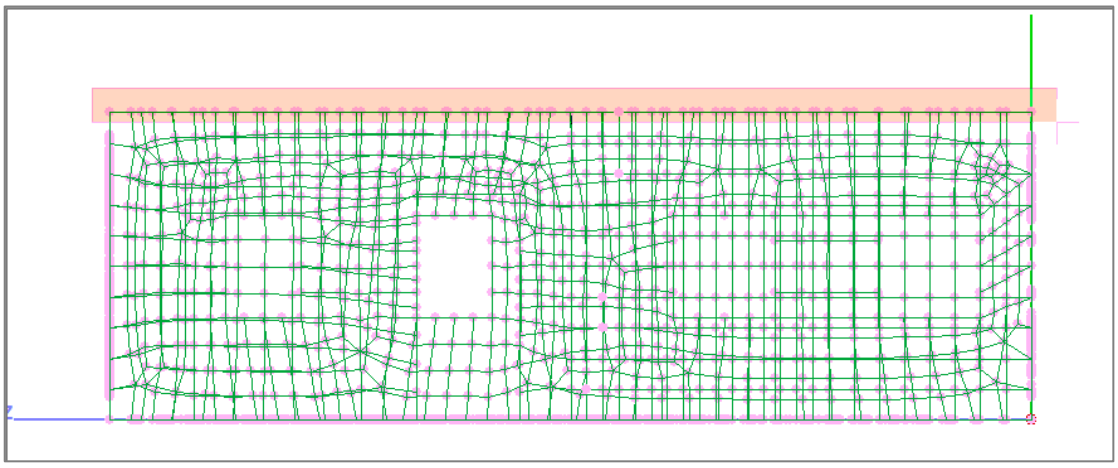
"Member Loads" commands' group contains the commands for insert, edit, view and copy the loads of members, nodes and surfaces finite elements.

For this example, to apply the loads regarding the slab that shelters the structure, to the top nodes of the perimeter, follow the procedure described next:

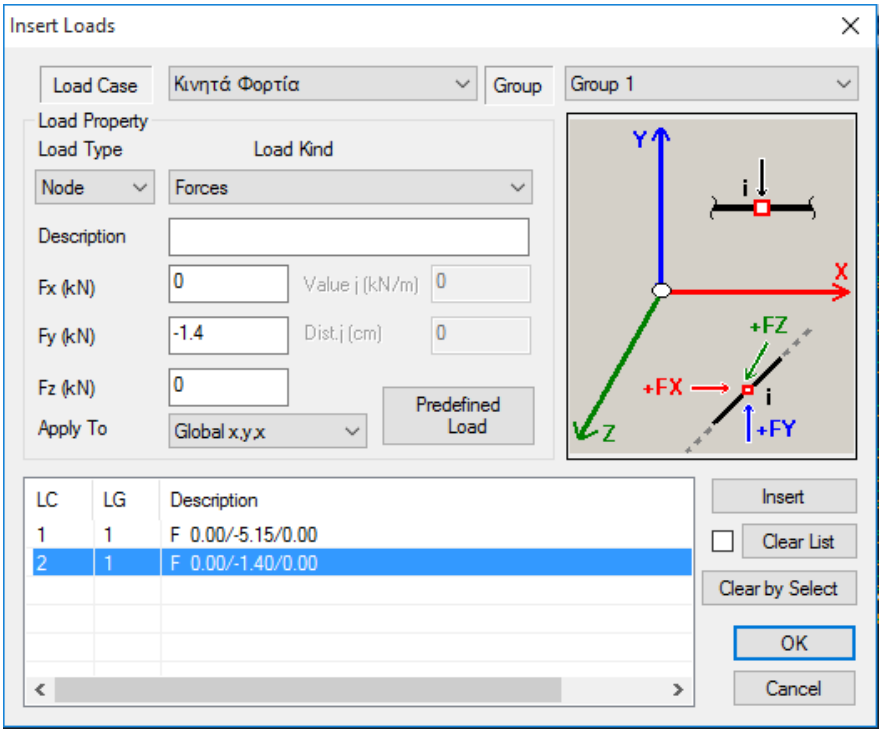
- Initially, we calculate the dead and live loads derived from the slab:
  Slab area 95 m2 x Slab thickness 0,2 m =19 m3 / Concrete 25 KN/m3
  19 m3 x 25 KN/m3=475KN
  Slab perimeter 40m node for each 0,3m = 133 nodes
  475/133=3,75 KN/node
  Extra dead load derived from coating 2KN/m2
  2 KN/m2 x 95 m2=190 KN
  190/133=1,40 KN/node
  TOTAL DEAD LOADS 5,15 KN/node
  Live 2KN/m2
  TOTAL LIVE LOADS 1,40 KN/node
- We rotate the model using the 

command ("View" unit>> "Views")
- Select the 

command
- Activate the  selection, to select all the nodes of the upper level



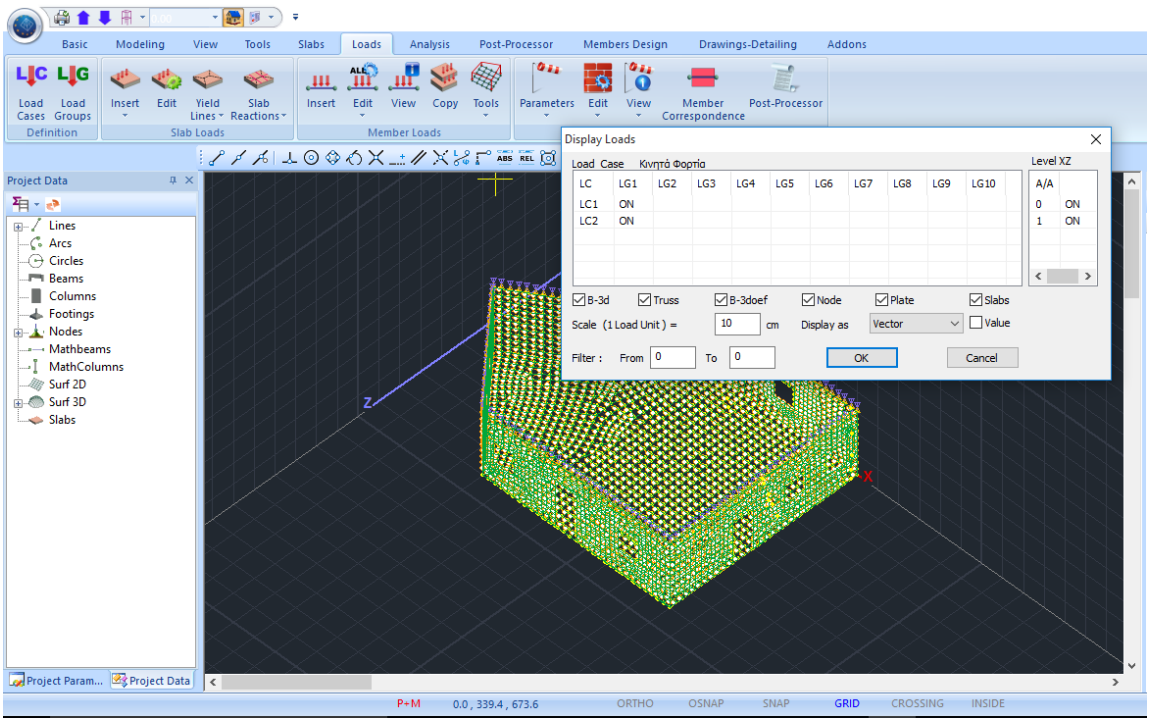
- Right click and in the dialog form:
  Select: Dead - Nodes, Forces,
  Type: 5,15 KN
  Click: Insert
  then
  Select: Live - Node, Forces,
  Type: 1,40 KN
  Click: Insert



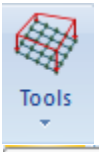
Click: OK to apply the defined loads

- Select 

to view the loads:



3.2 Load Destribution on the Surface



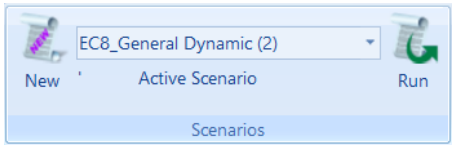
The new version of SCADA Pro comes with a new tool for the automatic distribution and application of loads on mesh areas.

Analytical description on how to use this command can be found in chapter 6 “LOADS” on page 21.

4. ANALYSIS

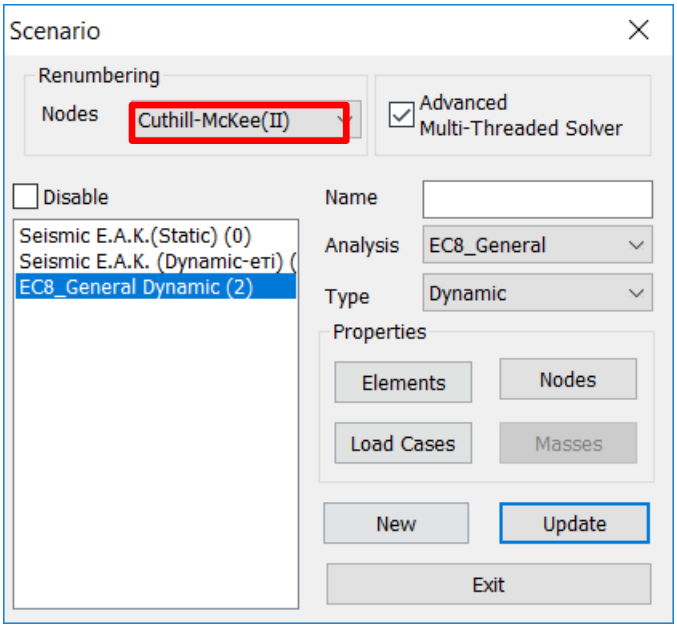
4.1 Masonry structure analysis by Eurocode:

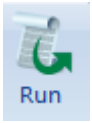
As soon as you complete the modeling and load definition processes, move on to analysis. For masonry structures analysis, create an Eurocode analysis scenario, so that SCADA Pro will perform the analysis by the provisions of the Eurocodes.



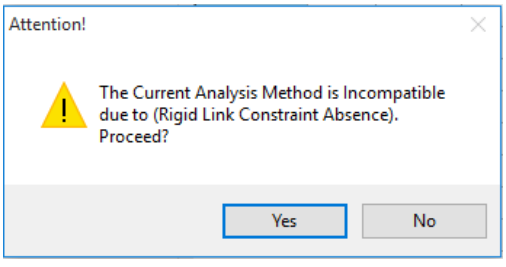
Move to “Analysis” unit and from the “Scenarios” command group, click “New” to create an Eurocode scenario for masonry structures analysis.

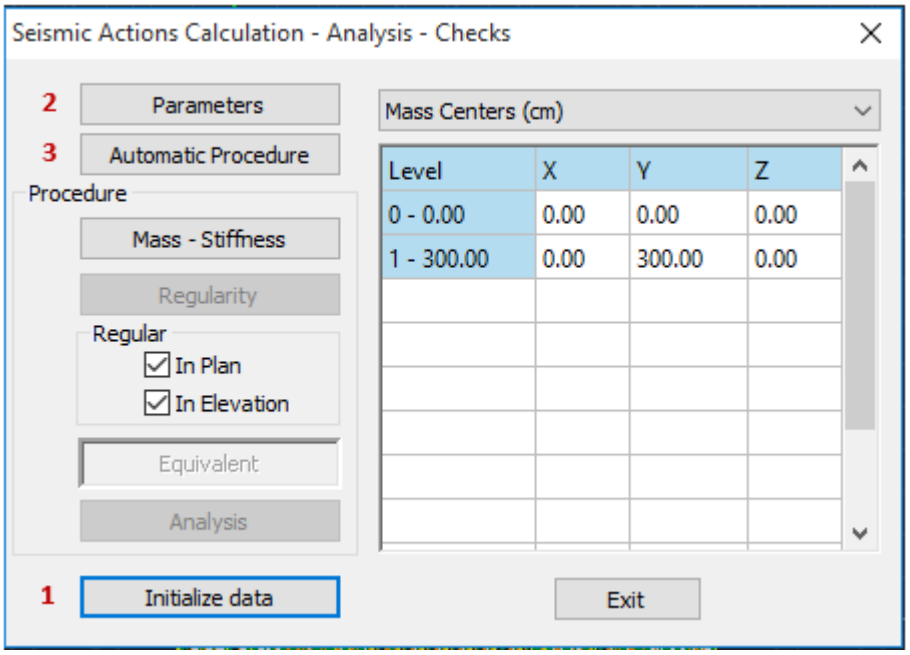
Click “New” and in the dialog window that opens:
-select Nodes Renumbering according to Cuthill-McKee(II) method
-create a new scenario by selecting the type and the respective annex -> EC-8_ Dynamic

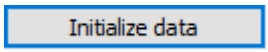


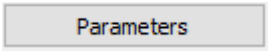
-select from the list the Eurocode scenario and click 

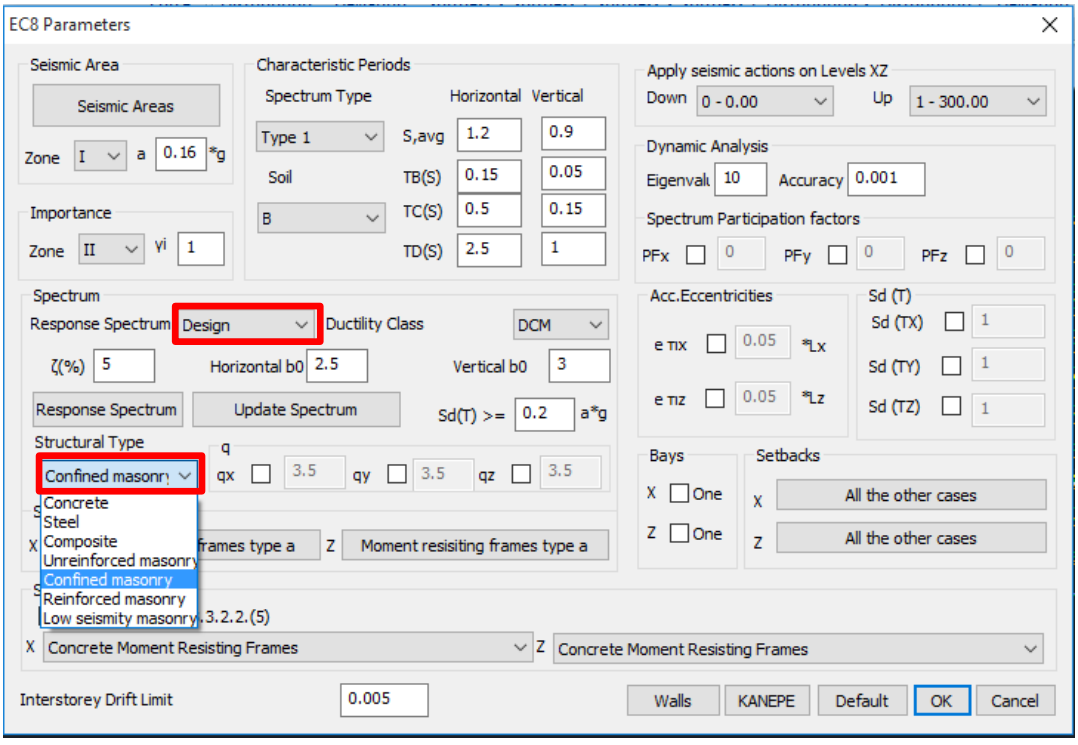
.
In the dialog box that opens, accept the warning regarding the diaphragm absence and click:





One  to update the parameters of the current scenario

Two  to define the analysis parameters



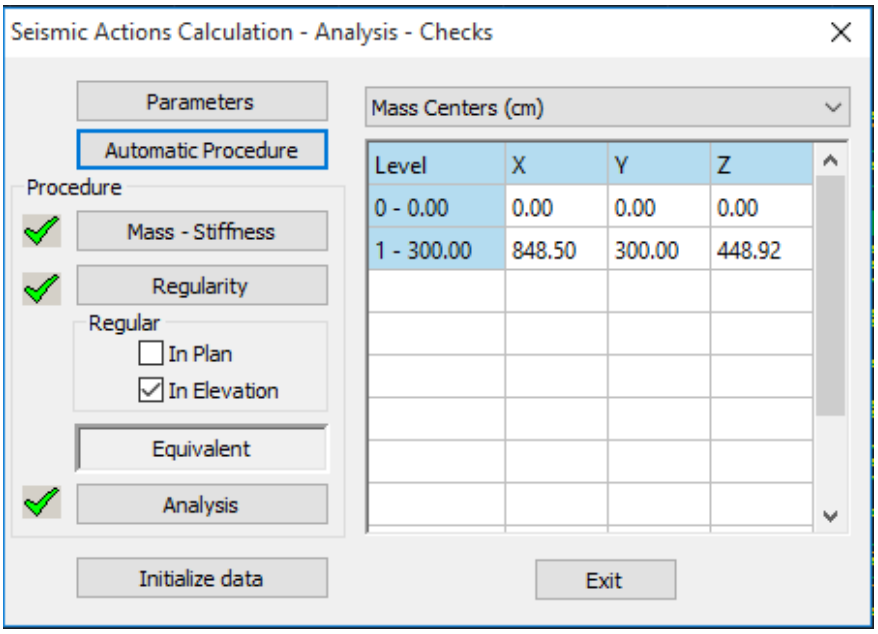
-Define “Zone”, “Importance” and “Soil”.
-select “Design” spectrum and

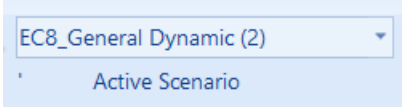
-at the type of structure field select “Confined Masonry”
-Click OK to update the parameters and close the window.

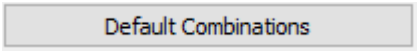
Three 

to run the analysis.

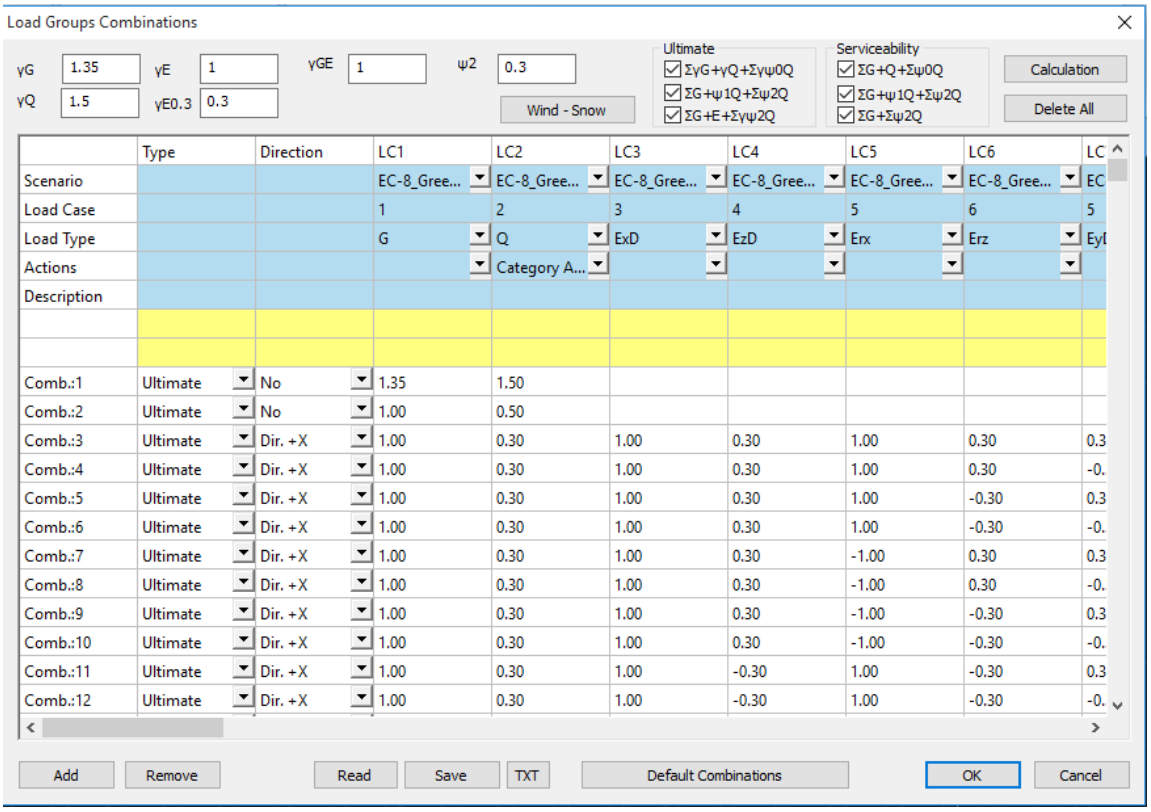
Let the program to complete the process and click Exit.

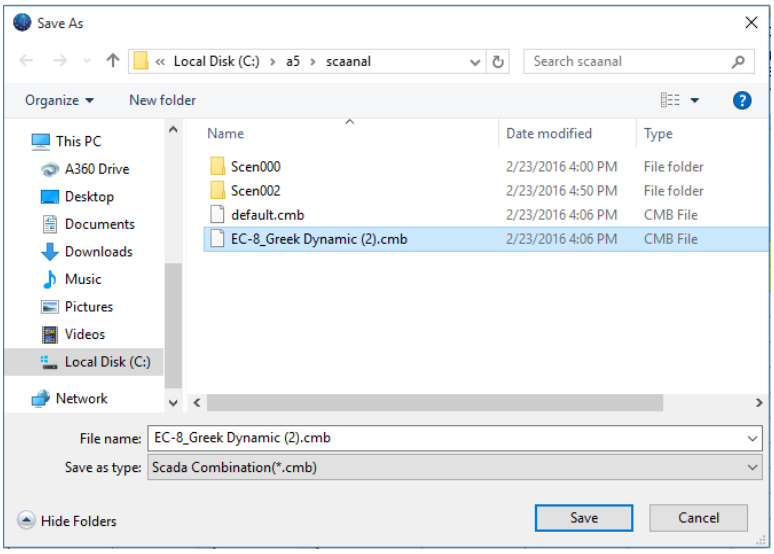


With the 

scenario activated, select “Combinations” and in the dialog form that opens click 

, to fill in automatically the coefficients of the dynamic analysis according to Eurocode.
Click “Save”, to store the combinations file inside the folder of your project to use it later on during the “Post-Processor” and “Member Design” process.

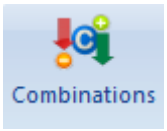

Load Groups Combinations
γG 1.35
γE 1
γGE 1
ψ2 0.3
γQ 1.5
γE0.3 0.3
Wind - Snow
Ultimate
ΣγG+γQ+Σγψ0Q
ΣG+ψ1Q+Σψ2Q
ΣG+E+Σγψ2Q
Serviceability
ΣG+Q+Σψ0Q
ΣG+ψ1Q+Σψ2Q
ΣG+Σψ2Q
Calculation
Delete All
Type
Direction
LC1
LC2
LC3
LC4
LC5
LC6
Scenario
Load Case
Load Type
Actions
Description
EC-8_Gree...
Category A...
Comb.:1 Ultimate No 1.35 1.50
Comb.:2 Ultimate No 1.00 0.50
Comb.:3 Ultimate Dir. +X 1.00 0.30 1.00 0.30 1.00 0.30
Comb.:4 Ultimate Dir. +X 1.00 0.30 1.00 0.30 1.00 0.30
Comb.:5 Ultimate Dir. +X 1.00 0.30 1.00 0.30 1.00 -0.30
Comb.:6 Ultimate Dir. +X 1.00 0.30 1.00 0.30 1.00 -0.30
Comb.:7 Ultimate Dir. +X 1.00 0.30 1.00 0.30 -1.00 0.30
Comb.:8 Ultimate Dir. +X 1.00 0.30 1.00 0.30 -1.00 0.30
Comb.:9 Ultimate Dir. +X 1.00 0.30 1.00 0.30 -1.00 -0.30
Comb.:10 Ultimate Dir. +X 1.00 0.30 1.00 0.30 -1.00 -0.30
Comb.:11 Ultimate Dir. +X 1.00 0.30 1.00 -0.30 1.00 -0.30
Comb.:12 Ultimate Dir. +X 1.00 0.30 1.00 -0.30 1.00 -0.30
Add
Remove
Read
Save
TXT
Default Combinations
OK
Cancel


Save As
Local Disk (C:) › a5 › scaanal
Search scaanal
Organize
New folder
This PC
A360 Drive
Desktop
Documents
Downloads
Music
Pictures
Videos
Local Disk (C:)
Network
Name
Date modified
Type
Scen000 2/23/2016 4:00 PM File folder
Scen002 2/23/2016 4:50 PM File folder
default.cmb 2/23/2016 4:06 PM CMB File
EC-8_Greek Dynamic (2).cmb 2/23/2016 4:06 PM CMB File
File name: EC-8_Greek Dynamic (2).cmb
Save as type: Scada Combination(*.cmb)
Hide Folders
Save
Cancel

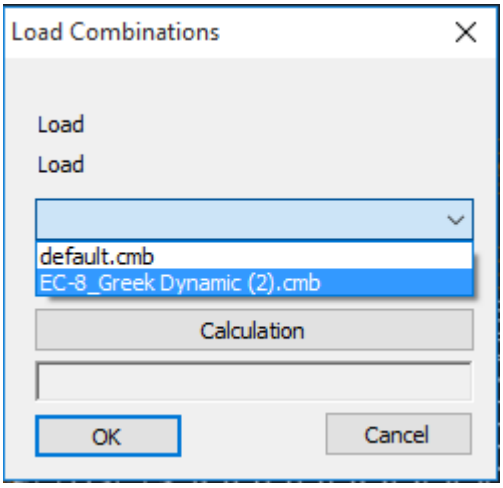
5. RESULTS

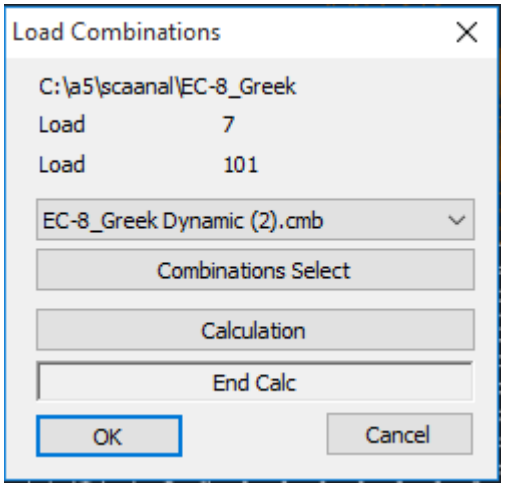
5.1 Deformed shape of Model:

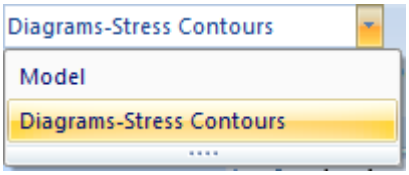
Move to “Results” unit and check the deformation of the model.

Select 

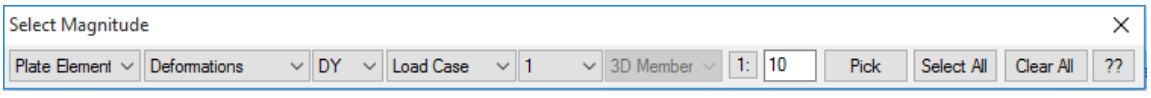
and calculate (click Calculation) the combinations that you previously saved (Select the File).



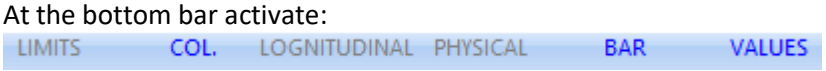


Select from the list 

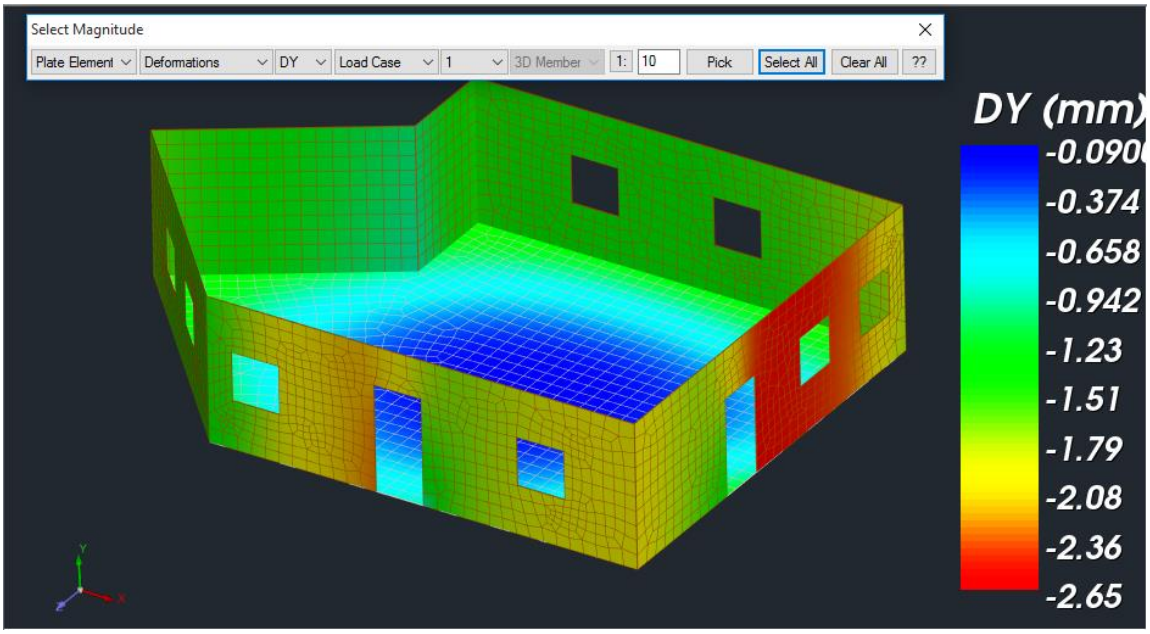
and in the dialog window, select to view “Plate Elements” -> “DY Deformations” caused by “Load Case 1” for all model (Select All):



At the bottom bar activate:



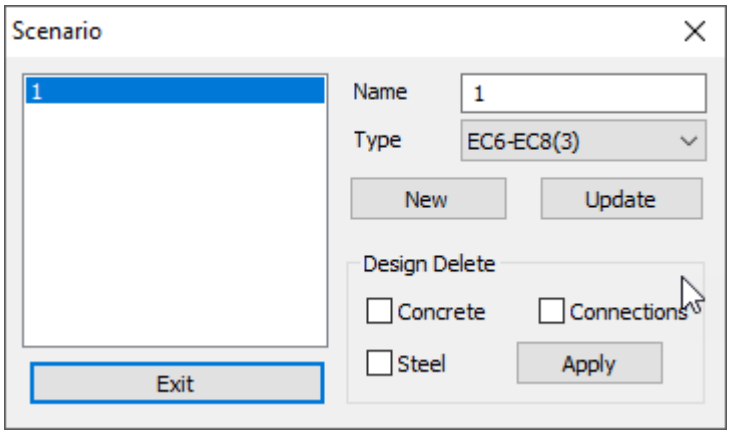
Color representation and Values bar, to view the next image:

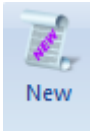

Select Magnitude
Plate Element
Deformations
DY
Load Case
1
3D Member
1:
10
Pick
Select All
Clear All
??
DY (mm)
-0.090
-0.374
-0.658
-0.942
-1.23
-1.51
-1.79
-2.08
-2.36
-2.65

6. DESIGN

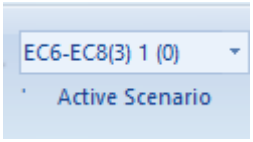
6.1 Design Scenario Creation in accordance to Eurocode provisions:

For masonry structures, SCADA Pro embeds the checks of the Eurocode 6. Thus it is necessary to create a Eurocode design scenario to perform the respective checks with the “Masonry Design” command.

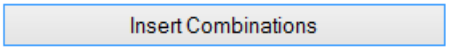


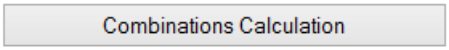
Move to “Design” unit and click 

to create the desired scenario by selecting EC6. Enter a name and click “New”.

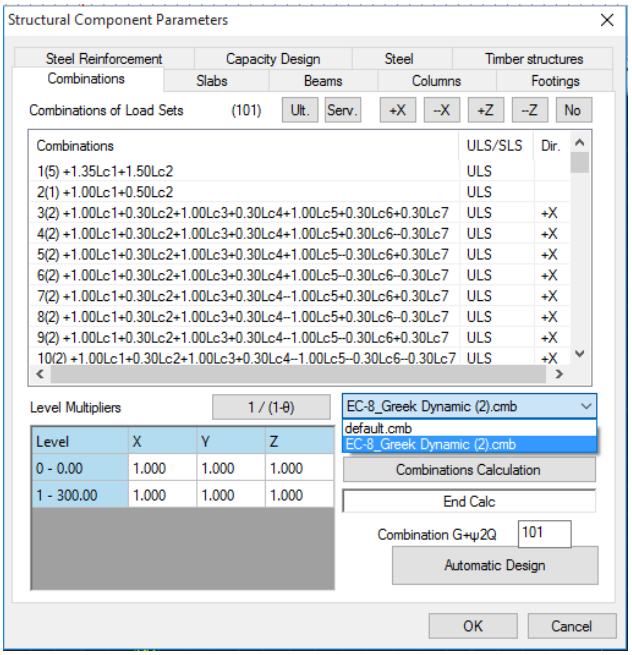
Select the considered scenario 

and and click 

. 

that opens the folder with the registered .cmb files. Select the file and press 

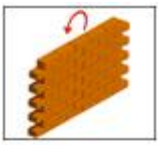
. The program calculates the combinations and by clicking OK the window closes.



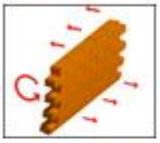
6.2 Masonry structure checks according to Eurocode 6:


New masonry building (EC6)

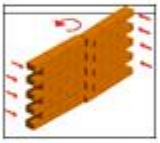
Masonry design according to Eurocode 6 includes seven checks:



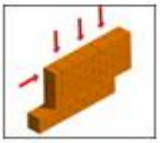
1. Wall subjected to in-plane bending



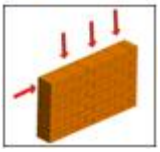
2. Wall subjected to out-of-plane bending across an axis parallel to the bed joints



3. Wall subjected to out-of-plane bending across an axis perpendicular to the bed joints



4. Wall subjected to shear loading

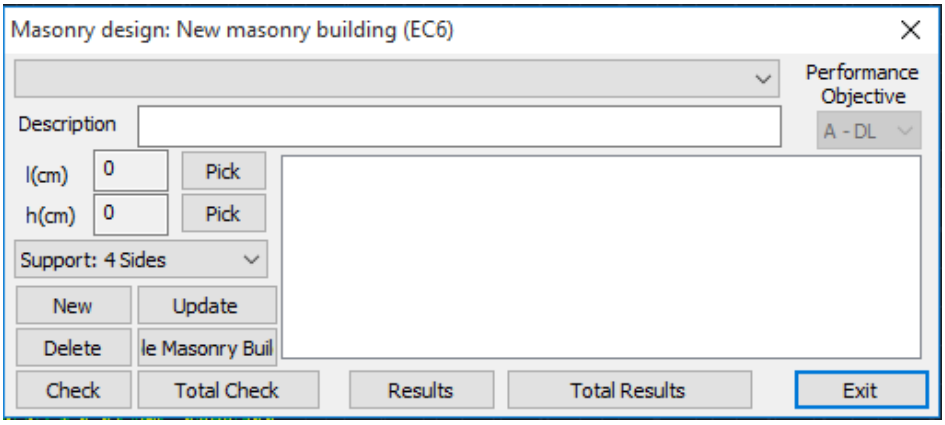


5. Wall subjected to mainly vertical loading, top
6. Wall subjected to mainly vertical loading, middle
7. Wall subjected to mainly vertical loading, bottom

⚠ These seven adequacy checks are defined for each wall or each wall section (spandrel), according to the user defined division.

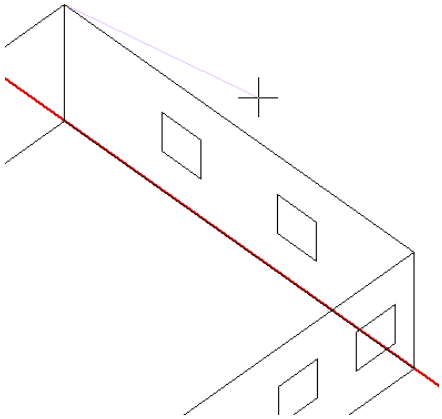
⚠ Buildings that meet the requirements to be identified as "Simple" are excluded from all the above adequacy checks.

In the dialog box that opens, the user must identify the parts of the walls to make the required checks:



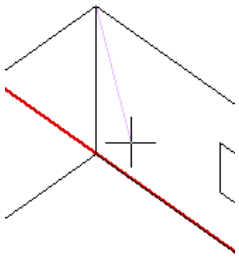
In "Description" field, type a name (at least four characters) for the considered wall or spandrel.

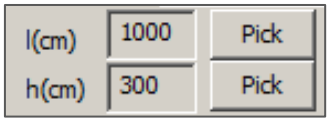
Use these fields to define the geometry of the considered wall (or spandrel):
Click the first "Pick" to define the length of the wall by left clicking at the end points.



Press the button "**Pick**" (the first one) to define the x starting and ending points of the part (i.e. length definition). After the starting point is clicked, an elastic chord emerges from it, waiting to link it with the ending point (second click).

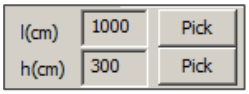
In the same manner, press the second button "Pick" (under the first one), to define the y starting and ending points of the part (i.e. height definition).

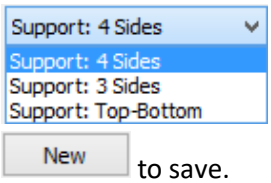




The values are automatically assigned to the fields "l" and "h".

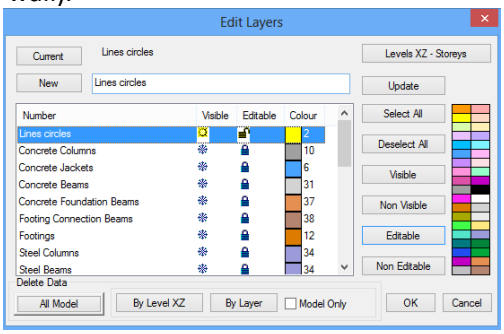
Next, choose the type of the wall and press the button "New"

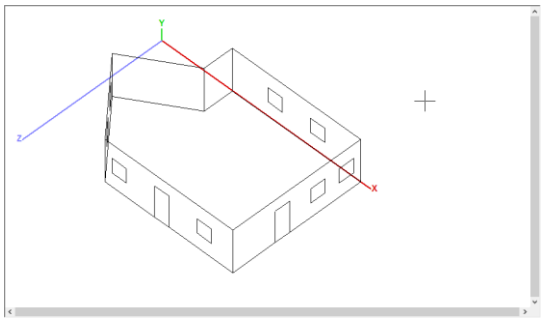




constraint to save.

⚠ In order to pick the points easily, take advantage of the object snap utility in a most efficient way by deactivating any layers that "confuse" the picking procedure (e.g. when a whole wall is to be picked, keep active only the layers of "Lines circles" to pick the corner points of the wall).





⚠ A selected wall can be recalled from the list and:

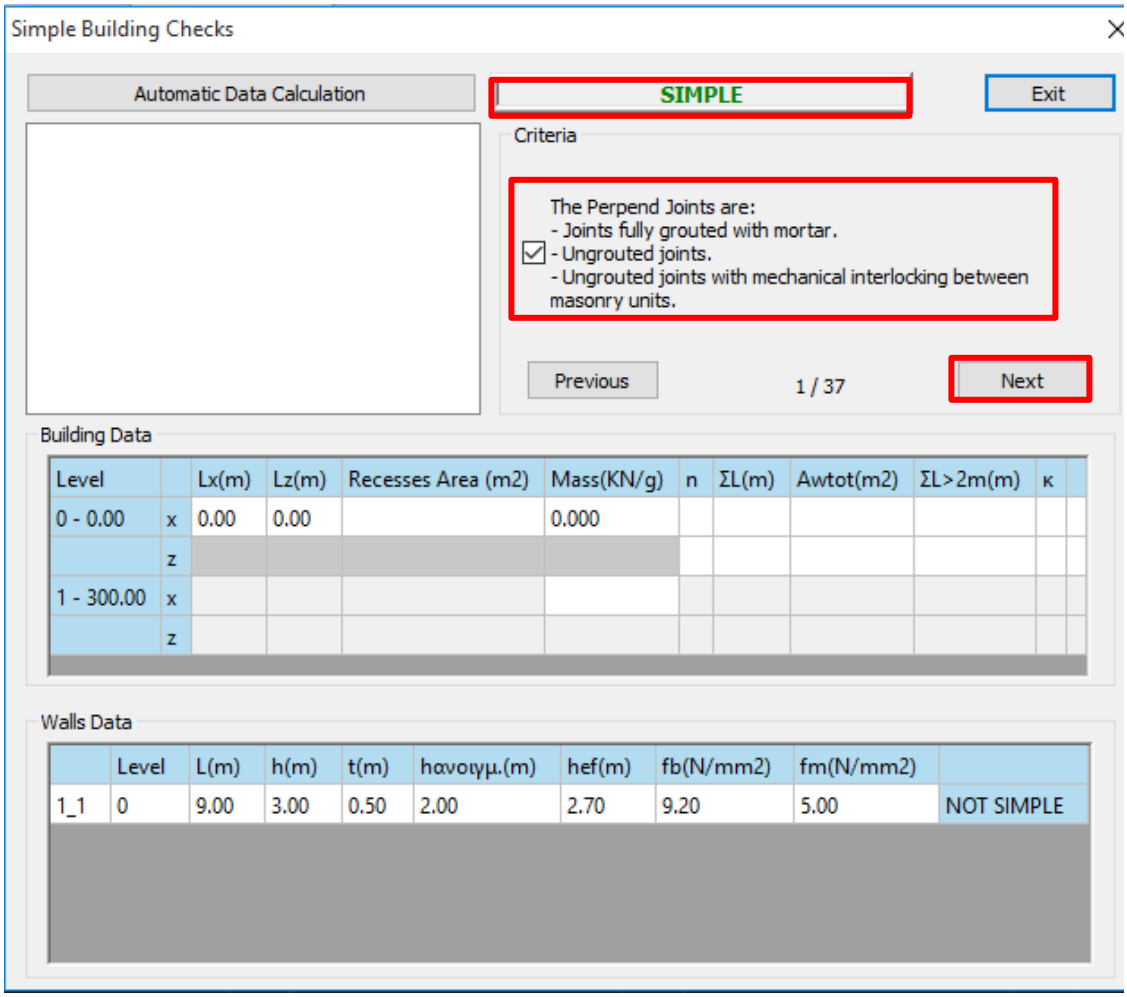
- Be modified. Make any change in the name, geometry, type of constraint and then press Update to save it.
- Be deleted by pressing Delete command. The wall will still be visible in the list but with a Delete marking (e.g. 1_1(Delete)).

⚠ This is an iterative procedure and is not accomplished until all the walls or parts of walls are defined.

⚠ Before executing the checks, examine the possibility that the building can be characterized as "Simple building". In such case, a verification by the checks above is not mandatory (EC8 – 9.7.1).

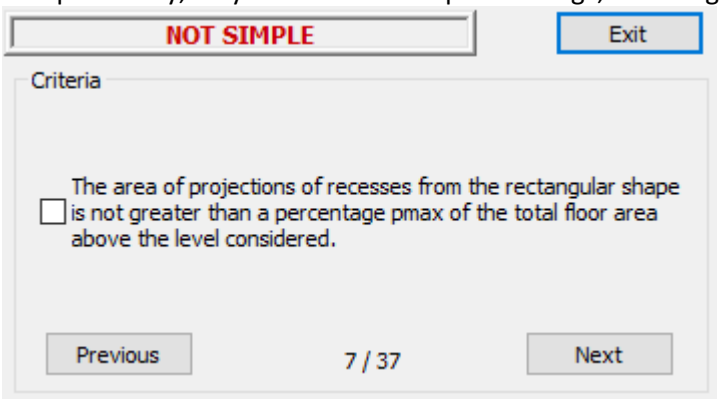
Simple Masonry Building Checks

Press the corresponding button and in the dialog box, the following parameters are presented



The field "Criteria" presents one by one, the 37 requirements according to EC8, in order a building to be defined as "Simple". The user should see a tick next to any of the requirements that is satisfied, and move on to the next one.

⚠ All the requirements must be satisfied or the building cannot be characterized as "Simple". As said previously, only in case of a "Simple building", the design checks of EC6 are optional.



The 37 criteria of the previous stage are the initial step of the "simple building" characterization procedure. It must also conform to the demands in Table 9.3 of EC8, in order the characterization to be finalized. These demands concern both the building in total and each wall consecutively, and the design check process starts with the command "Automatic Data Calculation" Automatic Data Calculation (per Level/Wall).

⚠ Again, a failed check means that the building cannot be characterized as "Simple"

Building Data

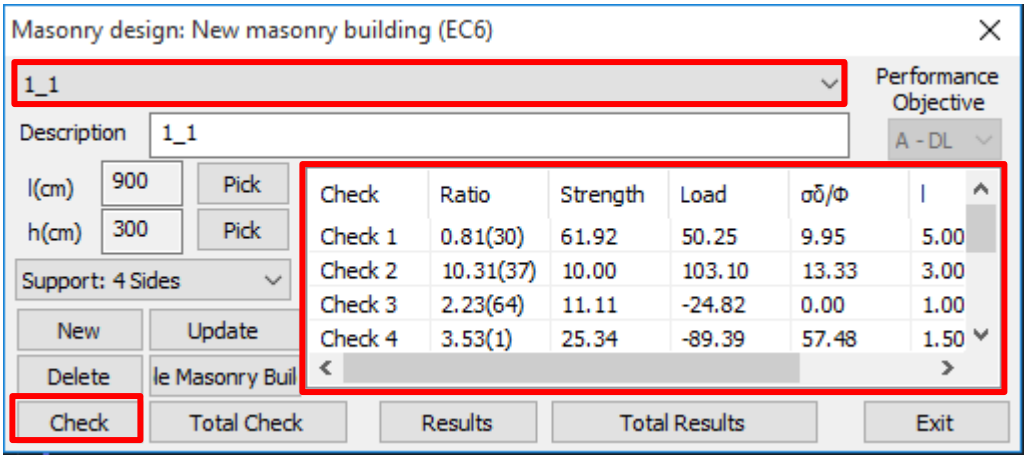
Level		Lx(m)	Lz(m)	Recesses Area (m2)	Mass(KN/g)	n	ΣL(m)	Awtot(m2)	ΣL>2m(m)	κ	
0 - 0.00	x	0.00	0.00		0.000	5	10.72	5.36	4.72		
	z					7	10.02	5.01	3.00		
1 - 300.00	x					0	0	0	0		
	z					0	0	0	0		

Walls Data

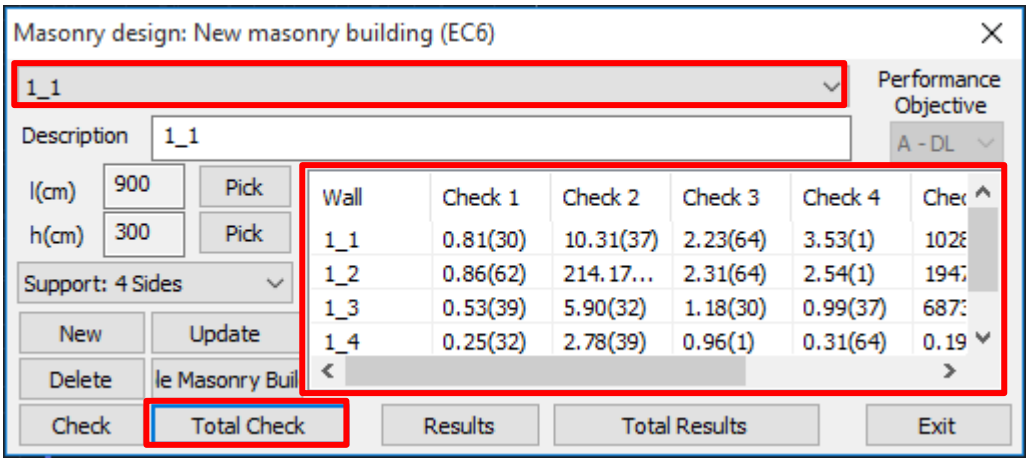
	Level	L(m)	h(m)	t(m)	ηανοιγμ.(m)	hef(m)	fb(N/mm2)	fm(N/mm2)	
1_1	0	9.00	3.00	0.50	2.00	2.70	9.20	5.00	NOT SIMPLE
1_2	0	9.00	3.00	0.50	2.20	2.70	9.20	5.00	NOT SIMPLE
1_3	0	6.02	3.00	0.50	1.00	2.40	9.20	5.00	NOT SIMPLE
1_4	0	4.72	3.00	0.50	0.00	2.14	9.20	5.00	NOT SIMPLE

⚠ In case of "NOT SIMPLE" structures, the adequacy checks by EC6 provisions must be performed.

Check Automatic application of the seven design checks for a selected part of the wall.



Total Check Automatic application of the seven checks in the structure in total.

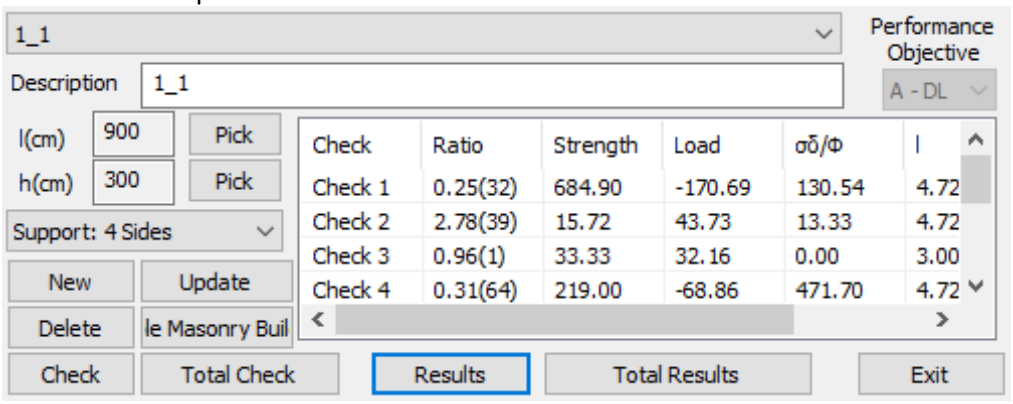


The design checks are applied on sections (horizontal and vertical) concerning the EC6 design code.

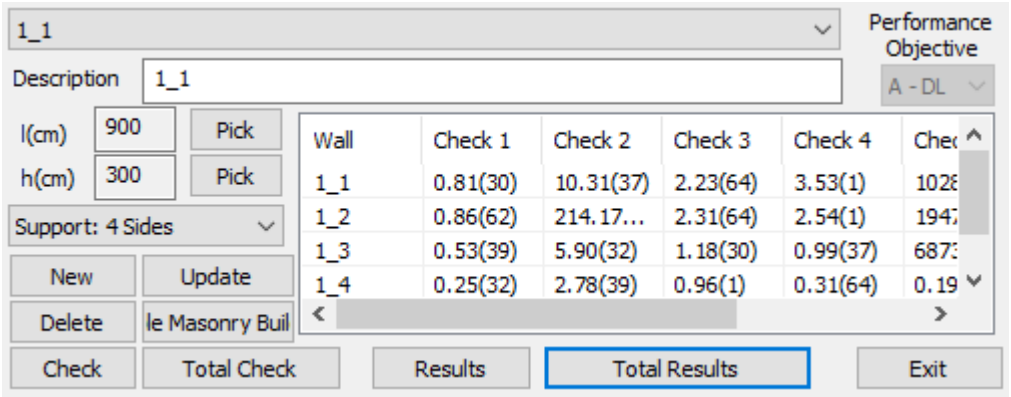
⚠ SCADA Pro scans each selected wall, at first horizontally and then vertically, the wall sections (strips of finite elements) are detected, and all the checks are applied in each section.
⚠ During the scan, each strip of finite elements is colored according to the results of the design checks; blue-green (all design checks of the section are satisfied) or red (one or more design checks of the sections are not satisfied.

Since the design checks' procedure has been completed, the user can elaborate on the results.

The command “Results” Results presents the results of all the design checks for the selected wall or part of wall



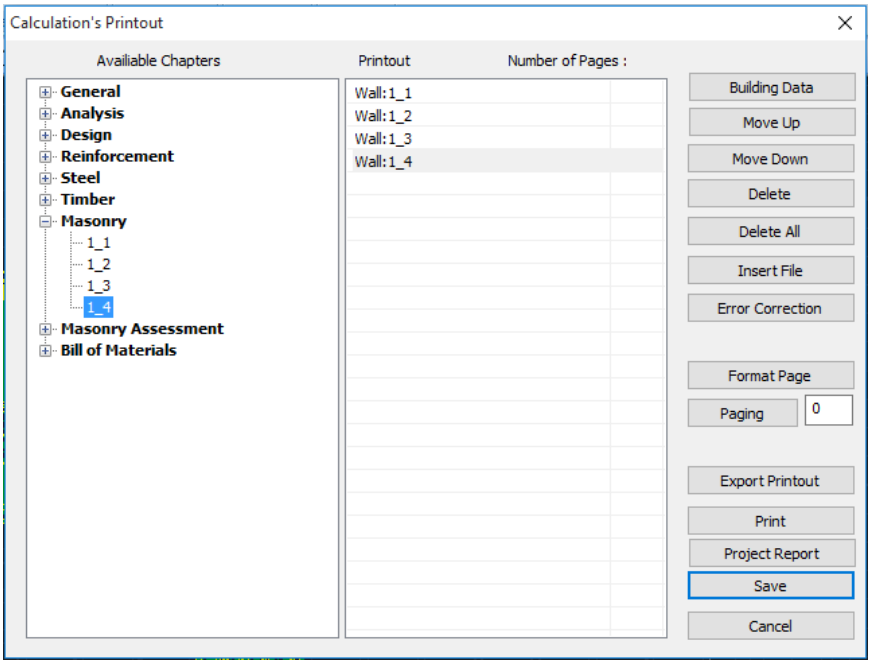
The command “Total Results” Total Results presents the results for all the walls of the building

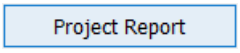


For better and more detailed appearance of these results view the "Printout”.

7. PRINTING

Through the “Addons” unit select the “Print” command and in the dialog box select Masonry, to expand the wall list.



Double click in each wall, to transfer the respective data to the report and click 

to export the Project Report.