SCADA Pro™
Structural Analysis & Design

Example 7
Masonry building – Assessment (ec8-3)

ACE HELLAS
INTEGRATED SOLUTIONS

CONTENTS

OVERVIEW ..... **Error! Bookmark not defined.**

INTRODUCTION ..... **Error! Bookmark not defined.**

THE NEW ENVIRONMENT ..... **Error! Bookmark not defined.**

GENERAL DESCRIPTION 6

- **Geometry** 6
- **Materials** 6
- **Regulations** 6
- **Load and Analysis assumptions** 6

1. STEP: DATA INPUT – MODELING 8

1.1 Masonry library – wall definition: 9

1.2 Masonry units 10

1.3 Mortar 12

1.4 Wall 12

1.5 Modeling: 14

1.5.1 Insert a dwg file and line recognition 14

1.5.2 Front view identification 16

1.5.3 Mesh Group Definition: 20

1.5.4 New Mesh sub-Group Definition for the slabs modeling 22

1.5.5 Surface Calculation 24

1.5.6 Mathematical Model calculation: 25

2. STEP: LOADS DEFINITION 28

2.1 Manually imported: 28

2.2 Load Destribution on the Surface 29

3. STEP: ANALYSIS 30

3.1 Masonry structure analysis in accordance with Eurocode 30

3.2 Combinations 33

4. STEP : RESULTS 34

4.1 Deformed shape of Model: 34

5. STEP: MEMBERS DESIGN 35

5.1 Design Scenario Creation in accordance to Eurocode provisions: 35

5.2 Masonry structure checks according to Eurocode 36

6. STEP: STRENGTHENING 39

6.1 Strengthening with concrete jacket..... 40

6.2 Textile Reinforced Mortar (TRM)..... 44

• 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!

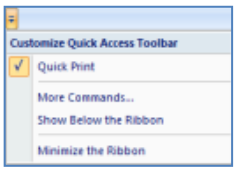
• INTRODUCTION

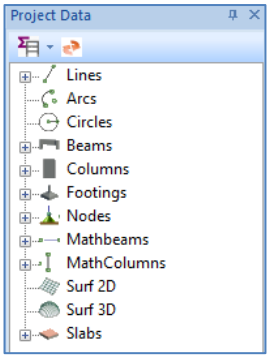
The current manual comes as an aid for a new user of SCADA Pro, making the interface of the software as familiar as possible. It consists of several chapters, where one after the other, describes the consecutive steps of a simple example of a loadbearing masonry project.
The most useful information is 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

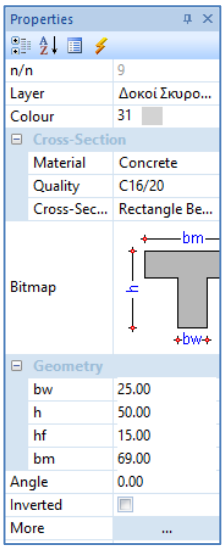
The new interface of the SCADA Pro software is based on the RIBBON structure, thus, the several commands and tools are reached neatly. The main idea of the RIBBON structure is the grouping of commands that have small differences and work in the same context, in a prominent position different to each 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.

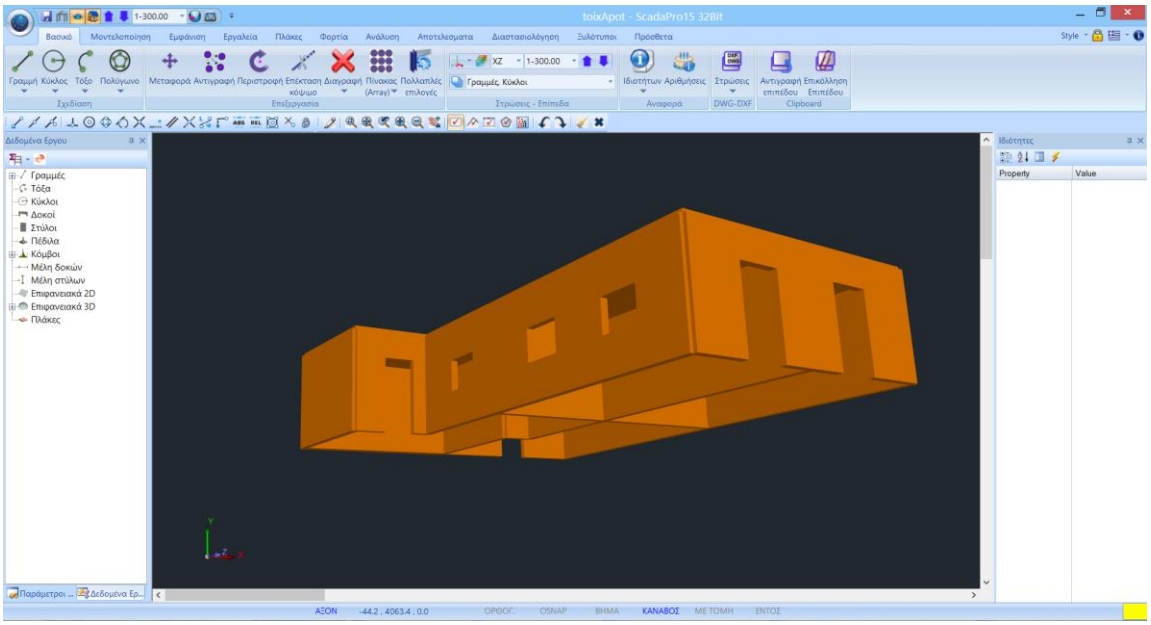


The "Properties" list that shows up at the right side of the window, not only shows all the properties of the entity shown but can be used for any quick and easy changes, the user wants to make, too.

GENERAL DESCRIPTION

Geometry

The considered single floor masonry structure consists of 10 external views with openings and six internal walls.



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. The building will be considered as anchored to the base.

Regulations

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

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).

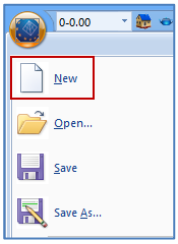
- **Notes**

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.

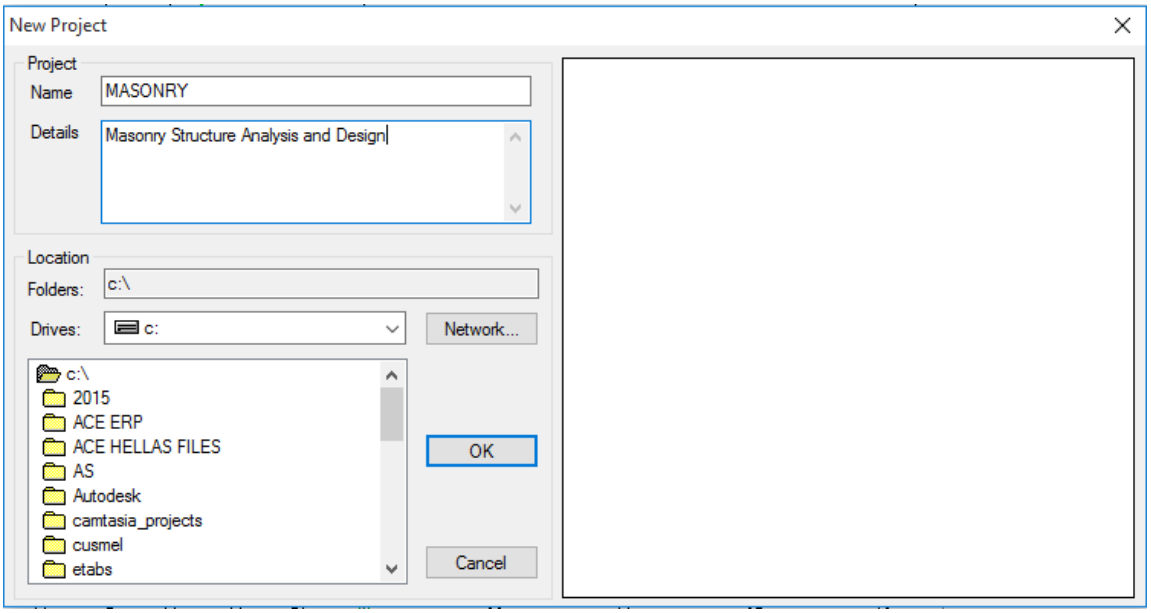
1. STEP: DATA INPUT – MODELING

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.

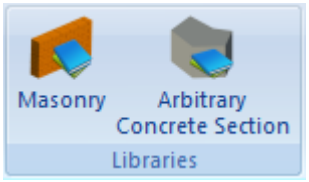
Use the  button located at the initialization window, or select ”New” 

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

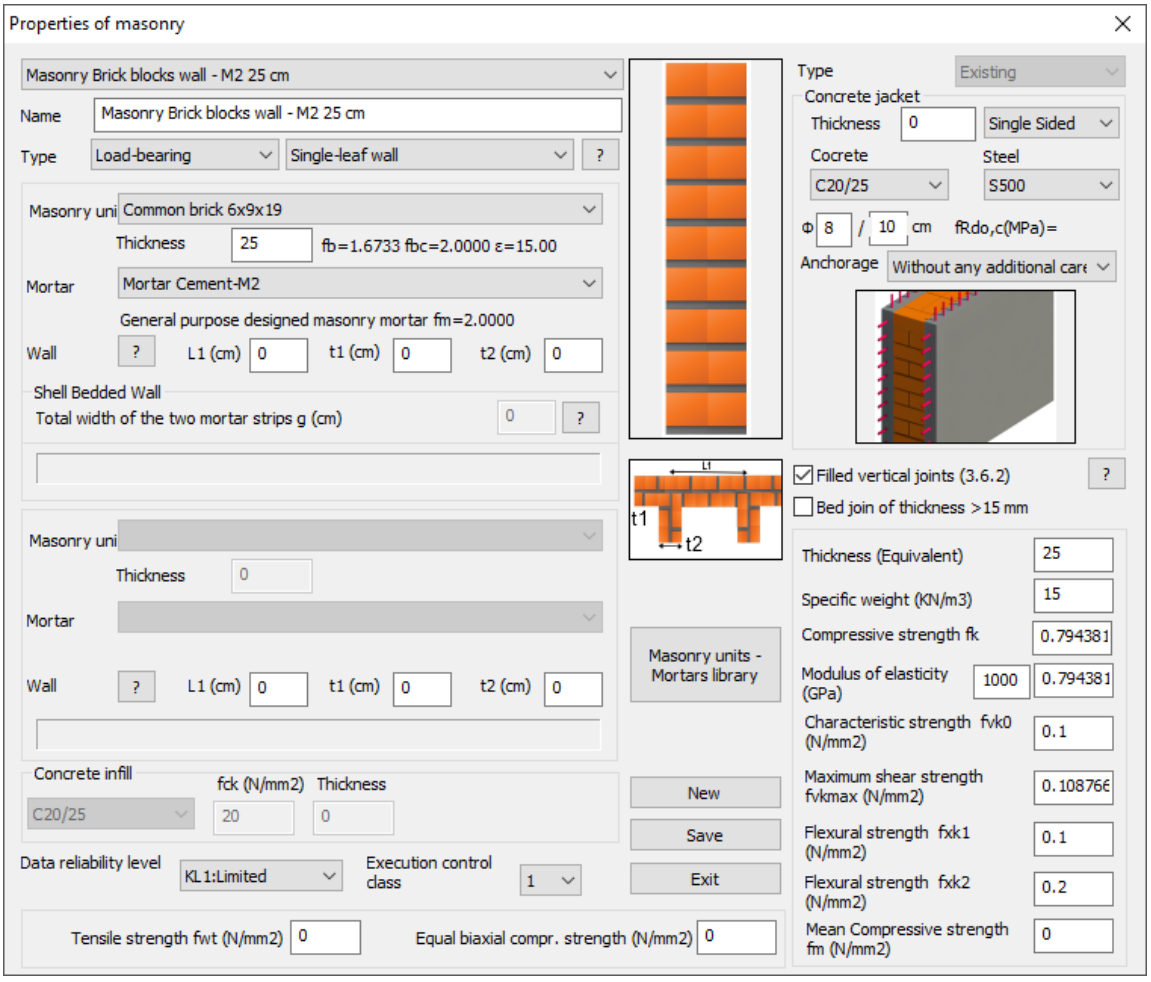


⚠ The name of the file can contain up to 8 characters of the Latin alphabet without any symbols (/, -, _) nor spaces. You can add a description or add some information related to the structure, in the “Info” field.

1.1 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".

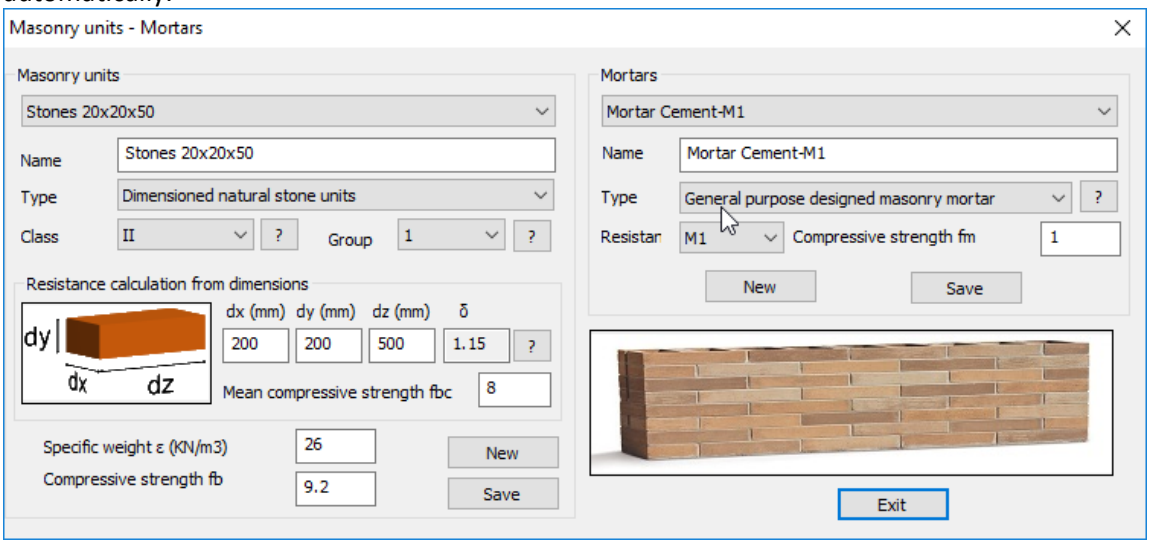
⚠ *Depending on the selected TYPE of masonry, in the dialog box, some fields are enabled or disabled.*

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

EXAMPLE

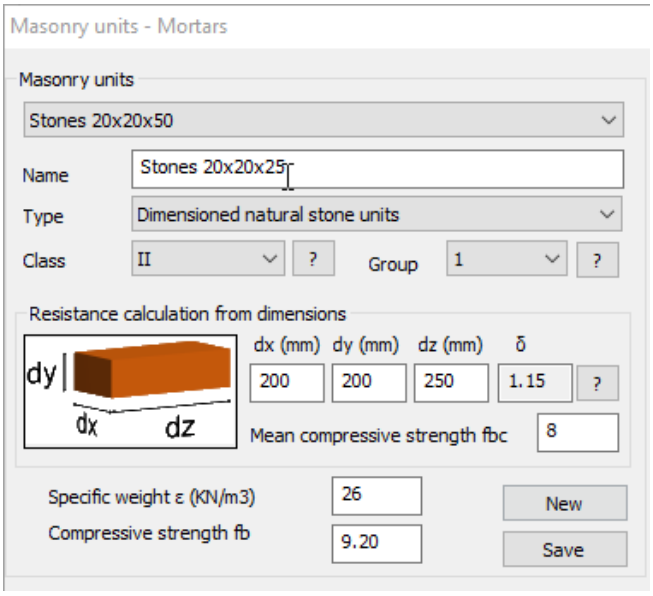
In this example all walls are of single-leaf type with dimensional natural stone units 20x20x25 and M5 mortar named, “Wall M5 0.50”.

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.



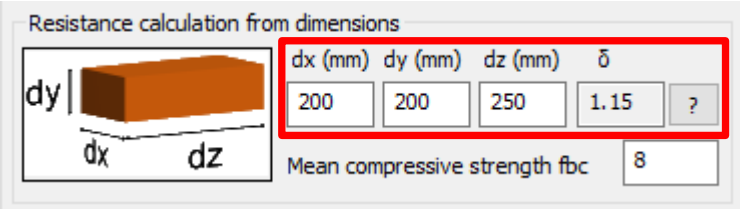
For this example we chose:

1.2 Masonry units

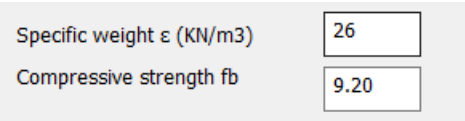


Name: Dimensional natural stone units 20x20x25 (type in)
Type: Dimensional natural stone units (select from list)
Category: II, Group: 1 (select from list)

For the Strength Calculation from Dimensions, type the dimensions of the masonry unit and the reduction factor δ, is automatically filled by the respective table ?



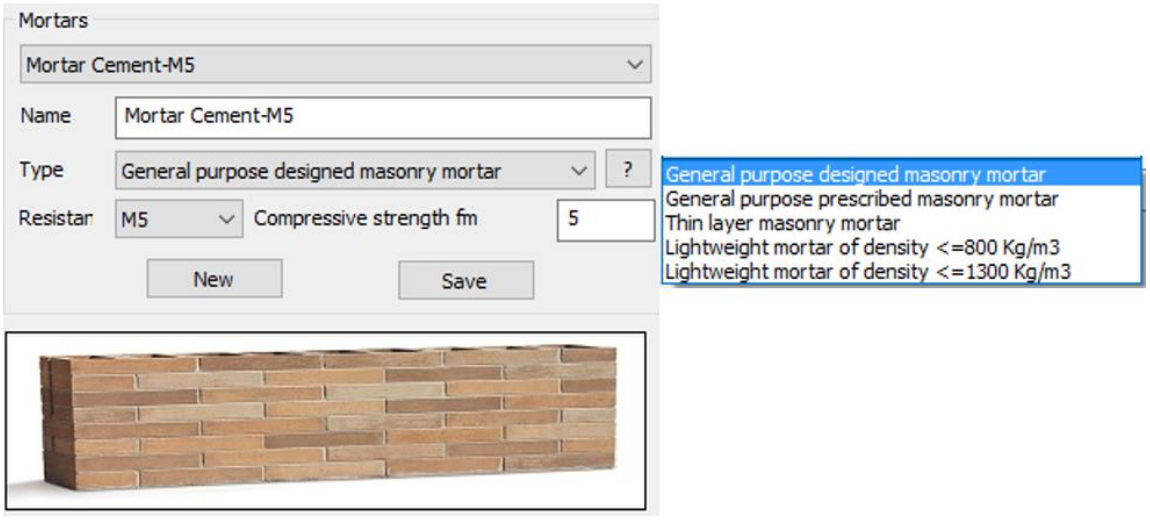
Type the “Compressive Strength” fbc, which is the average value of experiments regarding the compressive strength of the masonry units and the “Specific Weight ε”.
The “Compressive Strength” fb is automatically calculated by the program.



Select New 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.

1.3 Mortar

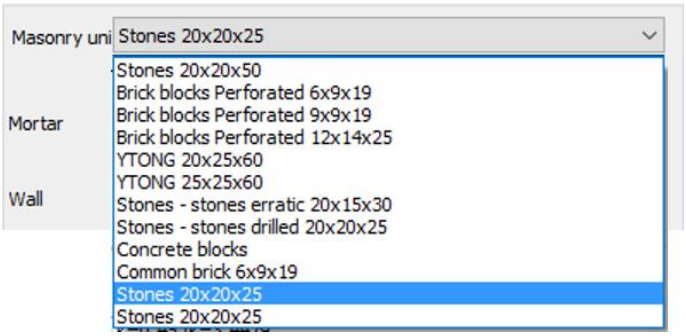


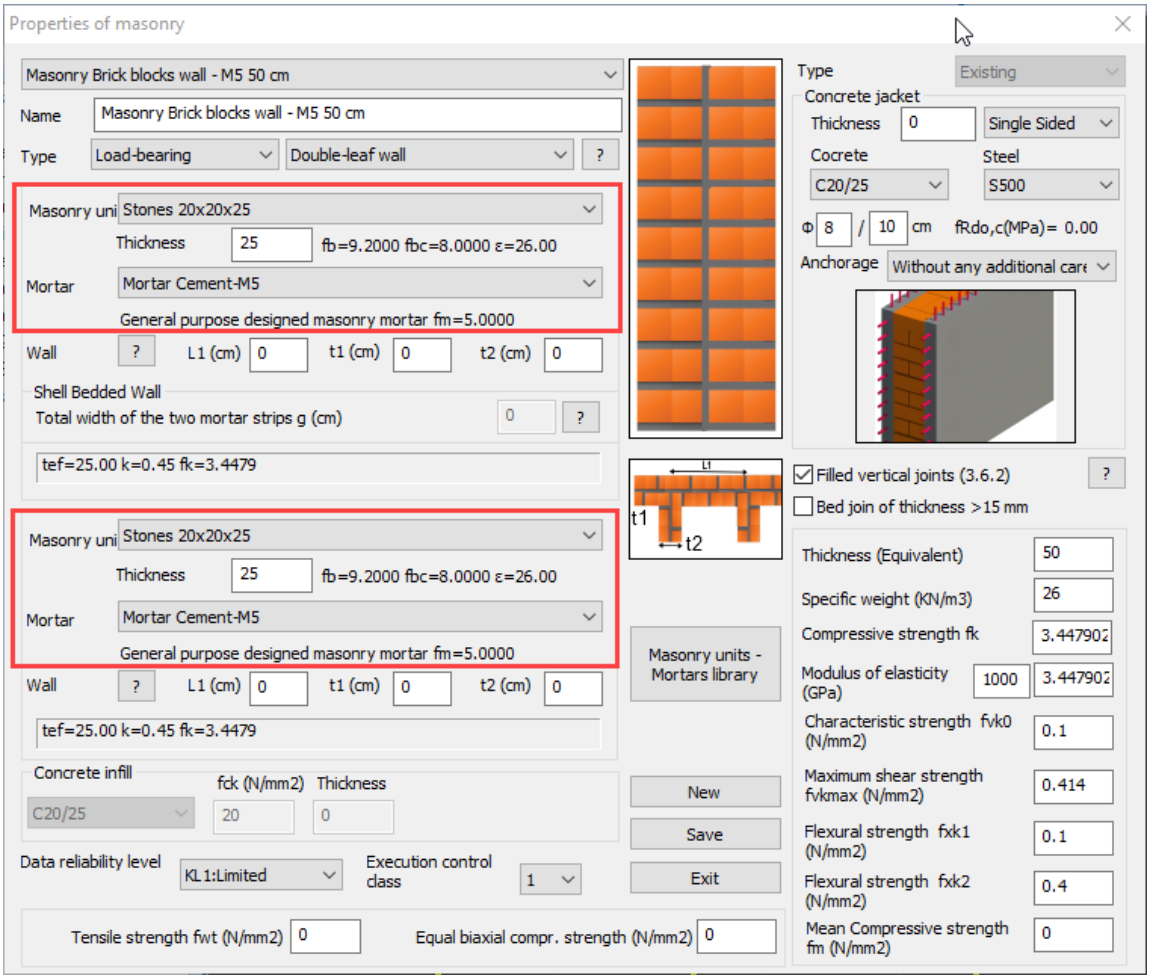
Name: Mortar-M5(select from list)
Type: General Purpose Mortar (select from list)
Strength: M5 (select from list)

The compressive strength Fm is automatically filled in by the program.

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

1.4 Wall





Name: Masonry stone wall M5 0,50 (typing)
Type: Double-leaf wall (select from the list)

Masonry Units: Stones – stones drilled (defined previously) and
Thickness: 25 cm
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
Mortars: Mortar cement-M5
Prescribed Masonry Unit fm is automatically updated.
General purpose designed masonry mortar fm=5.0000
Double-leaf:
Thickness (Equivalent): 50 cm

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

1.5 Modeling:

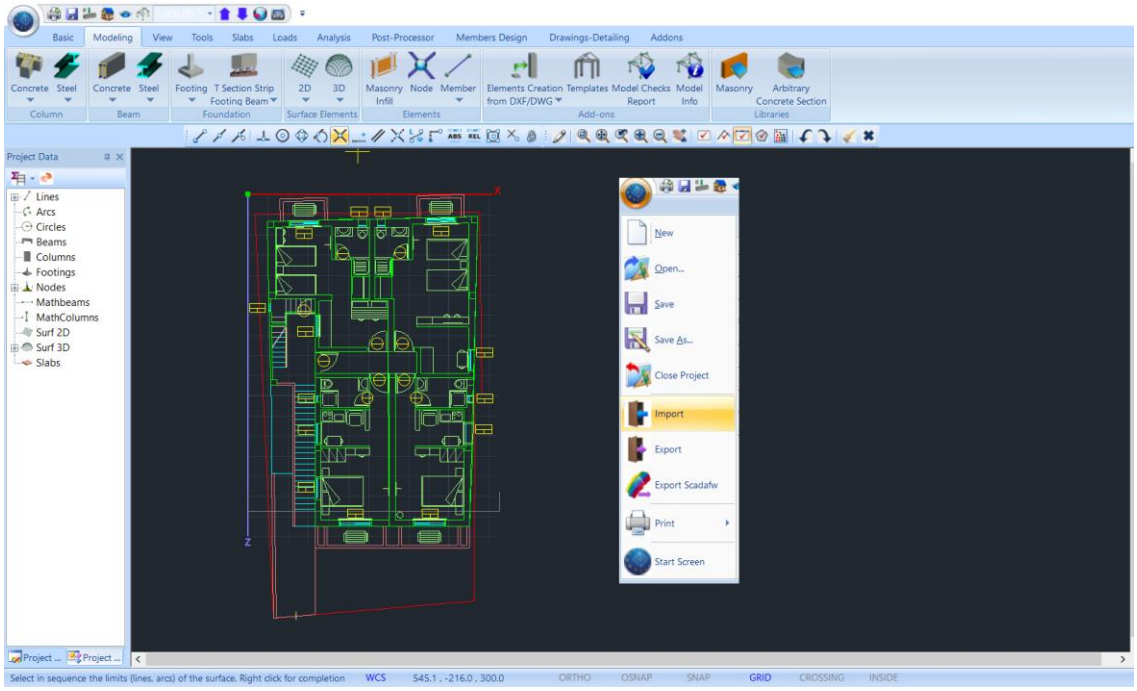
1.5.1 Insert a dwg file and line recognition

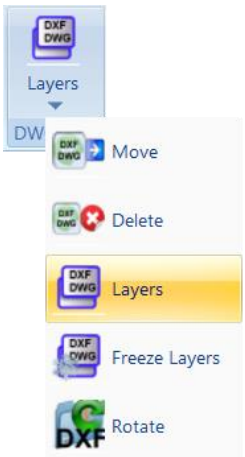
SCADA Pro gives you the possibility to create a masonry structure on any external boundary, by using the tool “Templates”, quickly and easily.

The procedure is as follows:

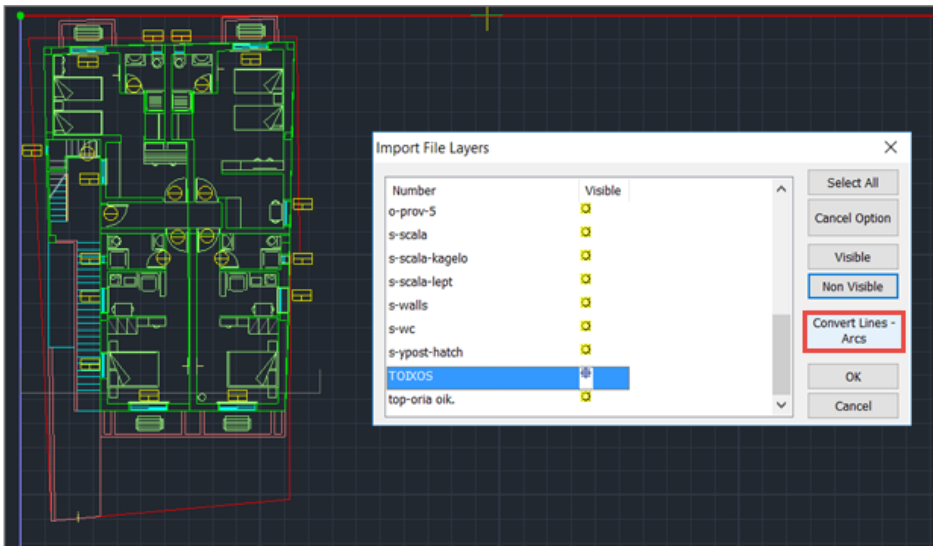
- Enter a plan view in DXF or DWG file format by using the 

command

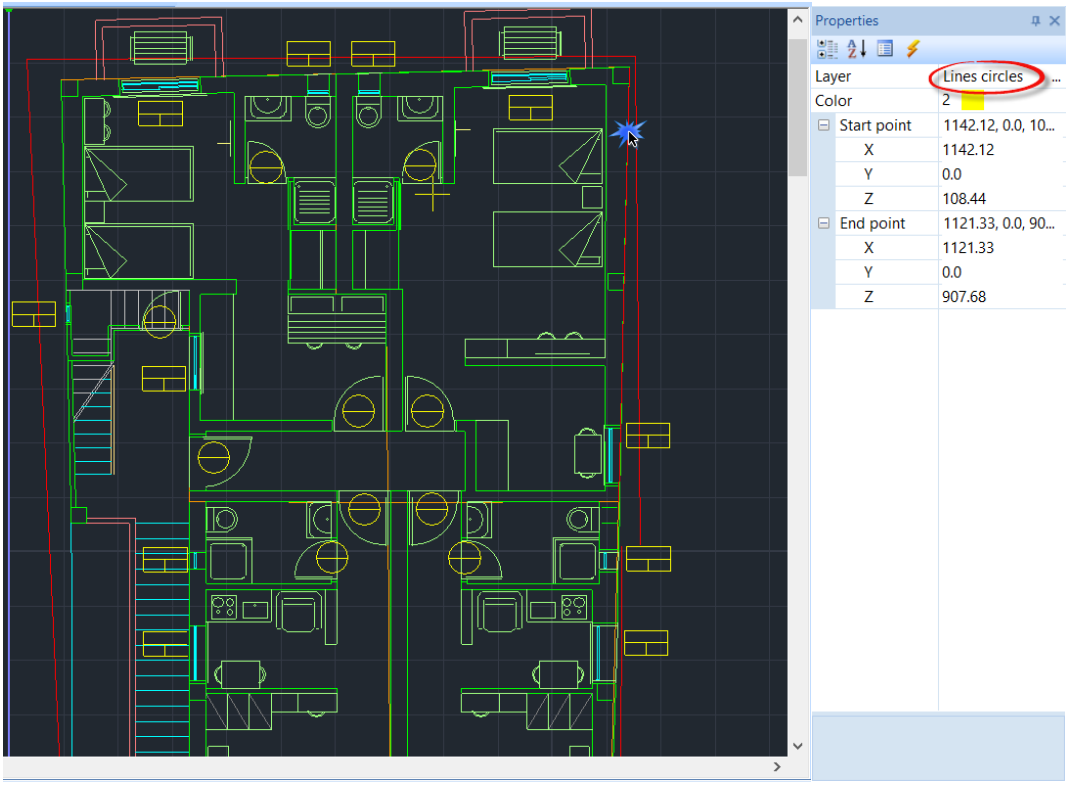


- Use the command “Layers” to open the list of the design layers. 

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

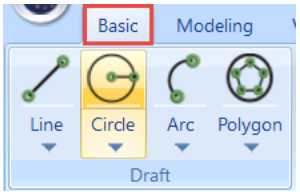


⚠ Using this command, all lines belonging to the dwg layer "TOIXOS", converted into drawing lines of SCADA and so is recognizable by the command "Front View Identification", explained in detail in the next chapter.



IMPORTANT NOTES

- 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.

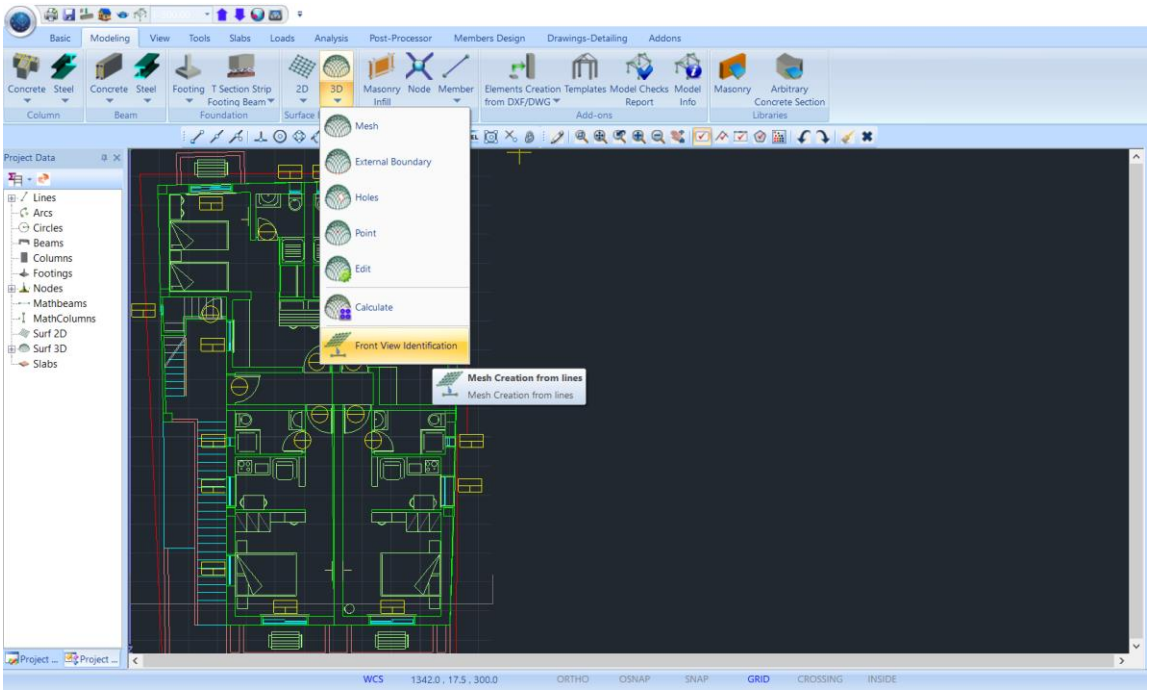


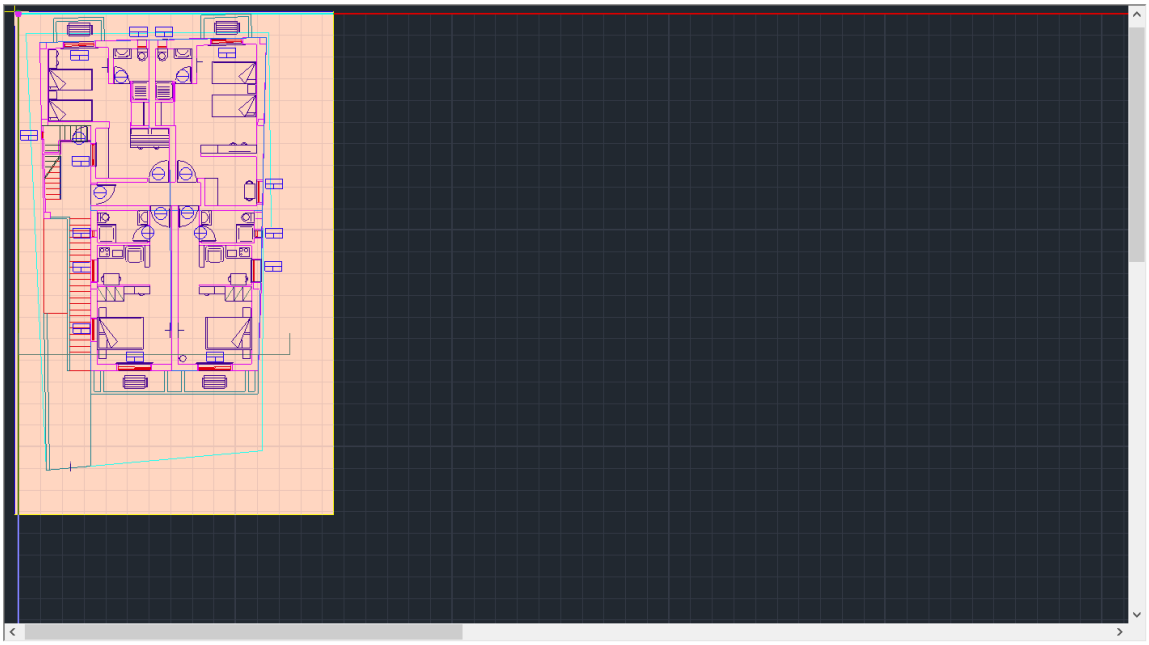
- The dwg file auxiliary file, inserted in the SCADA environment in the active XZ level identifying the origin to the upper left point of the design  .

- Lines and polylines defining the static walls, to be recognized as SCADA's lines, should belong to a separate layer, and using the command "Convert Lines, Arcs" to obtain the identification.

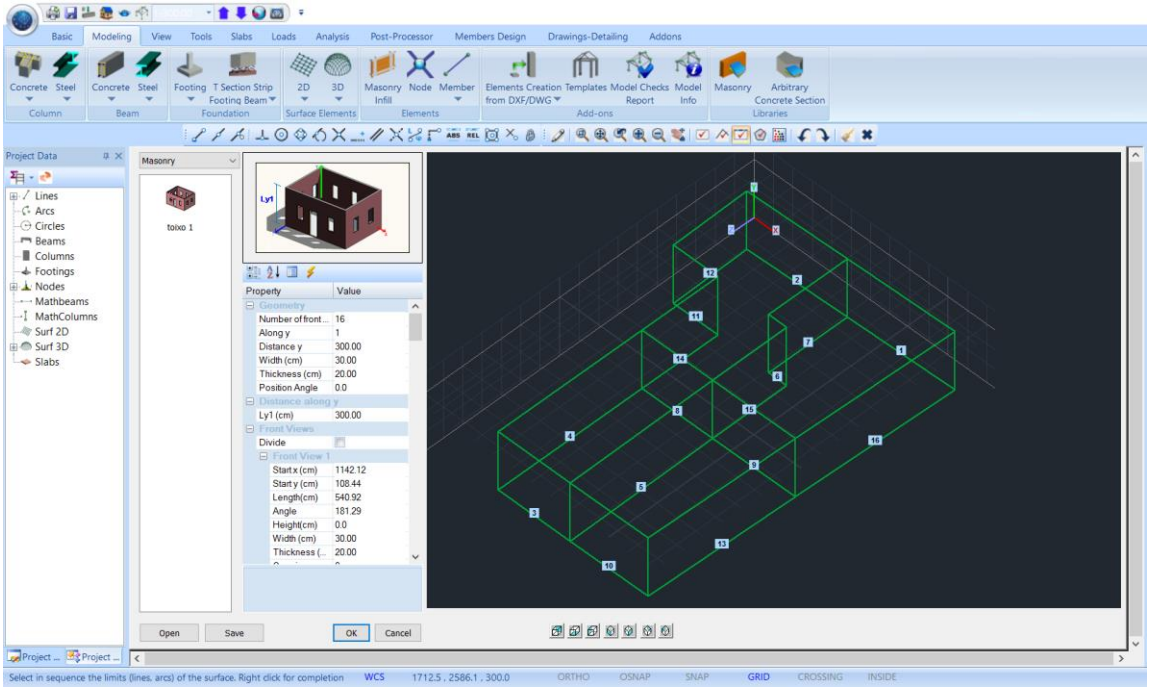
1.5.2 Front view identification

In "Modeling" select the command "3D">>"**Front View Identification**", and use the window  to select the entire floor plan.





Right click and opens the Templates window:



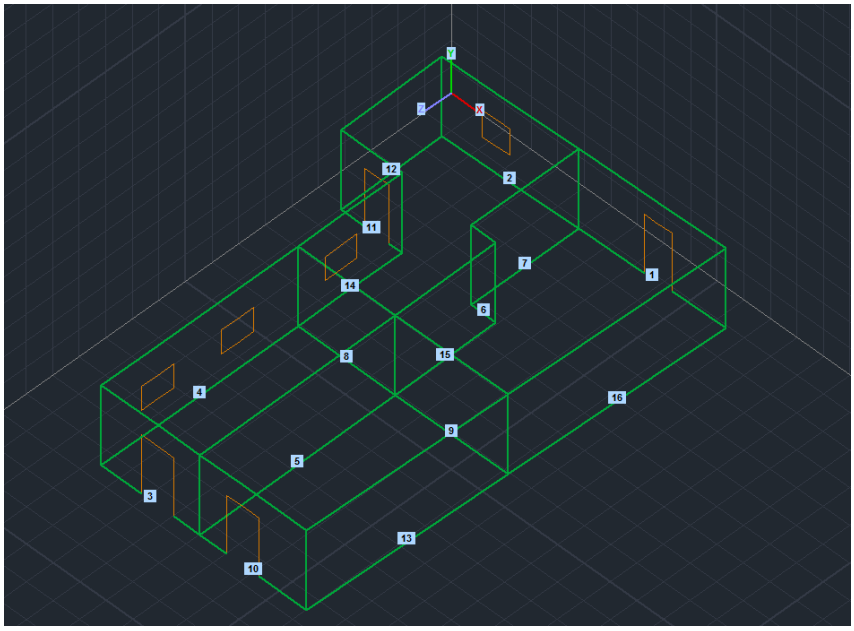
The program automatically recognizes the geometry of the floor plan. Proposes by default the height and creates the views as to the Universal axes.

Property	Value
Geometry	
Number of front ...	16
Along y	1
Distance y	300.00
Width (cm)	50.00
Thickness (cm)	20.00
Position Angle	0.0
Distance along y	
Ly1 (cm)	300.00
Front Views	
Divide	
Front View 1	

The user must define the number of floors and the individual altitudes, the thickness of the walls and openings on each side.

Property	Value
Divide	
Front View 1	
Start x (cm)	1142.12
Start y (cm)	108.44
Length(cm)	540.92
Angle	181.29
Height(cm)	0.0
Width (cm)	50.00
Thickness (...	20.00
Opening	1
Opening 1	
Start x (c...	**200.00**
Start y (c...	**0.0**
Width(cm)	100.00
Height(c...	**220.00**
Front View 2	
Start x (cm)	95.15
Start y (cm)	132.03
Length(cm)	506.32

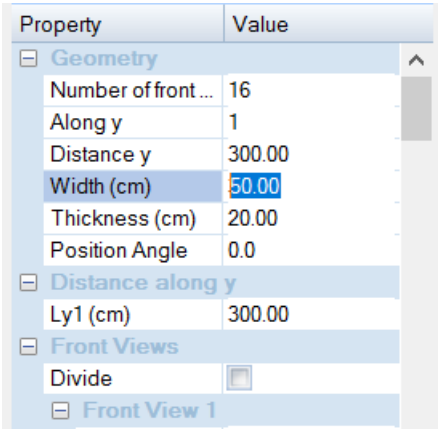
Start y (cm)

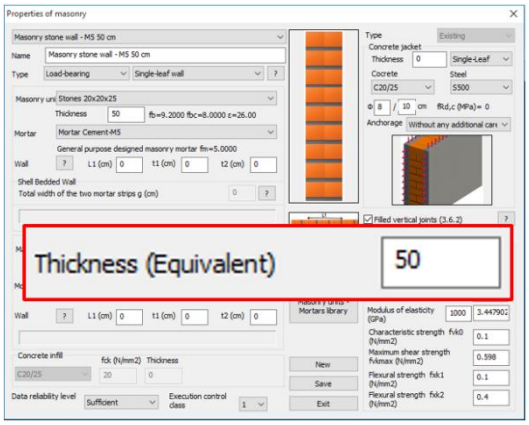


After completing the process for each view and each opening, click the OK button.

- 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.

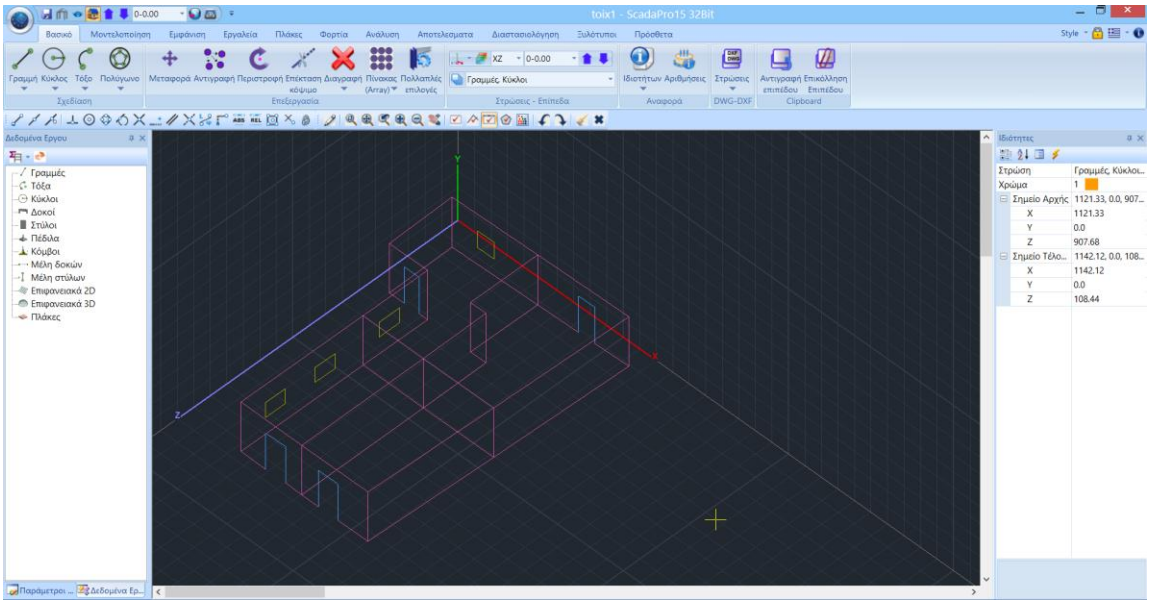
- WARNING: Make sure that the Equivalent Thickness of the wall defined to the library has the same value as the Thickness defined in the Templates.



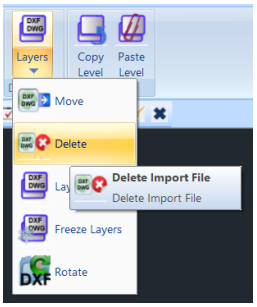


- WARNING: In the templates field you can define a single value of thickness for all walls. To edit the thickness of some walls, you open the “Plate Elements Creation” form and you modify the values respectively.

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”.



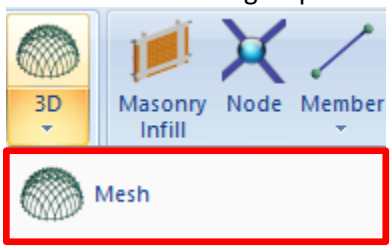
Inside SCADA environment, you can see the outlines for each view and its openings in 3D presentation.



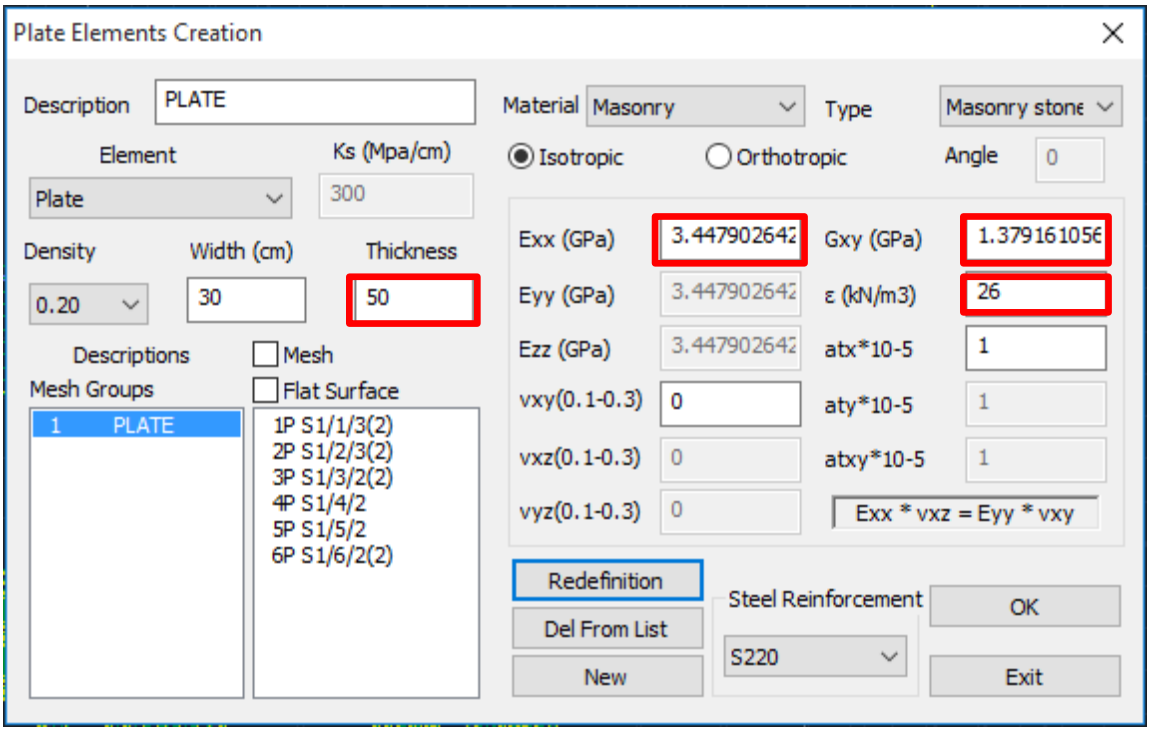
⚠ Use Delete command to cancel the auxiliary file.

1.5.3 Mesh Group Definition:

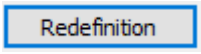
As soon as the model is imported in SCADA environment, select the 3D "Mesh" command inside "Surface Elements" group.



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.

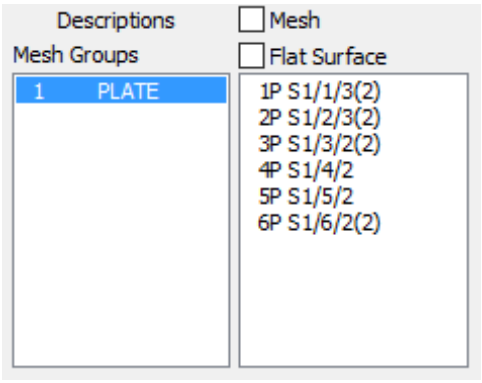


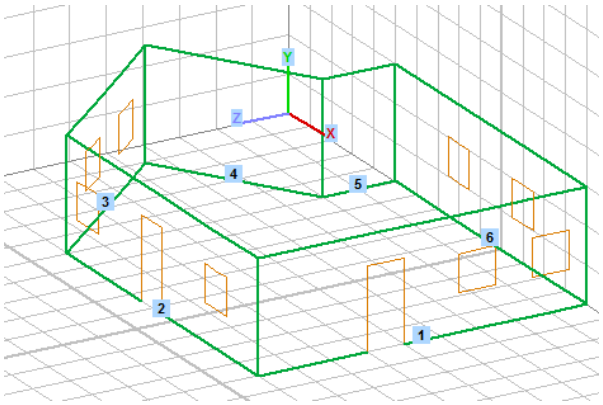
In the type, select from the list the previously defined wall from the library, and the respective fields Exx, Gxy and special weight ε are automatically updated.

Click the 

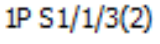
button to update the mesh and store any modifications.

- **Mesh sub-Group Definition:**





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

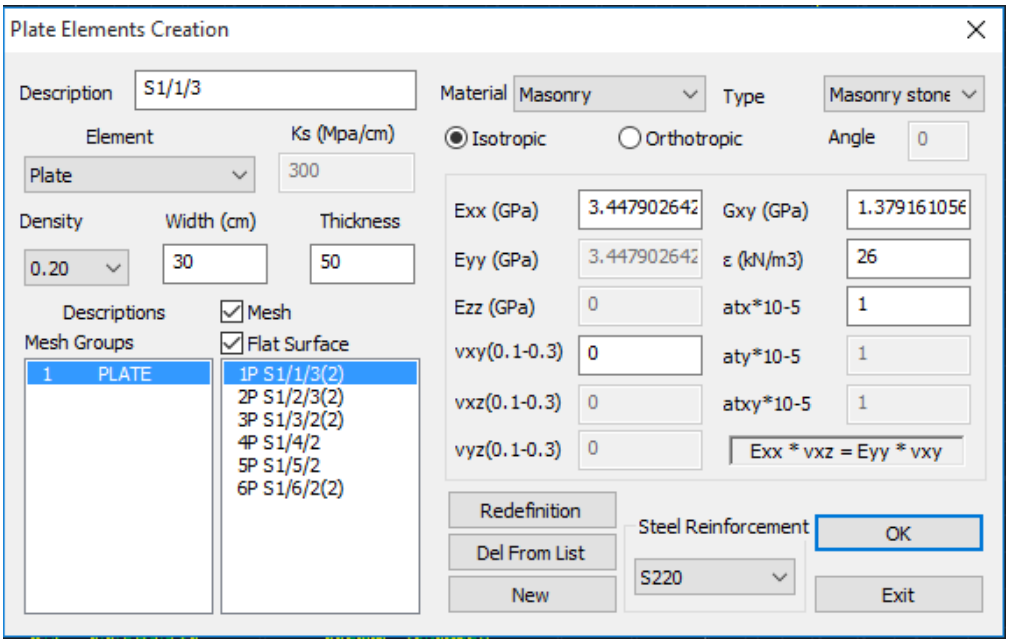
In the Surface name 

:

- 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 

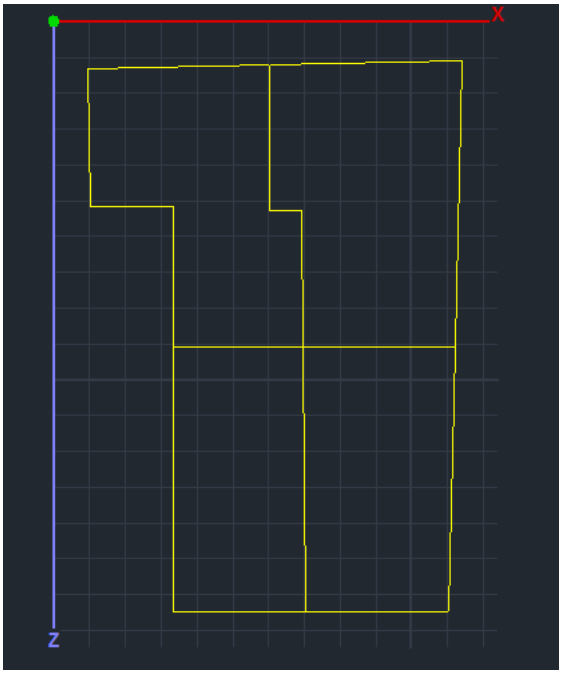
and select a surface. The fields are updated accordingly by the defined values of the selected surface,



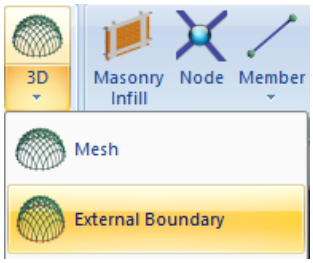
Enabling in this way the modification of any parameter (name, density, width, thickness type etc.) regarding the selected surface. Finally, click Redefinition to apply the modifications.

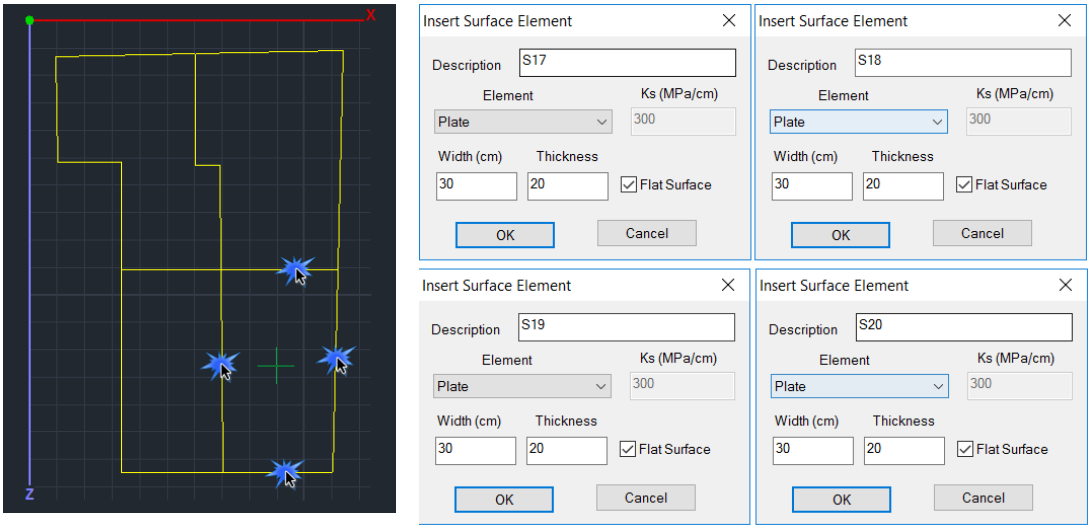
1.5.4 New Mesh sub-Group Definition for the slabs modeling

For the slabs modelling, turn to the two-dimensional display and with the help of/, display the floor plan level 1-300.00.



Then select “3D”>> “External Boundary” and left click to select the lines of the first boundary and right click to complete. Repeat this process for all four slabs:

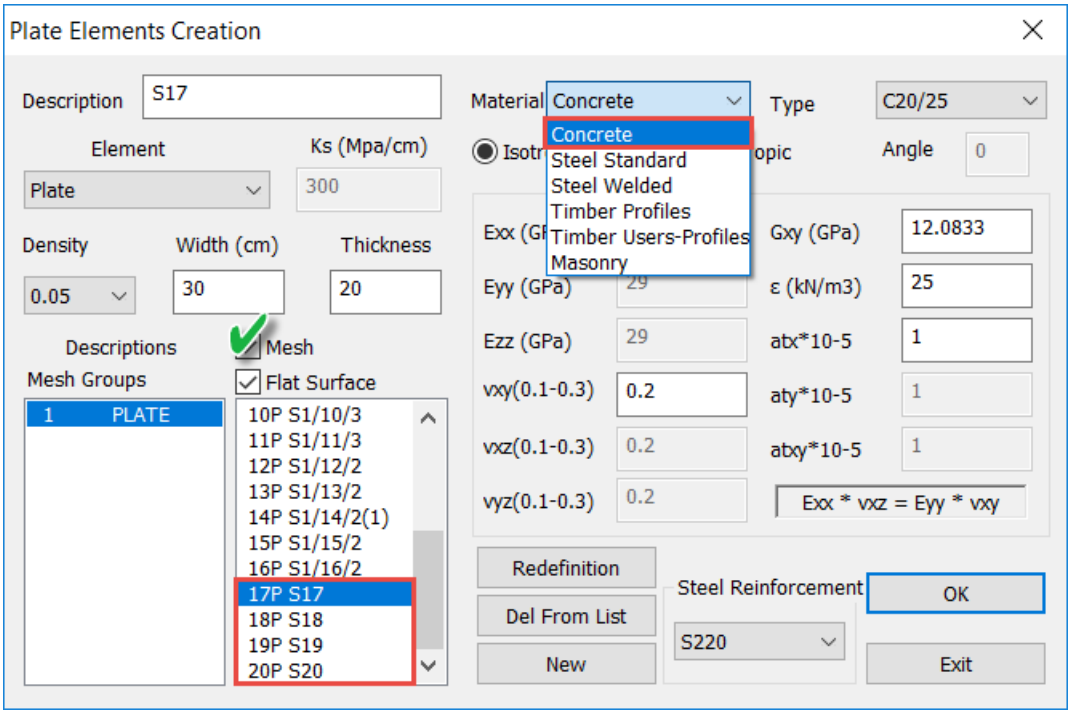




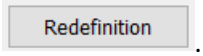
The characteristics of the first mesh subgroup are displayed in the dialog box. The active "Flat Surface" command means that the surface belongs to the level.
Set the grid parameters of each slab:
-Set, width and thickness (30, 20)
Press the OK button.

Return to 

command to see the new sub-groups "S17-20" containing in the PLATE group.

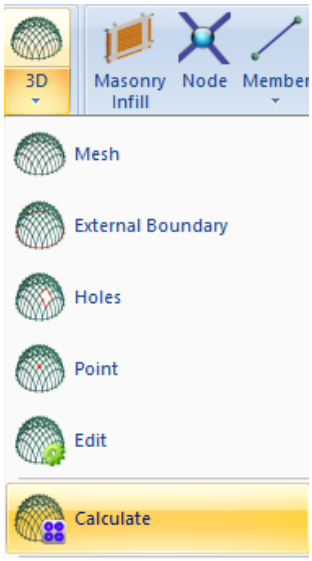


Activate 

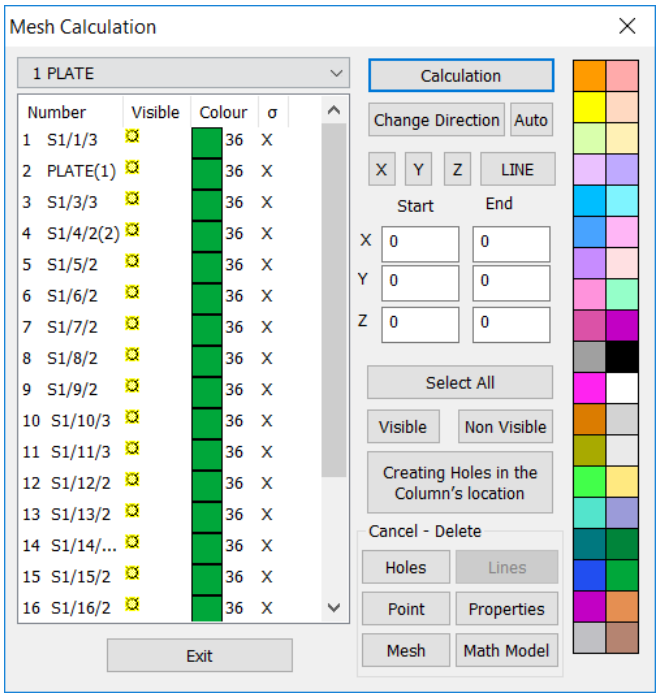
, click on each slab sub-group, select Concrete as material and press 

.

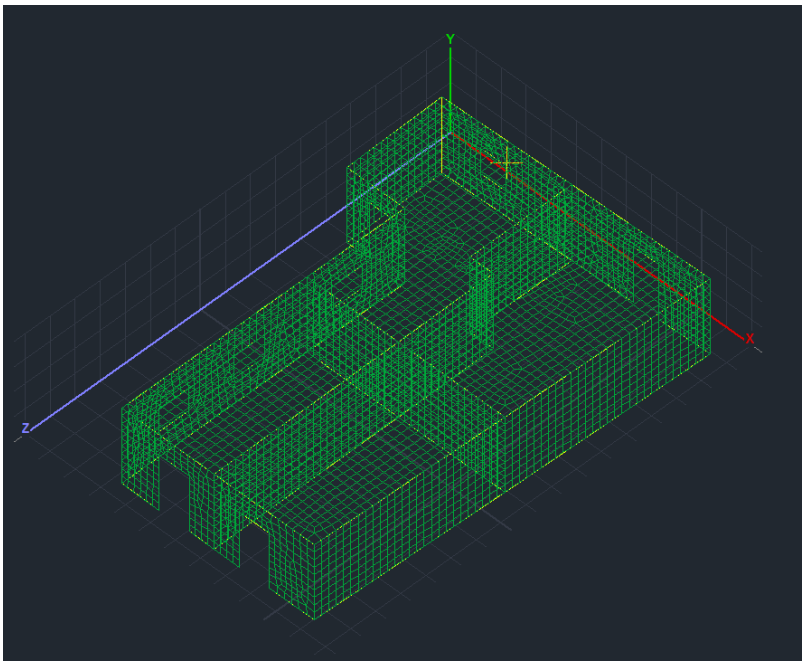
1.5.5 Surface Calculation



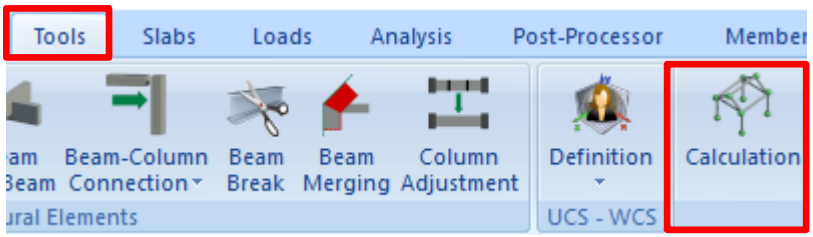
Select the Calculation command. In the dialog box that opens, the mesh list contains the 1PLATE group and its respective surfaces.



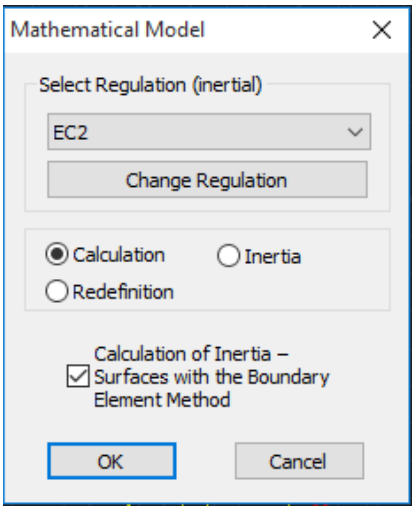
The Calculation command creates automatically the surfaces for all views.

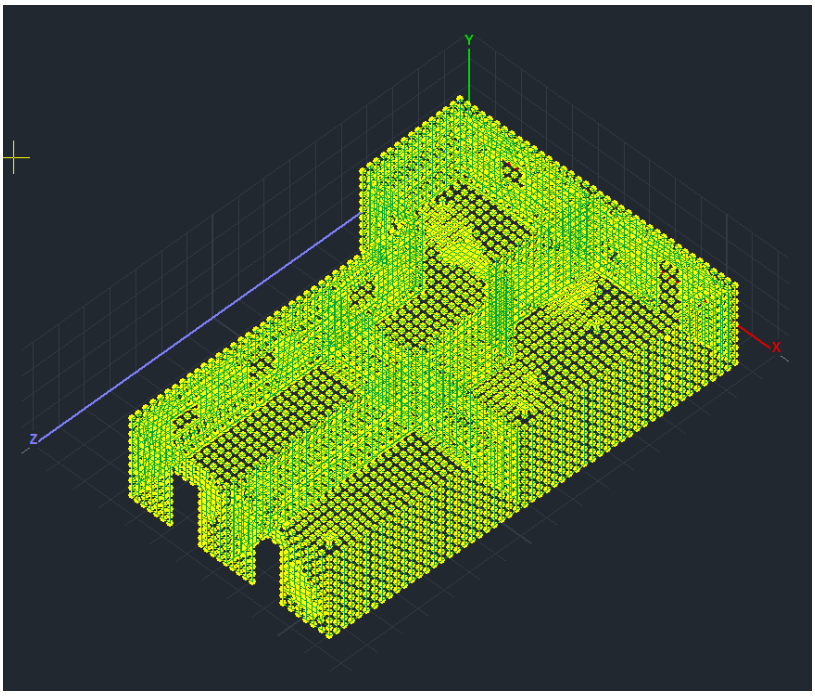


1.5.6 Mathematical Model calculation:



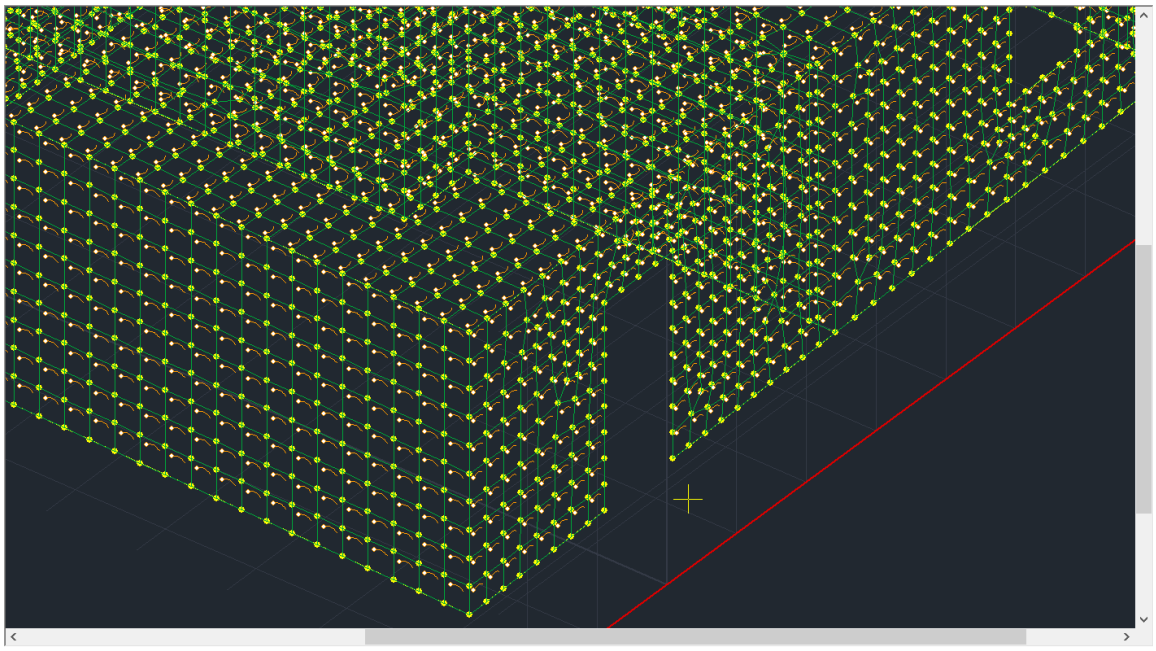
To create the mathematical model of the structure, from “Tools” unit select “Calculation” and click OK on the dialog window that opens:





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

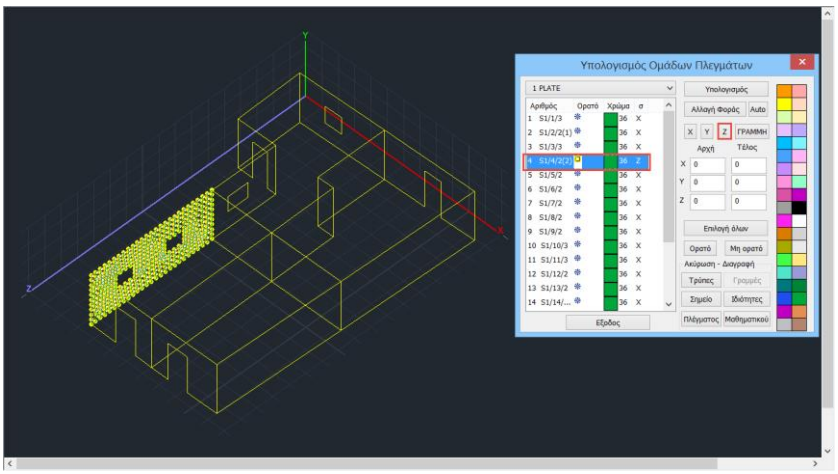
- Inside the View unit, in Switches command, activate the Local Axes option.
- Return to the "3D Mesh >> Calculation" command, and in the dialog form, select all the surfaces through the Select All command and click Auto to adjust local axes of all the surface finite elements of the plate to have the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



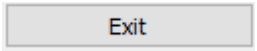
- Finally, for views of which the local axis is parallel to the global axes X or Z, select them and click or respectively, to define the main direction of the steel reinforcement (direction X or Z). For surfaces that run along X direction (vertical to Z axis) click X, while for surfaces that run along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

  - for views where the x local axis is parallel to global X, leave X
  - for views where the z local axis is parallel to global Z, press Z
  - for views that are not parallel or perpendicular to the global axes, leave X, because the main reinforcement direction is automatically defined.

In this example:



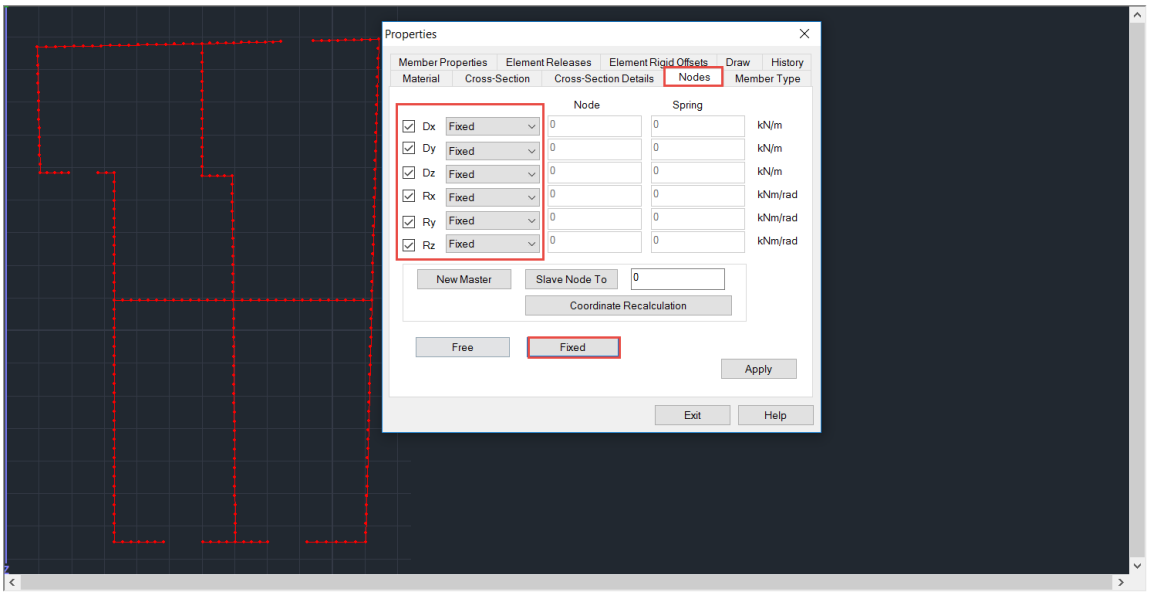
Αριθμός	Ορατό	Χρώμα	σ
1 S1/1/3		36	X
2 S1/2/2(1)		36	X
3 S1/3/3		36	X
4 S1/4/2(2)		36	Z
5 S1/5/2		36	Z
6 S1/6/2		36	X
7 S1/7/2		36	Z
8 S1/8/2		36	X
9 S1/9/2		36	X
10 S1/10/3		36	X
11 S1/11/3		36	X
12 S1/12/2		36	Z
13 S1/13/2		36	Z
14 S1/14/...		36	Z
15 S1/15/2		36	Z
16 S1/16/2		36	Z
17 S17		36	X
18 S18		36	X
19 S19		36	X
20 S20		36	X

- Click 

to apply the modifications and close the window.

Finally, for this example, we want to consider it **Fixed** at its base. So, using the command 

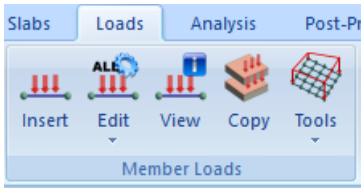
and the window selection  , select all nodes of the foundation level and **Fixed** them.



Loads

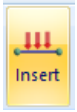
2. STEP: LOADS DEFINITION

2.1 Manually imported:

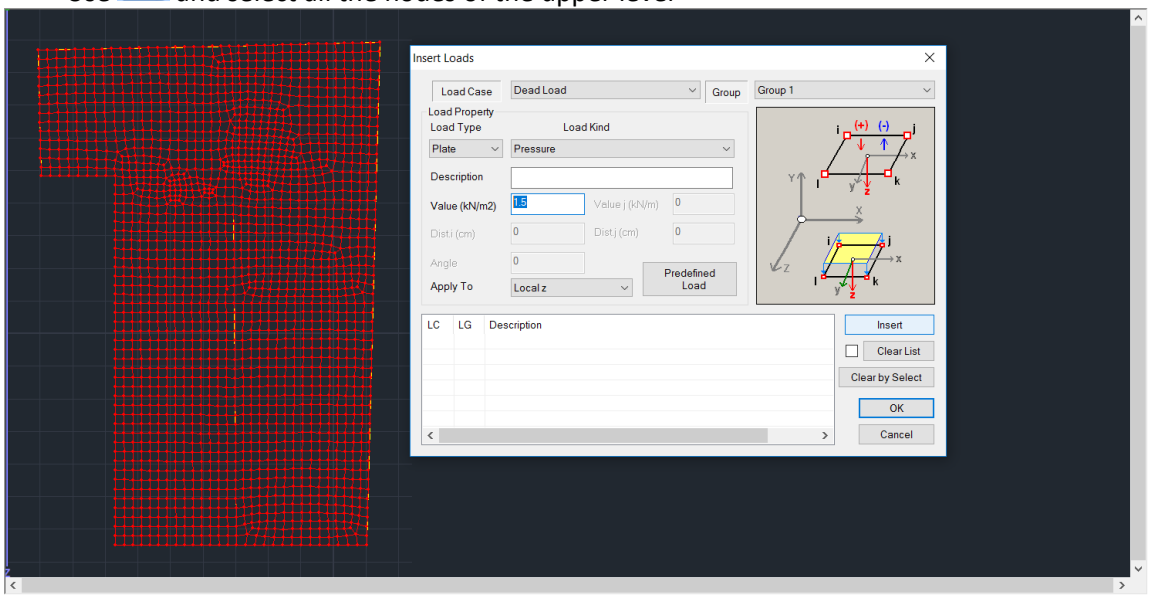


"Member Loads" commands' group contains the commands for insert, edit, view and copy the loads of members, nodes and surfaces finite elements.

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:

- Select the command 

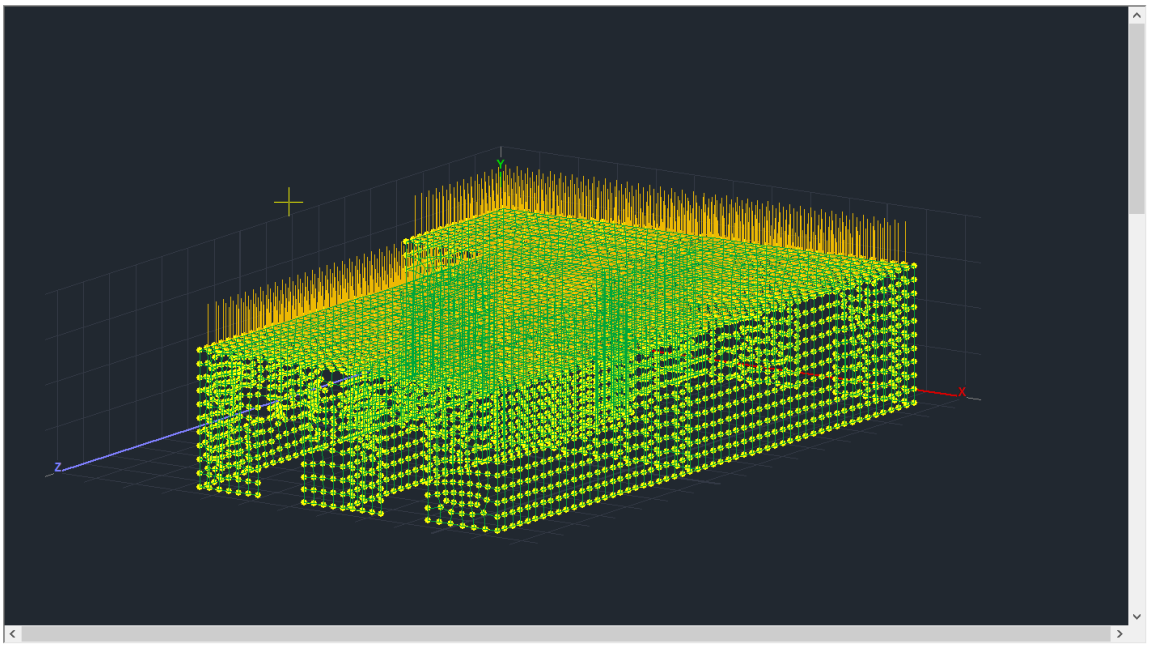
- Use  and select all the nodes of the upper level



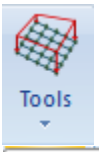
- Right click and in the dialog form:
  Select: Dead - Plate, Pressure,
  Type: 1.5 KN/M2
  Press: Insert
  then
  Select: Live - Plate, Pressure,
  Type: 2 KN/M2
  Press: Insert
  Click: OK to apply the defined loads

- Select 

to display the loads:



2.2 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.

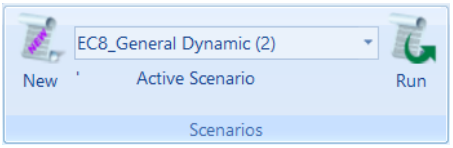
Analytical description on how to use this command can be found in chapter 6 “LOADS”.

Analysis

3. STEP: ANALYSIS

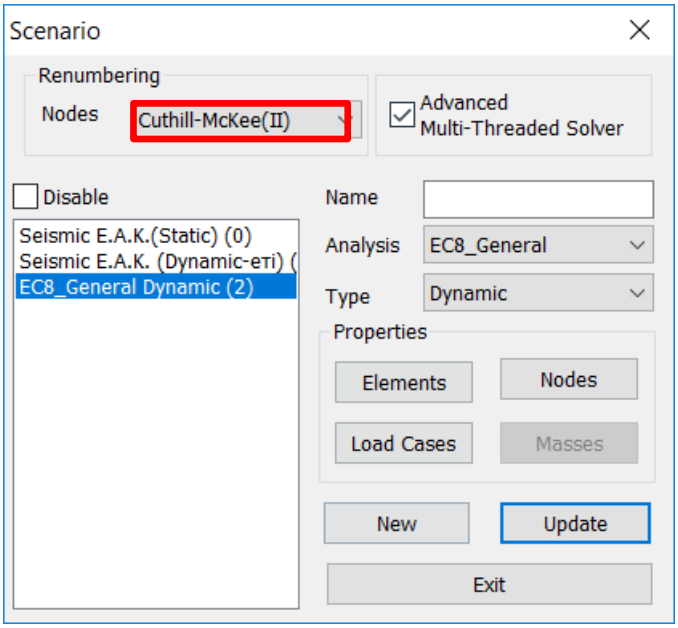
3.1 Masonry structure analysis by Eurocode

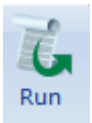
As soon as you complete the modeling and load definition processes, move on to analysis. For masonry structures analysis, create a Eurocode analysis scenario, so that SCADA Pro will perform the analysis by the provisions of the Eurocodes.



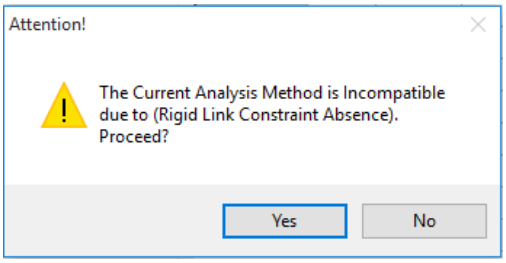
Move to "Analysis" unit and from the "Scenarios" command group, click "New" to create an Eurocode scenario for masonry structures analysis.

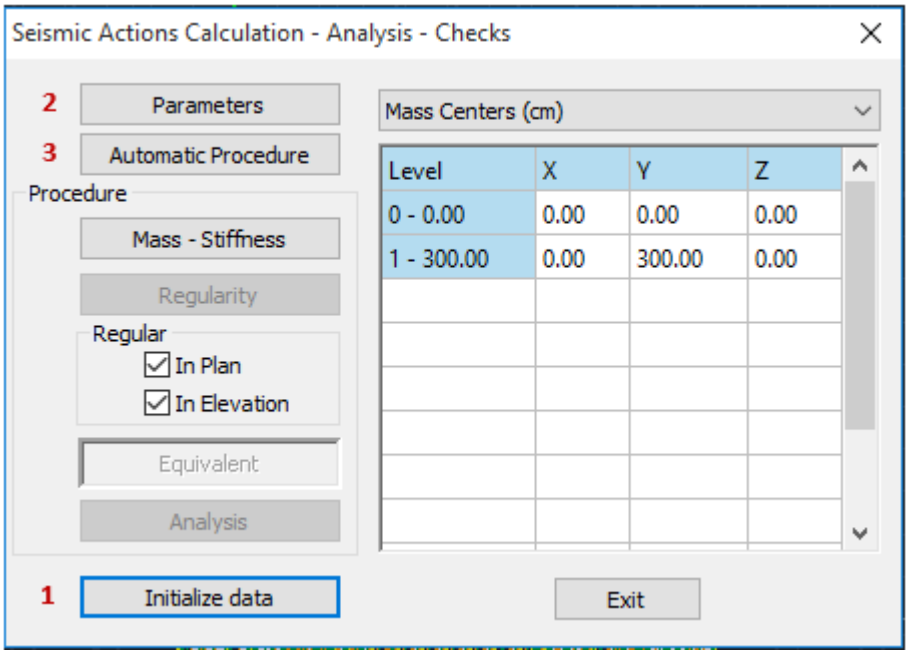
Click "New" and in the dialog window that opens:
-select Nodes Renumbering according to Cuthill-McKee(II) method
-create a new scenario by selecting the type and the respective annex -> EC-8_ Dynamic

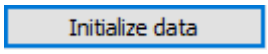


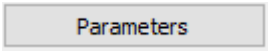
-select from the list the Eurocode scenario and click 

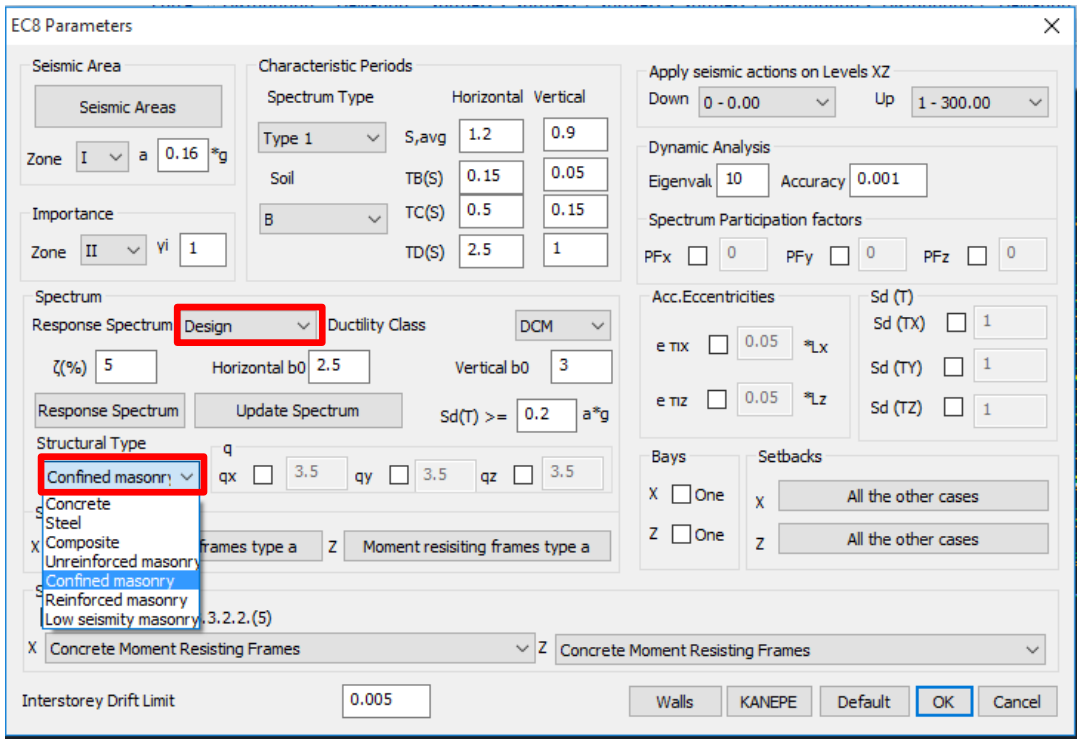
.
In the dialog box that opens, accept the warning regarding the diaphragm absence and click:





One  to update the parameters of the current scenario

Two  to define the analysis parameters

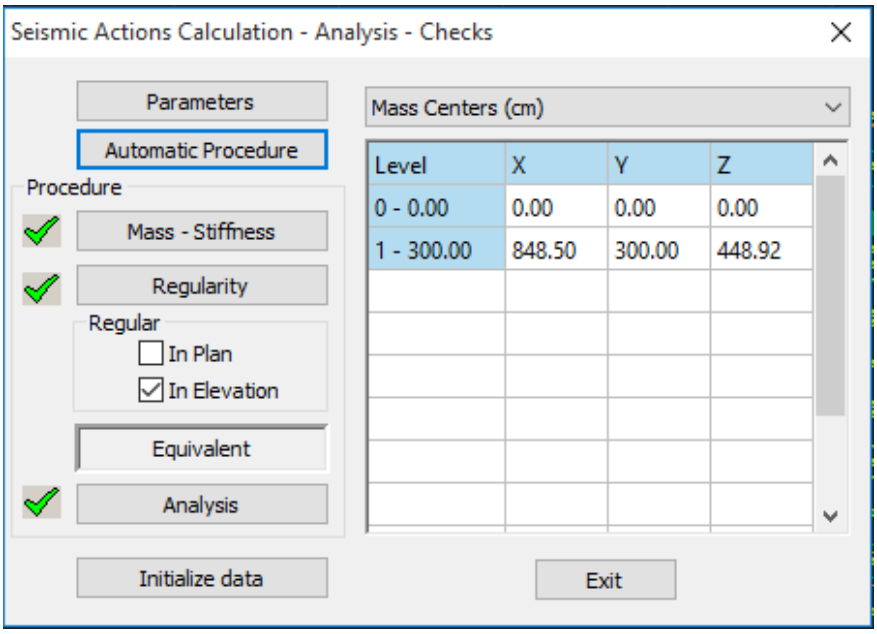


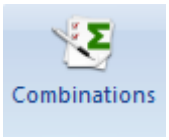
-Define “Zone”, “Importance” and “Soil”.
-select “Design” spectrum and

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

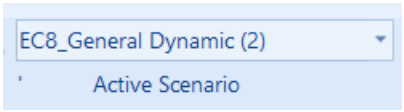
Three  to run the analysis.

Let the program to complete the process and click Exit.



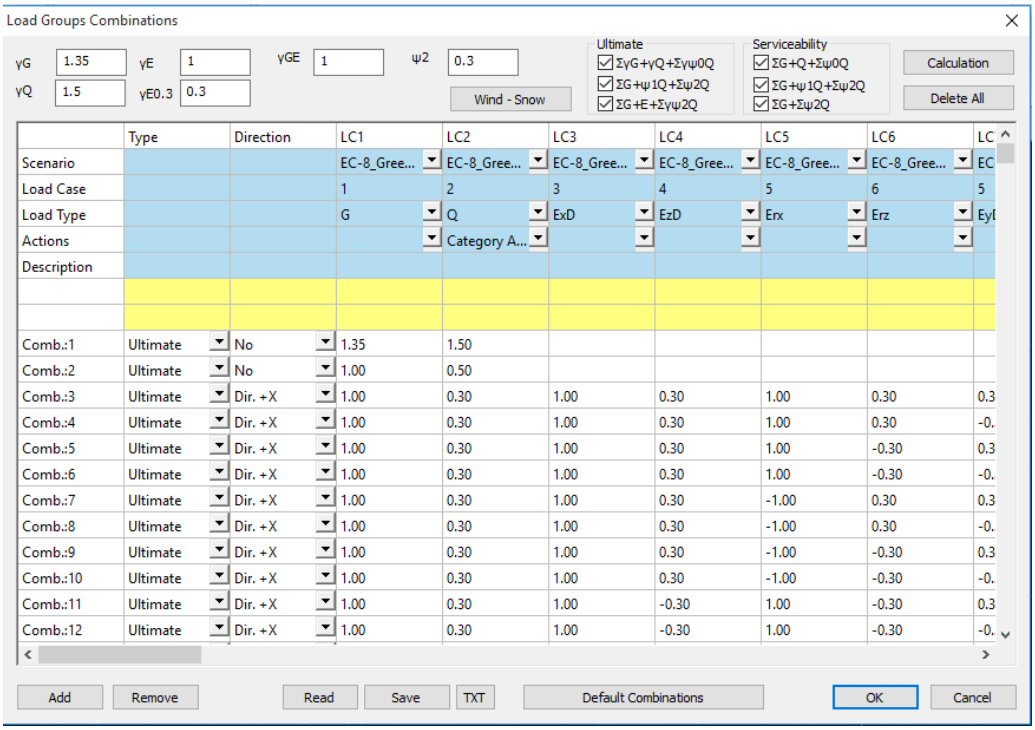


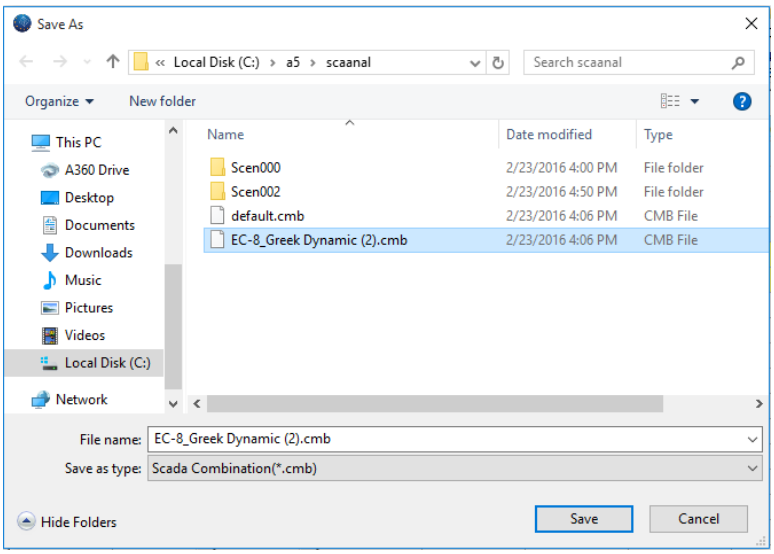
3.2 Combinations



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.



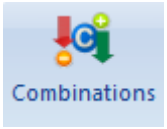


Post-Processor

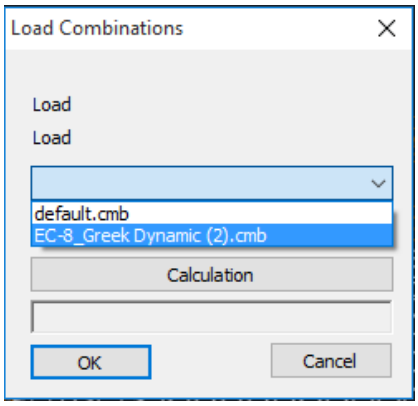
4. STEP: RESULTS

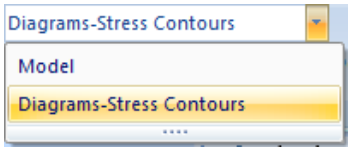
4.1 Deformed shape of Model:

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

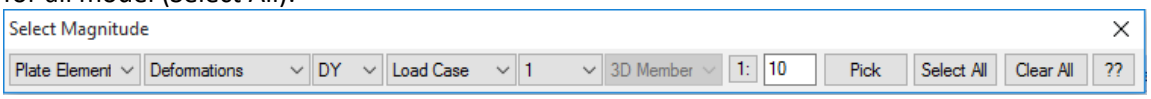
Select 

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

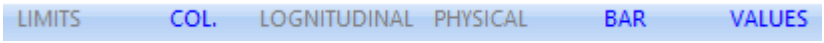


Select from the list 

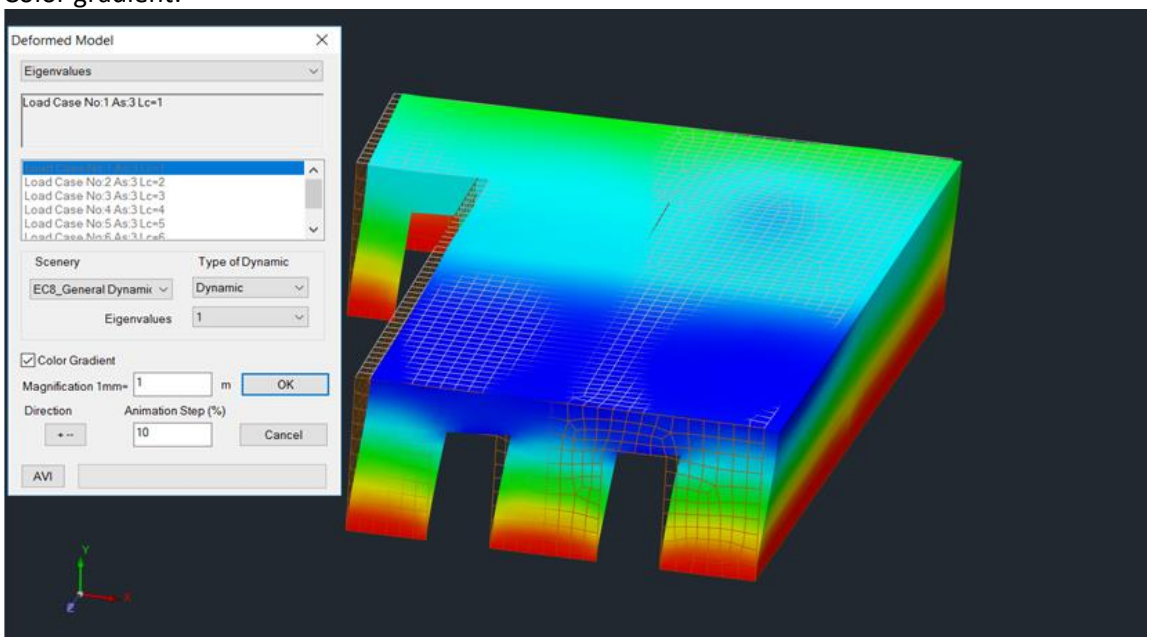
and in the dialog window, select to view “Plate Elements” -> “DY Deformations” caused by “Load Case 1” for all model (Select All):



At the bottom bar activate:



Or, select to see the Deformed Model by Eigenvalues, choosing the Dynamic scenario and the Color gradient:

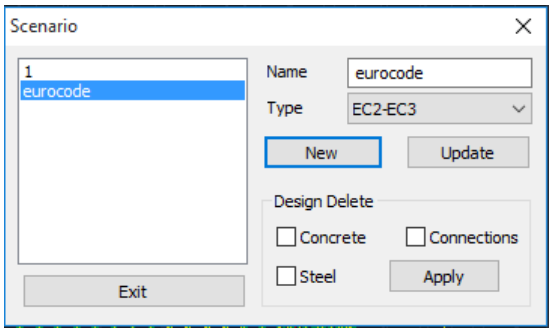


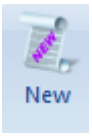
Members Design

5. STEP: MEMBERS DESIGN

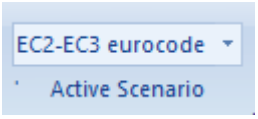
5.1 Design Scenario Creation in accordance to Eurocode provisions:

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.



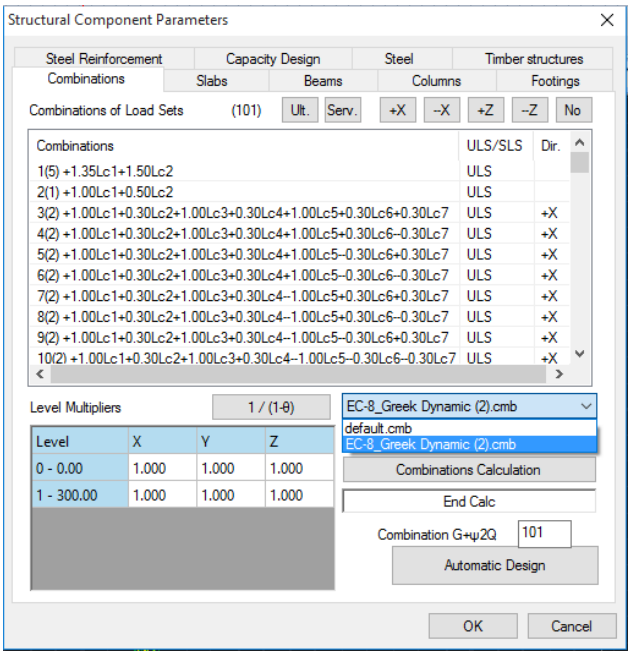
Move to “Design” unit and click 

to create the desired scenario by selecting EC2. Enter a name and click “New”.

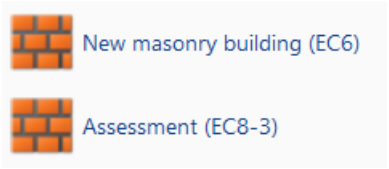
Select the considered scenario 

and and click 

. Insert Combinations that opens the folder with the registered .cmb files. Select the file and press Combinations Calculation. The program calculates the combinations and by clicking OK the window closes.



5.2 Masonry structure checks according to Eurocode



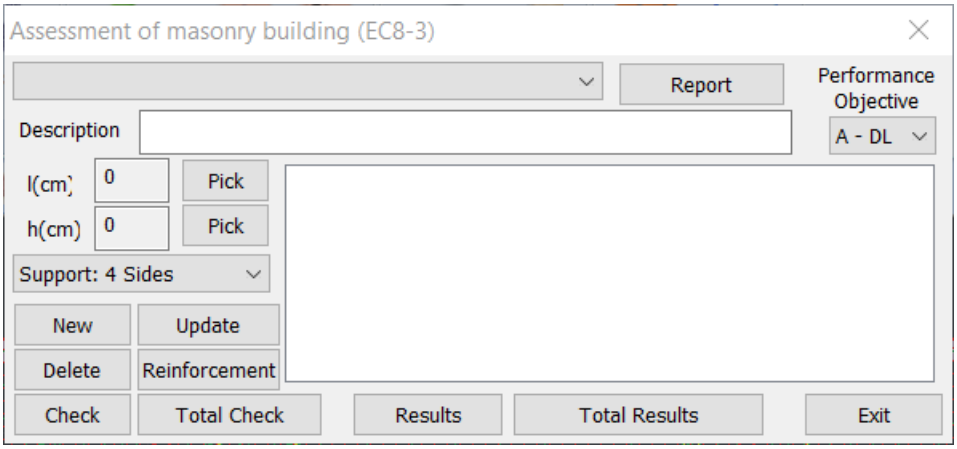
SCADA Pro implements the provisions of EC8-3 for assessing masonry buildings under seismic loading. The recommendations of the regulation applied to masonry that resist lateral forces within their level. These are both pier and spandrel of a wall.
The checks are made for each section of pier/spandrel, where the predominant intensive size is either:

- Axial force and bending, either
- Shear force

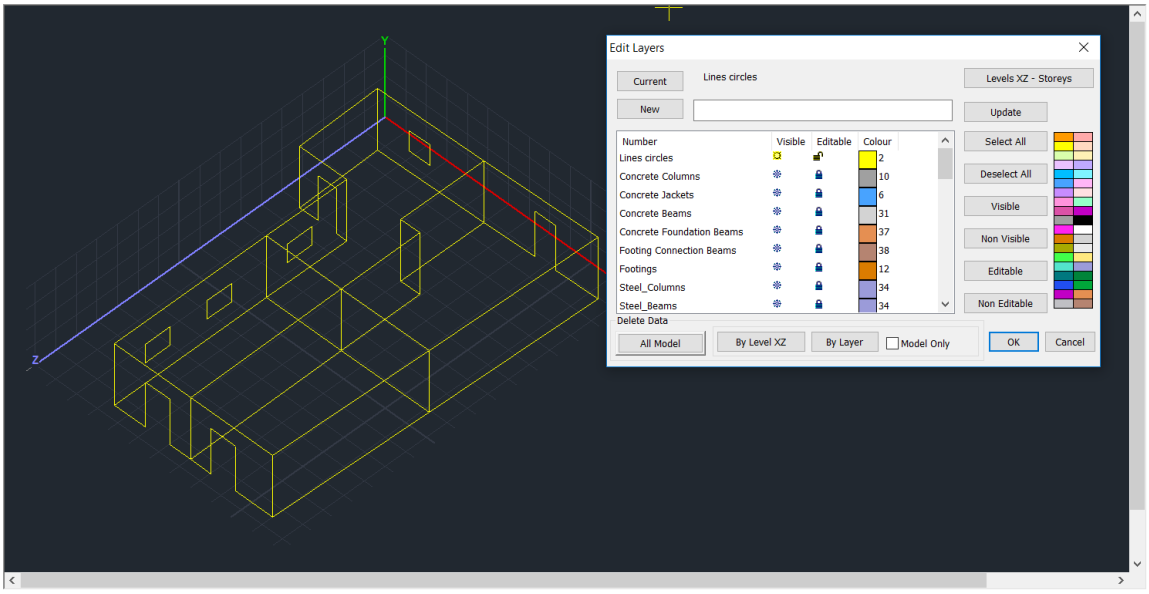
Resulting the critical failure of the wall element, is calculating respectively the bearing capacity for all three Performance Objective A, B and C.

Select the command 

In the dialog box that opens, the user must identify the parts of the walls such as described in "New masonry building".

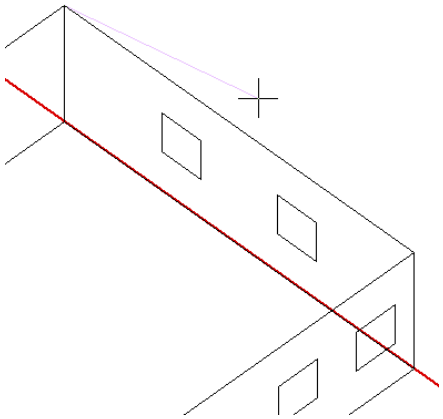


⚠ In order to pick the points easily, take advantage of the object snap utility in a most efficient way by deactivating any layers that “confuse” the picking procedure (e.g. when a whole wall is to be picked, keep active only the layers of “Lines circles” to pick the corner points of the wall).

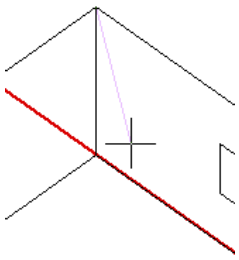


Define the walls typing one description (at least four characters and numbers) and press "New".

Use these fields to define the geometry of the considered wall: 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”.

Note that:

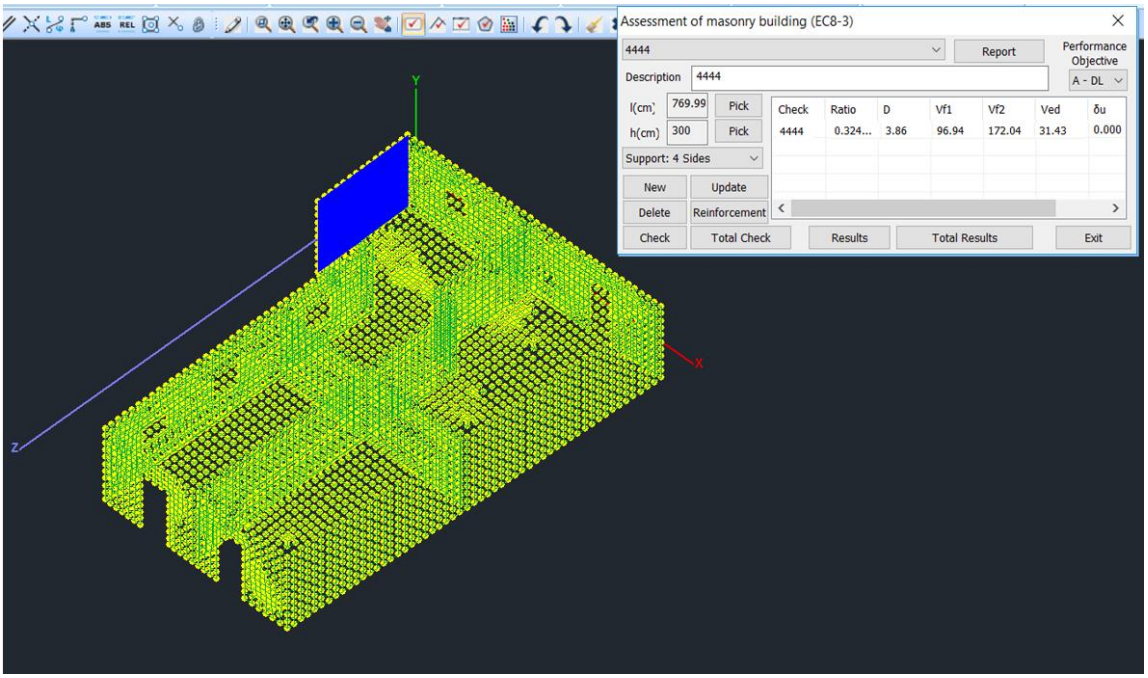
- ⚠ The identification of pier/spandrel comes automatically by the program. Means that the user defines the whole wall including the openings and the program checks automatically separating the piers and the spandrels (means the wall portions above and below the openings)

Complete the definition of all the building's walls. Then select Performance Level/.

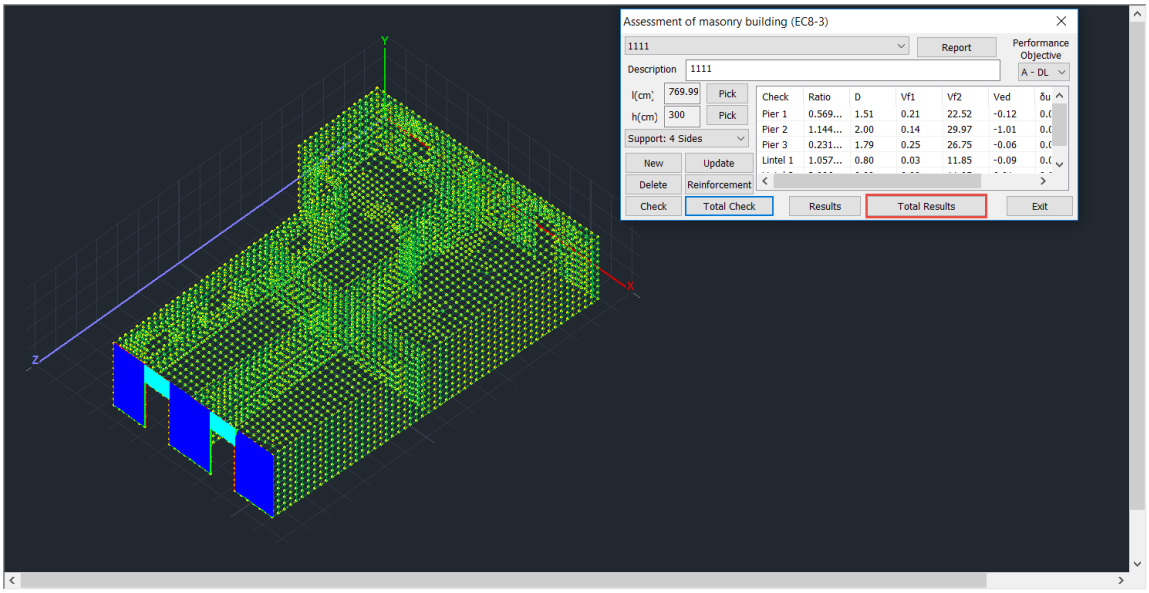
Press Check to check each section of pier/spandrel of the selected wall.

Total Check command makes checks for each section of pier/spandrel of all the defined walls.

⚠ Pier and Spandrel strength checks are made regarding forces and deformations, for each section, depending on the Performance Level.



After completing their strength checks regarding forces and deformations depending on the Performance Level, display the "Results" for each wall or through the "Total Results", all of the walls.



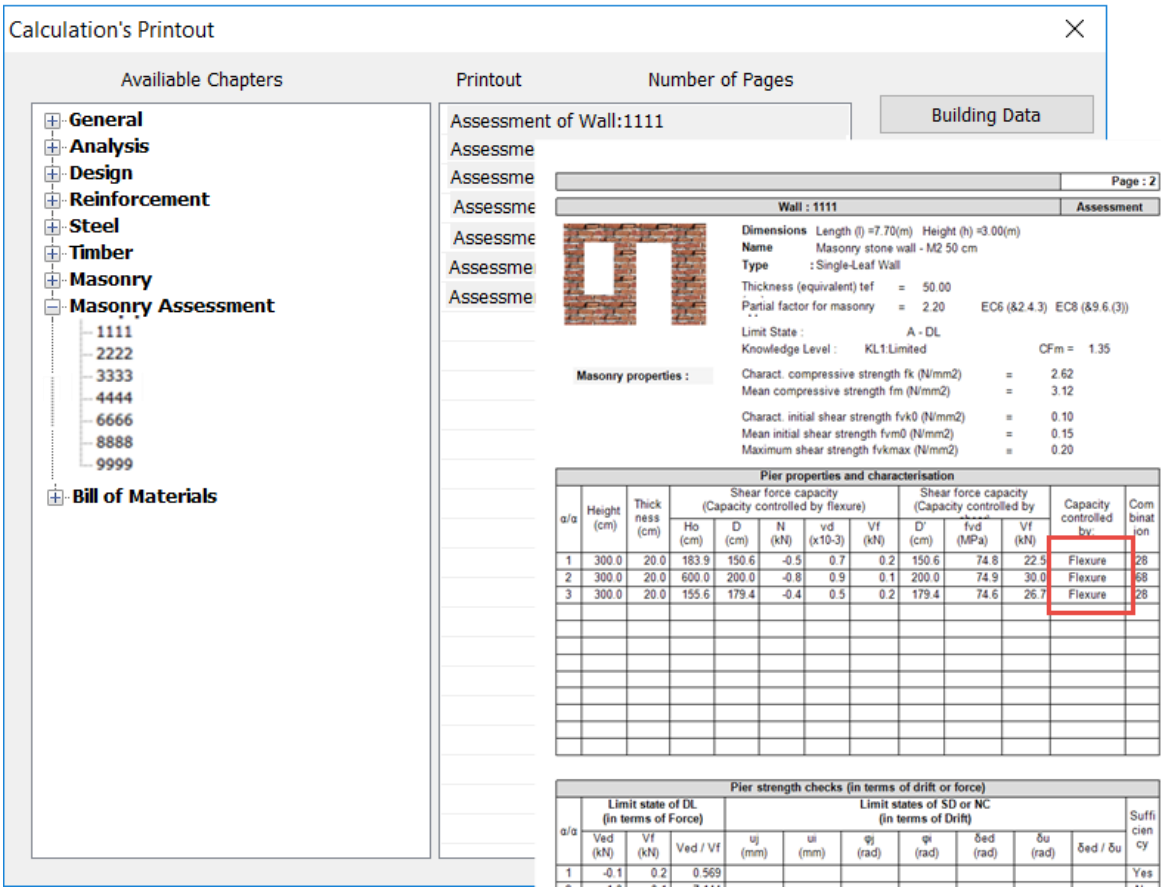
In “Ratio” show the adequacy ratios, where rises the possible need to reinforce.

6. STEP: STRENGTHENING

SCADA Pro offers the possibility of strengthening the masonry with:

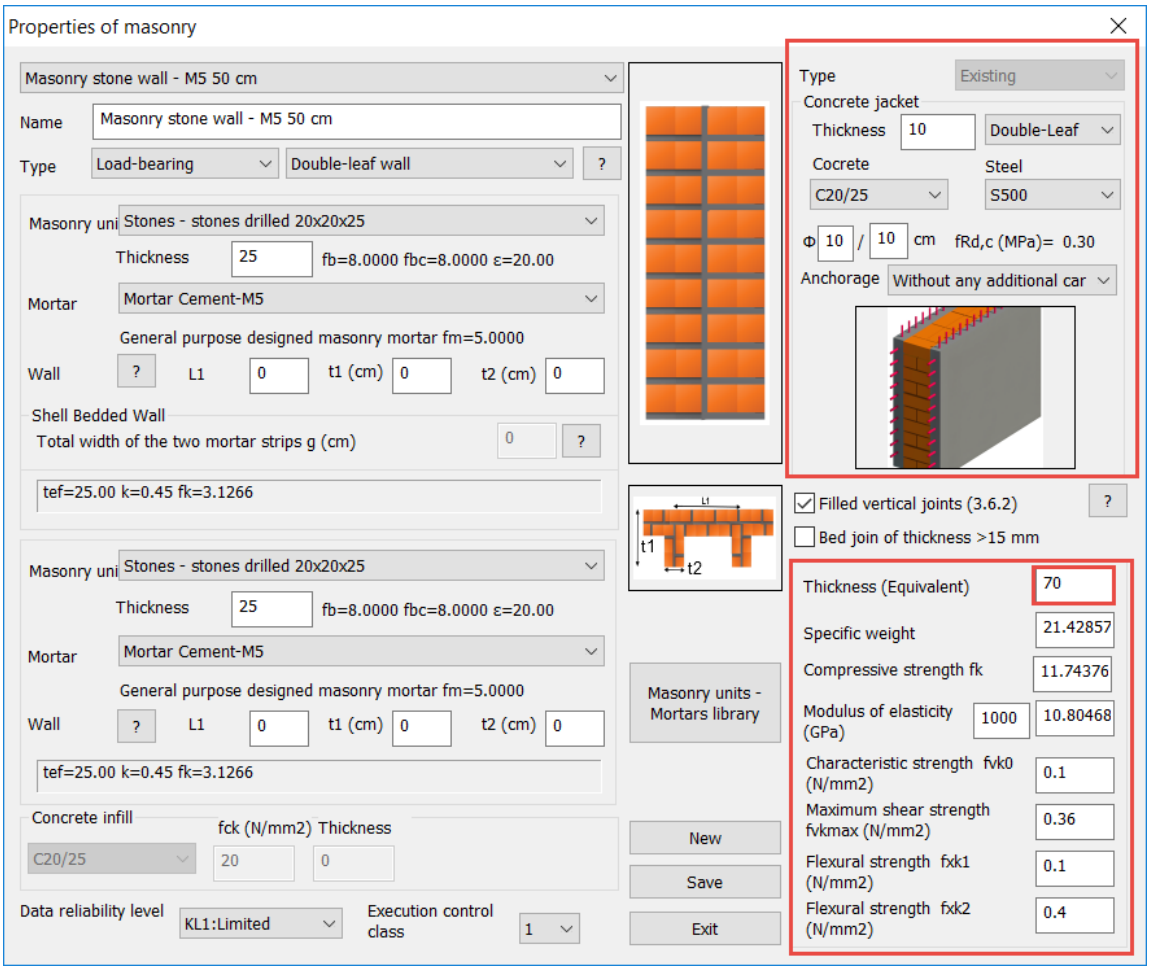
- simple or double of reinforced concrete jacket to increase the compressive and the in-plane shear and flexural strength of the element
- Textile Reinforced Mortar for in-plane shear strength

After completing the checks through the "Masonry Assessment" printing files, you can read the characterization of the failure and strengthen accordingly.

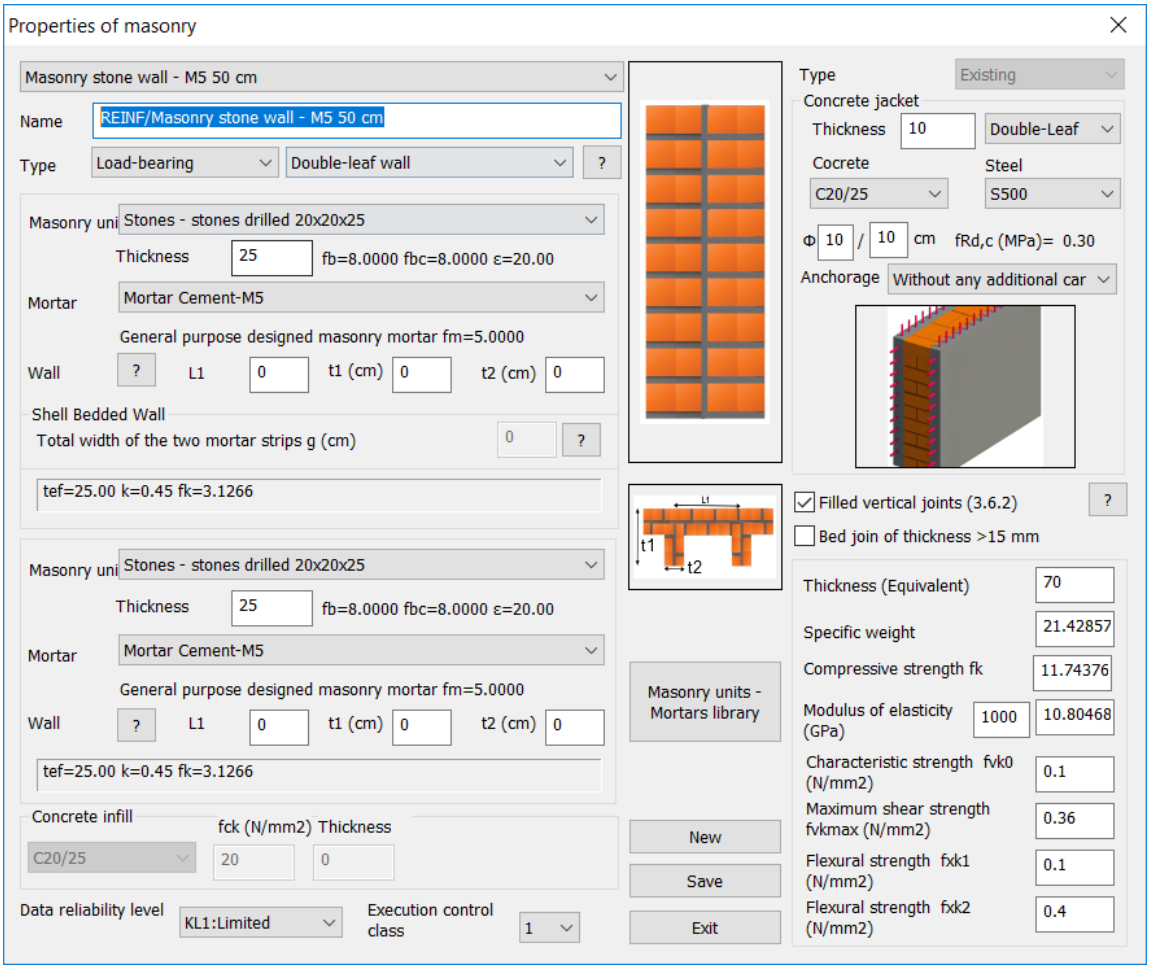


6.1 Strengthening with concrete jacket

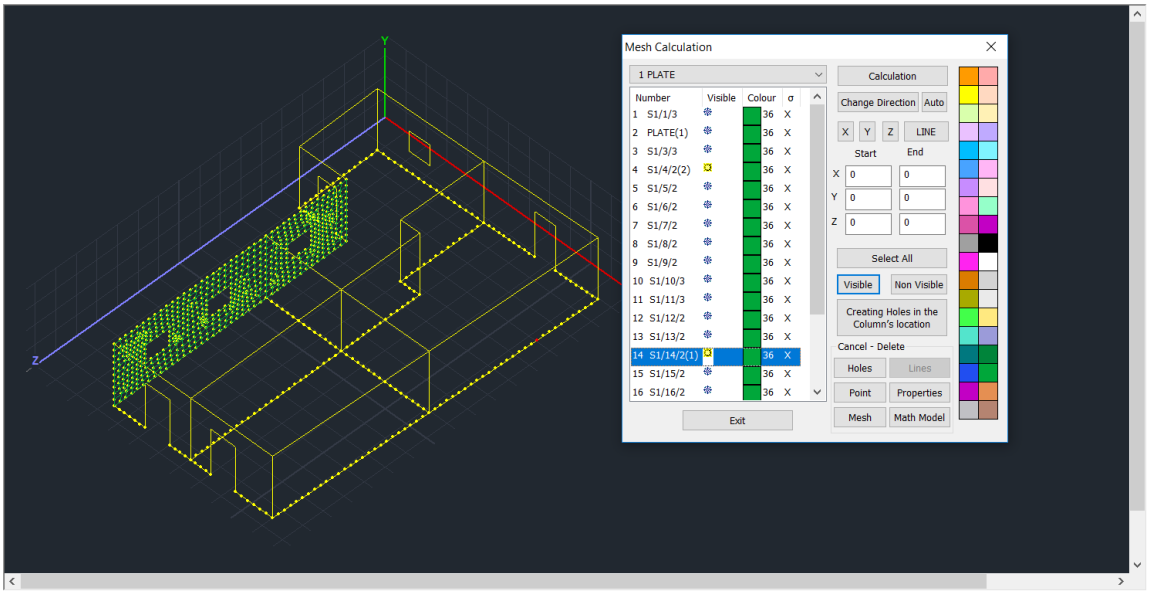
To strengthen a wall with single or double jacket, in “Masonry” Library define the characteristics for the concrete jacket. Automatically change all the characteristics of the initial wall also.



Set a new name for the strengthened element and save it for using it later, defining the reinforced wall.

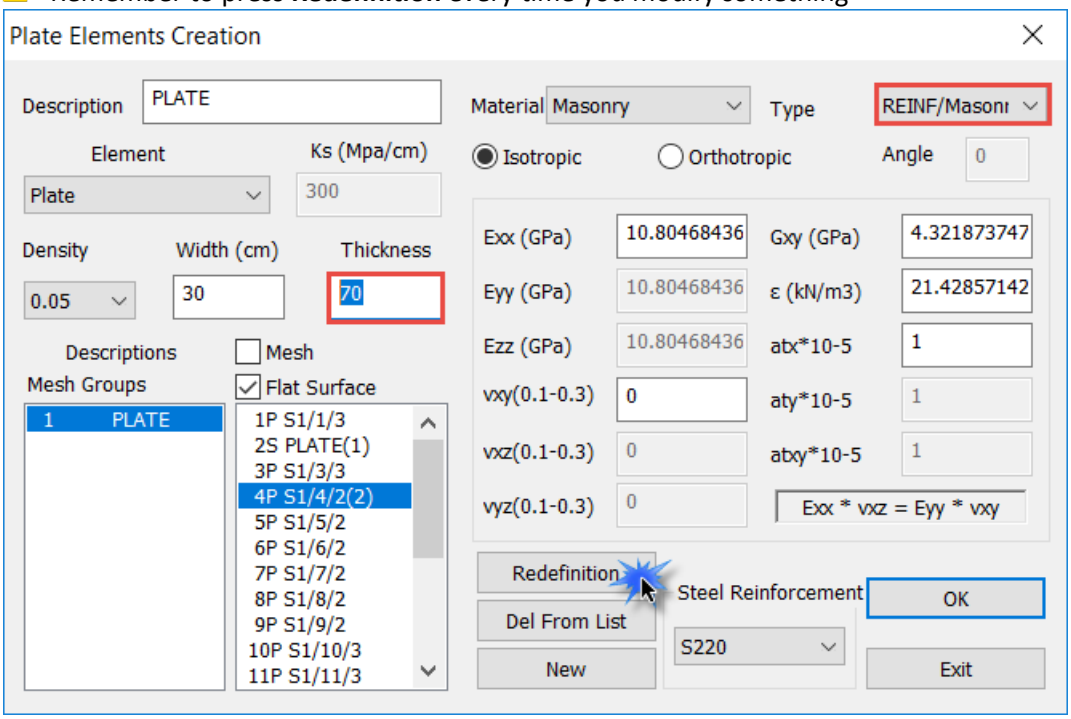


Open Mess 3D command and Calculation to identify the sub-groups that need reinforcement:



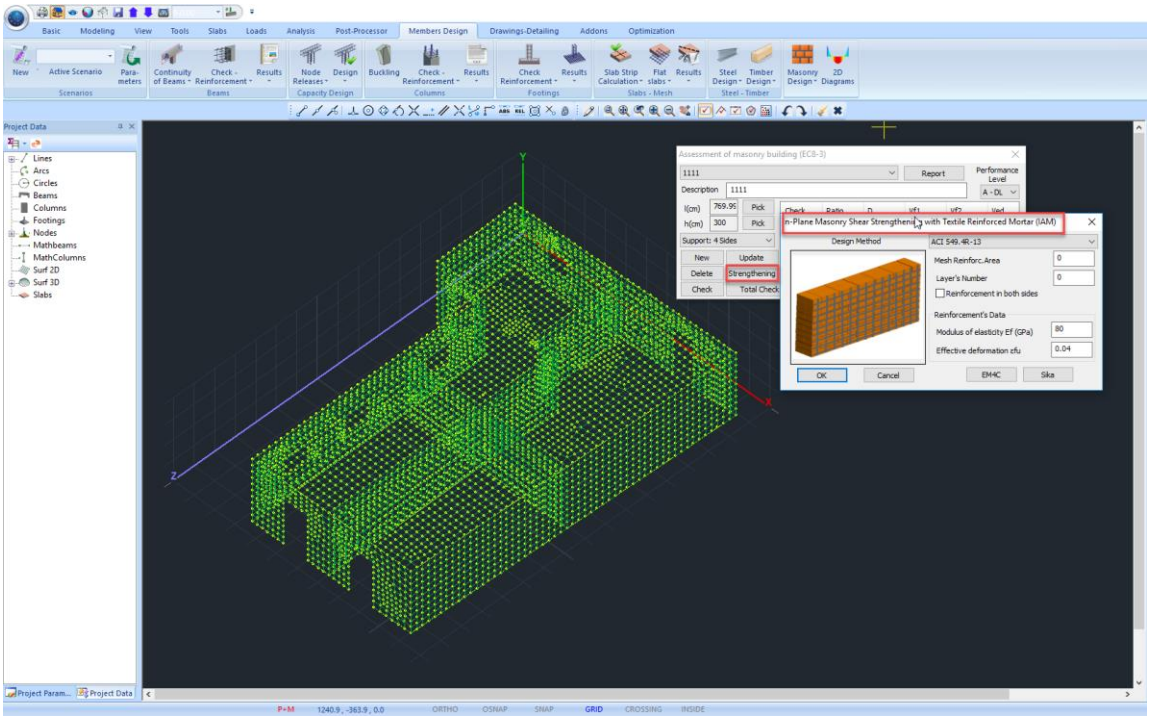
Then inside the window of the mesh group identify the sub-groups and modify their **quality** and **thickness:**

⚠ Remember to press **Redefinition** every time you modify something



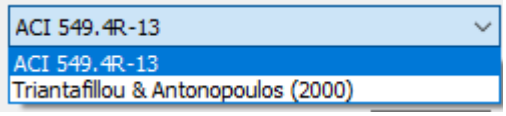
Then, repeat the analysis process, updating with new data, and check again the reinforced wall to receive new adequacy ratios, until you manage to get ratios smaller than one. The process is iterative and could be done repeatedly.

6.2 Textile Reinforced Mortar (TRM)



Use Textile Reinforced Mortar for in-plane shear strengthening, defined by the corresponding window for the selected wall from the list.

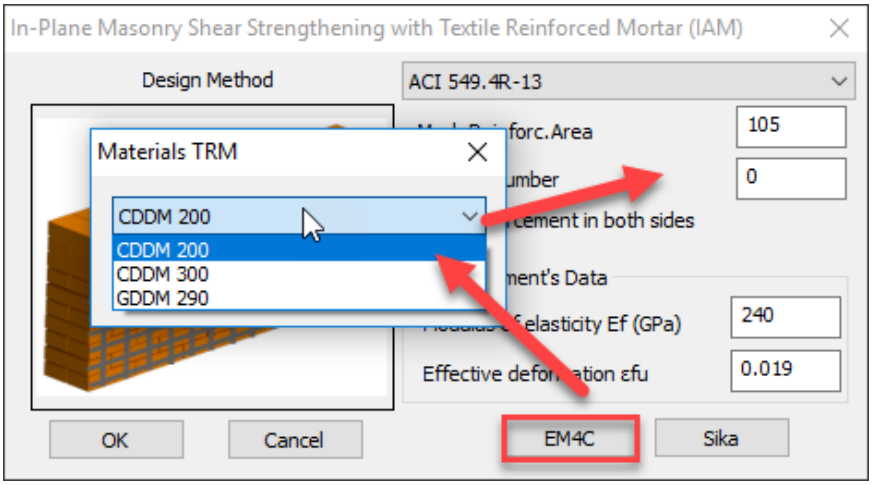
Select the “Design Method”.
To SCADA Pro contains two methods and you can select between

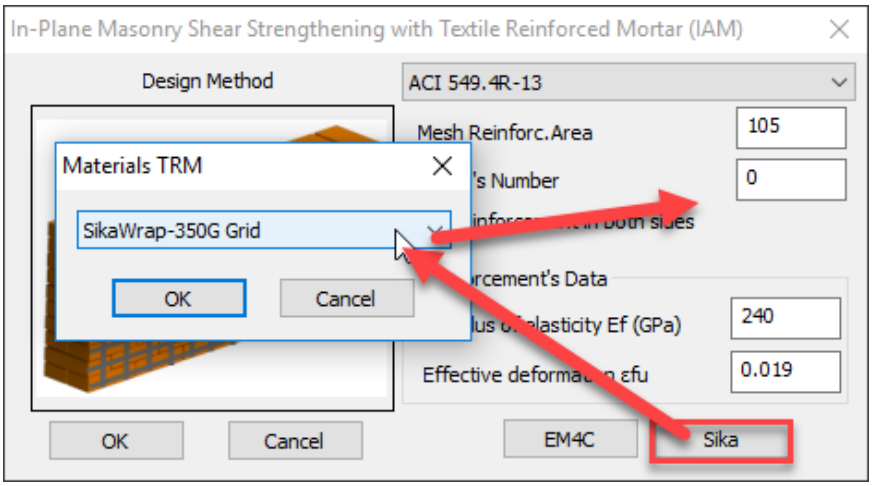


Specify the characteristics of the material, based on catalogs and commercial materials.

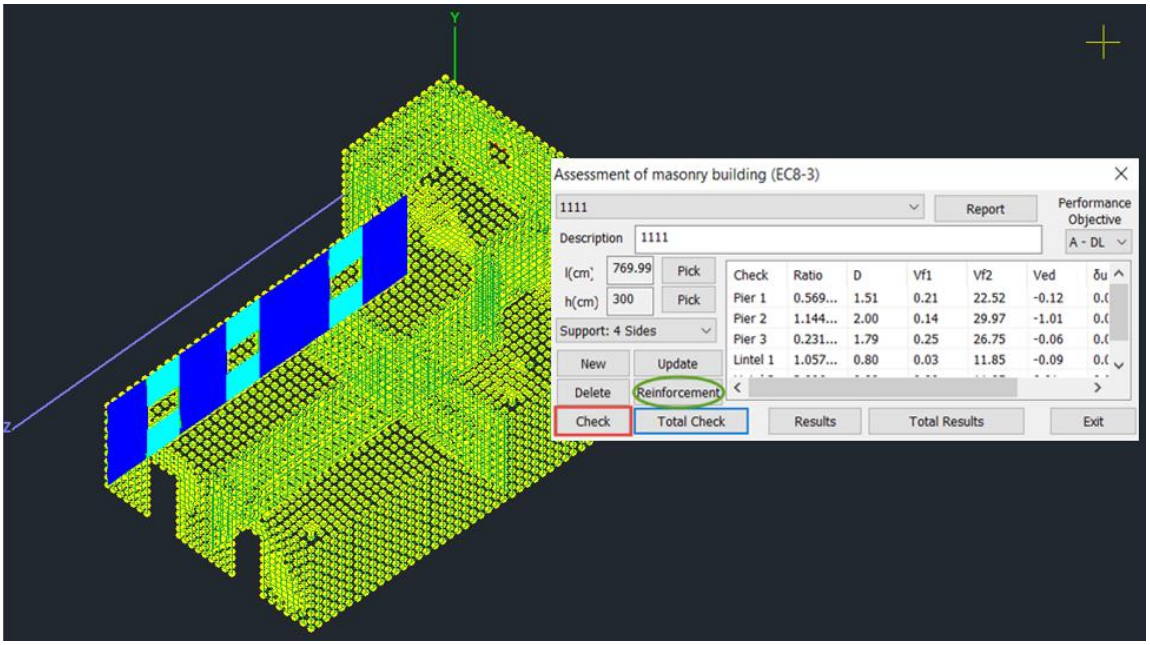
⚠ In SCADA Pro commercial materials have been introduced 

By selecting the company and the corresponding material the mesh properties are automatically filled in by the program.





Then press again the "Checks" button and check the results obtained after the introduction of the TRM. The software check under shear only the walls and the spandrels that initially failed under shear. Additionally, the capacity under flexure is rechecked to ensure the sufficiency of the elements now strengthened for shear.
You can repeat the process until sufficiency is reached.


Assessment of masonry building (EC8-3)
1111
Report
Performance Objective
Description
1111
A - DL
l(cm)
769.99
Pick
h(cm)
300
Pick
Support: 4 Sides
New
Update
Delete
Reinforcement
Check
Total Check
Results
Total Results
Exit
Check Ratio D Vf1 Vf2 Ved
Pier 1 0.569... 1.51 0.21 22.52 -0.12
Pier 2 1.144... 2.00 0.14 29.97 -1.01
Pier 3 0.231... 1.79 0.25 26.75 -0.06
Lintel 1 1.057... 0.80 0.03 11.85 -0.09