## Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>OVERVIEW</strong></td>
<td></td>
</tr>
<tr>
<td>defined.</td>
<td></td>
</tr>
<tr>
<td><strong>INTRODUCTION</strong></td>
<td>5</td>
</tr>
<tr>
<td><strong>THE NEW ENVIRONMENT</strong></td>
<td>5</td>
</tr>
<tr>
<td><strong>GENERAL DESCRIPTION</strong></td>
<td>7</td>
</tr>
<tr>
<td>A. Geometry</td>
<td>7</td>
</tr>
<tr>
<td>B. Materials</td>
<td>7</td>
</tr>
<tr>
<td>C. Regulations</td>
<td>7</td>
</tr>
<tr>
<td>D. Load and Analysis assumptions</td>
<td>7</td>
</tr>
<tr>
<td>E. Notes</td>
<td>8</td>
</tr>
<tr>
<td><strong>STEP 1 DATA INPUT – MODELING</strong></td>
<td>9</td>
</tr>
<tr>
<td>1.1 Masonry library – wall definition</td>
<td>10</td>
</tr>
<tr>
<td>1.1.1 Masonry units</td>
<td>12</td>
</tr>
<tr>
<td>1.1.2 Mortar</td>
<td>14</td>
</tr>
<tr>
<td>1.2 Modeling</td>
<td>17</td>
</tr>
<tr>
<td>1.2.1 Templates</td>
<td>17</td>
</tr>
<tr>
<td>1.2.2 Front View Identification:</td>
<td>18</td>
</tr>
<tr>
<td>1.3 Mesh Group Definition:</td>
<td>22</td>
</tr>
<tr>
<td>1.3.1 Mesh sub-Group Definition:</td>
<td>23</td>
</tr>
<tr>
<td>1.3.2 Raft and mesh areas external boundary definition:</td>
<td>24</td>
</tr>
<tr>
<td>1.4 Surface Calculation:</td>
<td>25</td>
</tr>
<tr>
<td>1.5 Mathematical Model calculation:</td>
<td>27</td>
</tr>
<tr>
<td><strong>STEP 2 LOADS DEFINITION</strong></td>
<td>30</td>
</tr>
<tr>
<td>2.1 Manually imported:</td>
<td>30</td>
</tr>
<tr>
<td>2.2 Load Distribution on the Surface</td>
<td>32</td>
</tr>
<tr>
<td><strong>STEP 3 ANALYSIS</strong></td>
<td>33</td>
</tr>
<tr>
<td>3.1 Masonry structure analysis in accordance with Eurocode:</td>
<td>33</td>
</tr>
<tr>
<td><strong>STEP 4 RESULTS</strong></td>
<td>37</td>
</tr>
<tr>
<td>4.1 Deformed shape of Model:</td>
<td>37</td>
</tr>
</tbody>
</table>
EXAMPLE: «MASONRY STRUCTURE ANALYSIS AND DESIGN»

STEP 5 DESIGN

5.1. Design Scenario Creation in accordance to Eurocode provisions: 39

5.2. Masonry structure checks according to Eurocode 5: 40

STEP 6 PRINTING 47
OVERVIEW

The new innovative and revolutionary SCADA Pro is a leading software for the analysis and design of structures. By incorporating 30 years of continuous research and development, and by using cutting edge technologies and having it designed based on your needs and requirements, it provides all the tools to quickly and easily create accurate, reliable and supervisory models of your structures. Using automated processes, your architectural designs are converted, with a single click, into a three-dimensional numerical model ready for analysis and design. SCADA Pro includes state of the art powerful solvers for all types of analyses (linear or nonlinear) and covers all code provisions and regulations applicable in most European countries and Saudi Building Code (SBC). It combines truss, beams 2D and 3D, plane, plate and shell finite elements in the same spatial model with unlimited number of nodes and finite elements. Tested by thousands of engineers around the world, it has been established as the most reliable, comprehensive and productive high-performance software for the analysis and design of any type of structure and for all structural materials (Reinforced Concrete, Steel, Masonry, Timber). SCADA Pro is a program that is constantly upgraded, evolves and adapts. ACE-Hellas technical department, in permanent cooperation with the Technical University of Athens, ensures the continuous development and updating. A "living organism" that matures.

INTRODUCTION

This manual is to lead the consultant engineer through his first steps in to SCADA Pro new environment. It is divided into chapters and plays the role of a simple yet powerful example-guide. Each chapter includes useful information, for the comprehension of the program commands as well as the procedure that is to be followed so that the input, analysis and check of a masonry structure can be performed.

THE NEW ENVIRONMENT

In the new environment, SCADA Pro uses the RIBBON technology for an easier access in the tools and commands of the program. The main idea of the ribbon design is the grouping of similar commands, so that the command-buttons can be located and executed faster and easier.

The user can create his own quick access toolbar and keep there the most commonly used commands. This toolbar is saved even after closing the program and you can add/remove commands through the “Customize quick access toolbar” command and even change its position.
The new SCADA Pro interface uses the TREE view. The TREE view is an "interactive" list that includes all the structural elements of the model in groups providing a detailed overview of the project; quick search and access of the data. This discretization is ideal for locating and highlighting easily any element. On the same time the current level is isolated, while on the right of the screen, all of its properties (based on their category) are listed with the ability for instant modification. This function is bidirectional i.e. it can be performed by selecting graphically on the model the current element, or by selecting it from the tree. Specific type of commands can also be performed by right clicking on the current element on the tree.

It's the list of properties that appears on the right and is filled in automatically when an existing element is selected. It provides user with information about the features of the structural element and allows model modifications.
GENERAL DESCRIPTION

A. Geometry

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

B. 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.

C. Regulations

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

D. Load and Analysis assumptions

Dynamic Spectrum Analysis with pairs of torsional moment along the same direction.
The loads in accordance with the aforementioned method 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 ±2etzi).
(6)Erz±(node torsional moments, derived from node seismic forces along ZII XI axes, offset by the accidental eccentricity ±2etxi.)
(7)EY (seismic vertical component –seismic force along y direction- derived from dynamic analysis).

**E. 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.
STEP 1 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 are able to 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.
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 and Wall2 fields define
  - Masonry units: the type and thickness
  - mortars: the type

and the corresponding factors are automatically updated

\[ fb = 3.3467 \, f_{bc} = 4.0000 \, \varepsilon = 15.00 \]
\[ tef = 9.00 \, k = 0.45 \, f_{k} = 1.6988 \]

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 or 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)

---

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

Type the “Compressive Strength” $f_{bc}$, which is the average value of experiments regarding the compressive strength of the masonry units and the “Specific Weight $\varepsilon$”.

The “Compressive Strength” $f_b$ 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.

### 1.1.2 Mortar

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

The compressive strength $F_m$ 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.
EXAMPLE: «MASONRY STRUCTURE ANALYSIS AND DESIGN»

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. $f_b = 9.2000$ $f_{fc} = 8.0000$ $\varepsilon = 26.00$

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 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 in order 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.

<table>
<thead>
<tr>
<th>Thickness (Equivalent)</th>
<th>50</th>
</tr>
</thead>
<tbody>
<tr>
<td>Specific weight (kN/m³)</td>
<td>26</td>
</tr>
<tr>
<td>Compressive strength ( f_{ck} ) (N/mm²)</td>
<td>3.447902</td>
</tr>
<tr>
<td>Modulus of elasticity (GPa)</td>
<td>1000</td>
</tr>
<tr>
<td>Characteristic strength ( f_{vk0} ) (N/mm²)</td>
<td>0.1</td>
</tr>
<tr>
<td>Maximum shear strength ( f_{vkmax} ) (N/mm²)</td>
<td>0.598</td>
</tr>
<tr>
<td>Flexural strength ( f_{vk1} ) (N/mm²)</td>
<td>0.1</td>
</tr>
<tr>
<td>Flexural strength ( f_{vk2} ) (N/mm²)</td>
<td>0.4</td>
</tr>
</tbody>
</table>
1.2 Modeling:
1.2.1 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.”

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 versus X, Z global axes in order to define the direction of the surface in the interface.

For more than one floors you can give different 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, particularly 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 versus X, Z global axes (according the drawing) counterclockwise, (ii) the length and the thickness of the wall and (iii) the number of the openings.

In a similar way, 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 next.
1.2.2 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”.

20
EXAMPLE: «MASONRY STRUCTURE ANALYSIS AND DESIGN»

⚠️ 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. In order 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.
1.3 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 $E_{xx}$, $G_{xy}$ and special weight $\varepsilon$ are automatically updated.

Click the [Redefinition] button to update the mesh and store any modifications.
1.3.1 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 [Mesh] and select a surface. The fields are updated accordingly in accordance with 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.
1.3.2 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 $K_s$ ($K_s=0.5$ MPa/cm)
- define, Width and Thickness (30, 50)
- click OK.

Return to command to see the surfaces of the “RAFT” mesh group.
1.4 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 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:

![Mathematical Model dialog window]

The image shows the mathematical model calculation window with options for selecting regulations (inertia), calculation method, and other settings. The model is displayed in 3D, representing the structure under analysis.
As soon as the mathematical model is created, the local axes and their direction (in respect with the global axes) must be redefined.

**NOTES:**

1. Inside the unit, in the command, activate the option.
2. Return to the “3D Mesh >> Calculation” command, and in the dialog form, select all the surfaces through the command and click in order to adjust local axes of all the surface finite elements of the plate in order to have the same direction.

3. Finally, for views of which the local axis is parallel to the global axes X or Z, select them and click or respectively, in order 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.

**EXAMPLE:**

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

While for views 1 and 5 the x local axis is parallel to global X.

4. Click to apply the modifications and close the window.
STEP 2 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, in order 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:
2.2. Load Distribution 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.
STEP 3 ANALYSIS

3.1. Masonry structure analysis in accordance with 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 in accordance with 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 Run.
In the dialog box that opens, accept the warning regarding the diaphragm absence and click:
- Define “Zone”, “Importance” and “Soil”.
- Select “Design” spectrum and
EXAMPLE: «MASONRY STRUCTURE ANALYSIS AND DESIGN»

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

3. Automatic Procedure 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 , in order to fill in automatically the coefficients of the dynamic analysis according to Eurocode. Click “Save”, in order to store the combinations file inside the folder of your project in order to use it later on during the “Post-Processor” and “Member Design” process.
STEP 4 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: Color representation and Values bar, to view the next image:
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 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.
5.2. Masonry structure checks according to Eurocode 6:

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 4 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 constraint and press the button “New” 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 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. ).

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 in accordance with the aforementioned checks 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.

**Warning:** 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” (per Level/Wall).

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

### Building Data

<table>
<thead>
<tr>
<th>Level</th>
<th>Lx(m)</th>
<th>Lz(m)</th>
<th>Recesses Area (m²)</th>
<th>Mass(KN/g)</th>
<th>n</th>
<th>ΣL(m)</th>
<th>Awtot(m²)</th>
<th>ΣL&gt;2m(m)</th>
<th>κ</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 - 0.00</td>
<td>x</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>5</td>
<td>10.72</td>
<td>5.36</td>
<td>4.72</td>
<td></td>
</tr>
<tr>
<td>z</td>
<td>7</td>
<td>10.02</td>
<td>5.01</td>
<td>3.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1 - 3.00</td>
<td>x</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>z</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Walls Data

<table>
<thead>
<tr>
<th>Level</th>
<th>L(m)</th>
<th>h(m)</th>
<th>t(m)</th>
<th>hεN(µ)(m)</th>
<th>hεF(m)</th>
<th>fb(N/mm²)</th>
<th>fm(N/mm²)</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>1_1</td>
<td>0</td>
<td>9.00</td>
<td>3.00</td>
<td>0.50</td>
<td>2.00</td>
<td>2.70</td>
<td>9.20</td>
<td>5.00</td>
</tr>
<tr>
<td>1_2</td>
<td>0</td>
<td>9.00</td>
<td>3.00</td>
<td>0.50</td>
<td>2.20</td>
<td>2.70</td>
<td>9.20</td>
<td>5.00</td>
</tr>
<tr>
<td>1_3</td>
<td>0</td>
<td>6.02</td>
<td>3.00</td>
<td>0.50</td>
<td>1.00</td>
<td>2.40</td>
<td>9.20</td>
<td>5.00</td>
</tr>
<tr>
<td>1_4</td>
<td>0</td>
<td>4.72</td>
<td>3.00</td>
<td>0.50</td>
<td>0.00</td>
<td>2.14</td>
<td>9.20</td>
<td>5.00</td>
</tr>
</tbody>
</table>

⚠️ In case of “NOT SIMPLE” structures, the adequacy checks in accordance with EC6 provisions must be performed.
Check Automatic application of the 7 design checks for a selected part of the wall.

<table>
<thead>
<tr>
<th>Check</th>
<th>Ratio</th>
<th>Strength</th>
<th>Load</th>
<th>∂ψ/∂φ</th>
</tr>
</thead>
<tbody>
<tr>
<td>Check 1</td>
<td>0.81(30)</td>
<td>61.92</td>
<td>50.25</td>
<td>9.95</td>
</tr>
<tr>
<td>Check 2</td>
<td>10.31(37)</td>
<td>10.00</td>
<td>103.10</td>
<td>13.33</td>
</tr>
<tr>
<td>Check 3</td>
<td>2.23(64)</td>
<td>11.11</td>
<td>-24.62</td>
<td>0.00</td>
</tr>
<tr>
<td>Check 4</td>
<td>3.53(1)</td>
<td>25.34</td>
<td>-89.39</td>
<td>57.48</td>
</tr>
</tbody>
</table>

Total Check Automatic application of the 7 checks in the structure in total.

The design checks are applied on sections (horizontal and vertical) with respect to 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” presents the results of all the design checks for the selected wall or part of wall.

<table>
<thead>
<tr>
<th>Description</th>
<th>Performance Objective</th>
</tr>
</thead>
<tbody>
<tr>
<td>1_1</td>
<td>A - DL</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Check</th>
<th>Ratio</th>
<th>Strength</th>
<th>Load</th>
<th>σν/σν</th>
<th>l</th>
</tr>
</thead>
<tbody>
<tr>
<td>Check 1</td>
<td>0.25(32)</td>
<td>684.90</td>
<td>-170.69</td>
<td>130.54</td>
<td>4.72</td>
</tr>
<tr>
<td>Check 2</td>
<td>2.78(39)</td>
<td>15.72</td>
<td>43.73</td>
<td>13.33</td>
<td>4.72</td>
</tr>
<tr>
<td>Check 3</td>
<td>0.96(1)</td>
<td>33.33</td>
<td>32.16</td>
<td>0.00</td>
<td>3.00</td>
</tr>
<tr>
<td>Check 4</td>
<td>0.31(64)</td>
<td>219.00</td>
<td>-68.86</td>
<td>471.70</td>
<td>4.72</td>
</tr>
</tbody>
</table>

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

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

<table>
<thead>
<tr>
<th>Description</th>
<th>Performance Objective</th>
</tr>
</thead>
<tbody>
<tr>
<td>1_1</td>
<td>A - DL</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Wall</th>
<th>Check 1</th>
<th>Check 2</th>
<th>Check 3</th>
<th>Check 4</th>
<th>Check</th>
</tr>
</thead>
<tbody>
<tr>
<td>1_1</td>
<td>0.81(30)</td>
<td>10.31(37)</td>
<td>2.23(64)</td>
<td>3.53(1)</td>
<td>1026</td>
</tr>
<tr>
<td>1_2</td>
<td>0.86(62)</td>
<td>214.17...</td>
<td>2.31(64)</td>
<td>2.54(1)</td>
<td>194</td>
</tr>
<tr>
<td>1_3</td>
<td>0.53(39)</td>
<td>5.90(32)</td>
<td>1.18(30)</td>
<td>0.99(37)</td>
<td>687</td>
</tr>
<tr>
<td>1_4</td>
<td>0.25(32)</td>
<td>2.78(39)</td>
<td>0.96(1)</td>
<td>0.31(64)</td>
<td>0.19</td>
</tr>
</tbody>
</table>
STEP 6 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.