SCADA Pro™
Structural Analysis & Design

Example 1
Concrete Structure Analysis and Design

ACE HELLAS
INTEGRATED SOLUTIONS

CONTENTS

• OVERVIEW

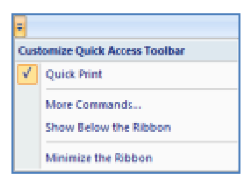
SCADA Pro new version is a result of more than 40 years of research and development while containing all the innovative capabilities and top-notch tools for the construction business.
SCADA Pro utilizes a compact and fully adequate platform for constructing new buildings (analysis and design) or existing 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

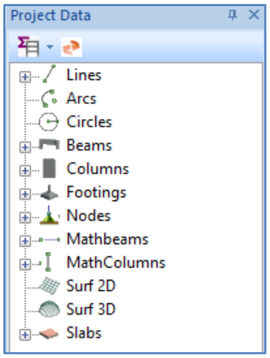
The current manual comes as an aid for a new user of SCADA Pro, making the interface of the software as familiar as possible. It consists of several chapters, where one after the other, describes the consecutive steps of a simple example of a loadbearing masonry project.
The most useful information is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

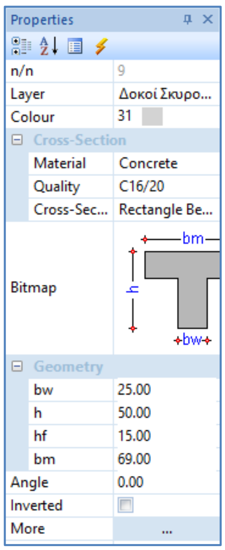
The new interface of the SCADA Pro software is based on the RIBBON structure, thus, the several commands and tools are reached neatly. The main idea of the RIBBON structure is the grouping of commands that have small differences and work in the same context, in a prominent position different to each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

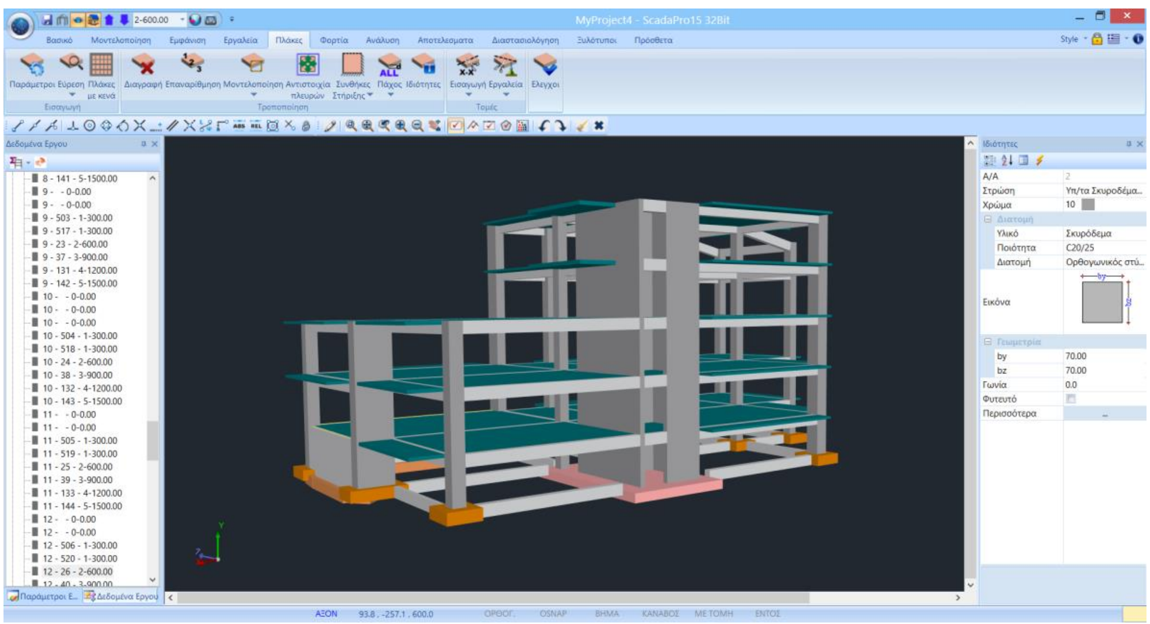
1. GENERAL DESCRIPTION

1.1 Geometry

⚠ For educational reasons, in this example, two different ways for modeling the structure will be presented.

1. The 1st way uses two different plan floors for the model creation along height.
2. The 2nd way uses only one plan floor, following the process of modifying the standar floor and the other elements to create the final model that will reflect the structure under consideration.

The considered structure consists of a basement and four upper structure floors. A part of the basement consists of basement walls, while at the 4th floor an inclined section is met. The mixed foundation consists of single footings, footing beams, connection footing beams and raft foundation as well.



1.2 Materials

The quality of the concrete and the reinforcing steel that was used is the same for all members **Concrete:** C20/25, **Reinforcing Steel:** B500C.

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 Analisys 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

2.1 How to start a new project:

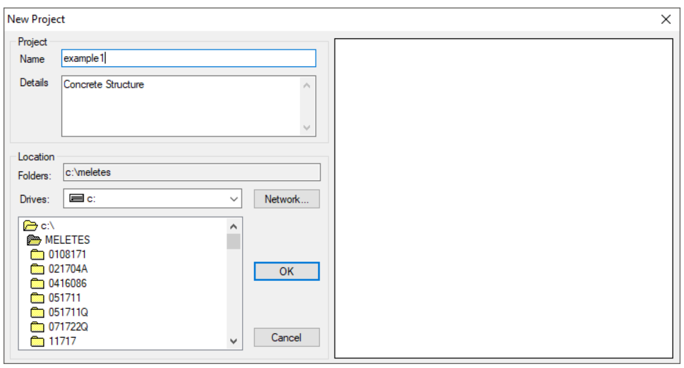
SCADA Pro offers several ways to start a new project. Some criteria related to the acceptance of the starting method are: materials, architectural files, floor plan shape, type of elements usage (beam/shell elements) etc.

⚠ In this example, the way of using the **help dwg file** for the modeling of a concrete structure, will be explained in detail.

Right after opening the program, the starting dialog form with a group of commands, related to the initialization of a project, is displayed:

By left clicking on the related icons, one of the following ways, for the project initialization, can be performed:

⚠ No matter which way you choose to start a new project, the same form always opens to set the project name and the path of the file, a necessary procedure so that the program commands can work.



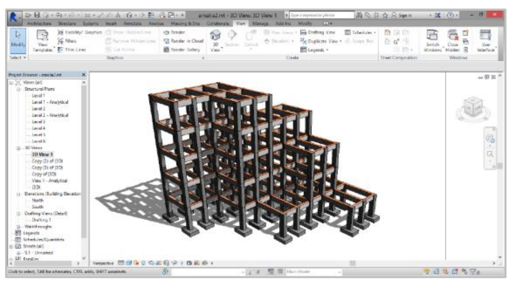
The name of the file can contain up to 8 characters of the Latin alphabet without any symbols (/, -, _) nor spaces. (eg FILE1). The program automatically creates a folder in which all the details of your project are recorded. The location of the folder, that is, the point where the folder will be created, should be on the hard drive. We suggest that you create a folder in C (eg MELETES) where all the SCADA projects are included. (eg C:\MELETES\EXAMPLE1).

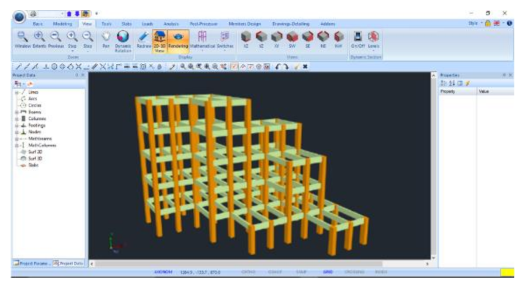
You can add a description or add some information related to the structure, in the "Info" field.

"**new**": It is used when there is no help file in electronic format. The startup is performed in an empty work sheet. The engineer starts with the definition of the height levels and the sections, and moves on to modeling, using the modeling commands and the snap tools of the program.

*"***REVIT***":* Reading ifc files created by the the Autodesk Revit.

By using appropriate libraries, SCADA Pro automatically recognizes all the structural elements (columns, beams, slabs, etc.) with their respective properties, generating in this way the ready for the analysis model.

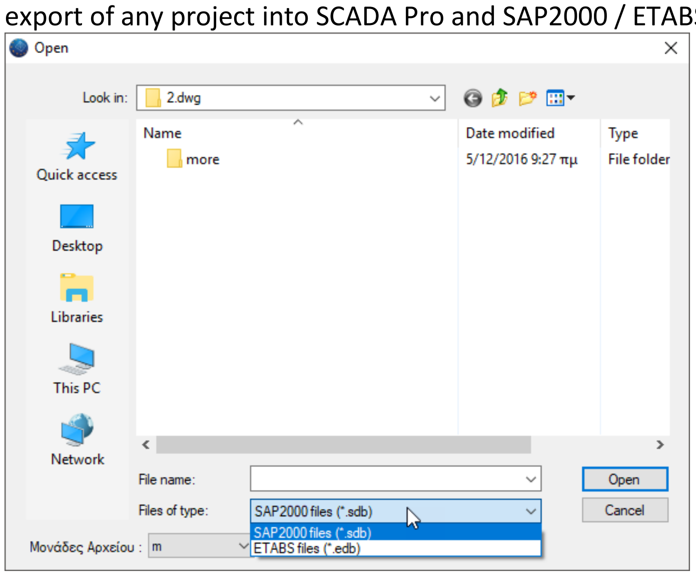




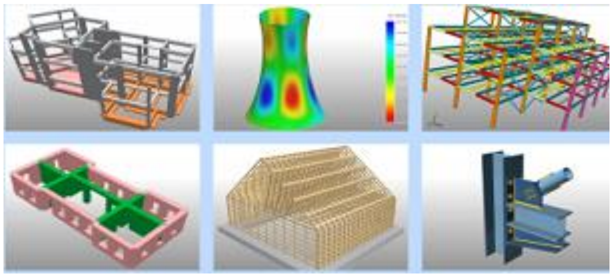
"**ArchlineXP**": Reading xml files from ArchlineXP architectural software.

"**Etabs/SAP2000":** Reading .edb & .sdb files from the static programs ETABS&SAP2000.

The new two-way communication of SAP2000 and ETABS with SCADA Pro allows import and export of any project into SCADA Pro and SAP2000 / ETABS, respectively.



"**Templates**": SCADA Pro carries a rich library of structure templates for every type of material. The command can be activated either by clicking on one of the startup icons or by accessing the Modeling>Add-ons>Templates. Detailed explanation of this command can be found at respective chapter of the manual (Chapter 2. Modeling)

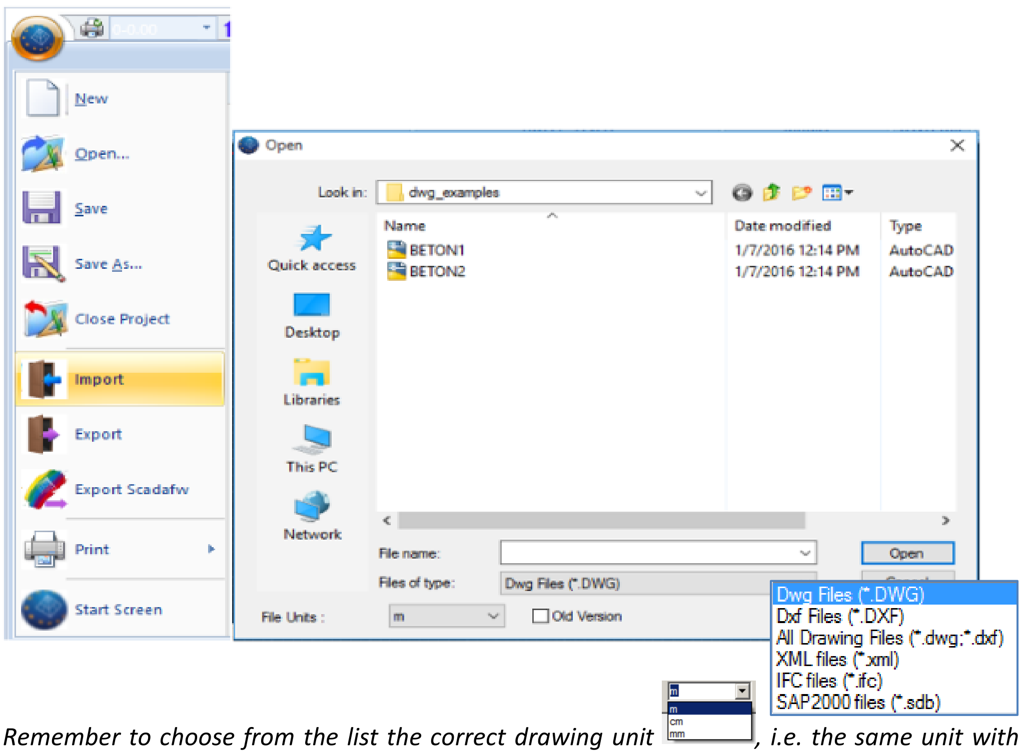


concrete shell elements steel

masonry timber connections (IDEA statiCa)

A structural analysis is usually proceeded by an architecture study which includes dwg or dxf files. These files can be recognized and used by SCADA in many ways.

The import of the dwg or dxf file as help file for the recognition of the cross sections of the elements, can be performed manually, on a semi-automatic way or a full automatic way.



Remember to choose from the list the correct drawing unit, *i.e. the same unit with that of the .dwg, .dxf file.*

- Also, besides the cad files, you may import Revit, SAP2000 etc. files inside SCADA Pro environment.
- The connection between SCADA Pro and Revit is even more powerful since it regards the import of the whole model and not just the help files.
- The connection between SCADA Pro and SAP2000 enables the import and analysis of any concrete, steel, masonry and timber model inside the SCADA Pro environment by the Eurocodes and the National Annexes.

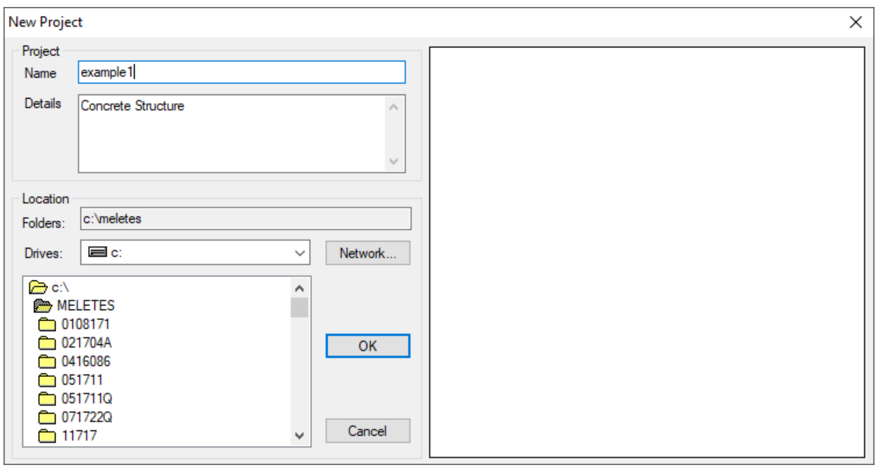
"**dwg-dxf**": Another way to import a dwg or dxf file which in SCADA pro is not just a background that draws snaps on the drawing lines, neither a semiautomatic way to import items by manual selection. It is a completely automated tool that allows you to reproduce a floor plan on the selected floors and automatically create the model.
This command is used for this example and it is extensively described next.

2.2 Automatic recognition of the cross sections derived from a dwg file:

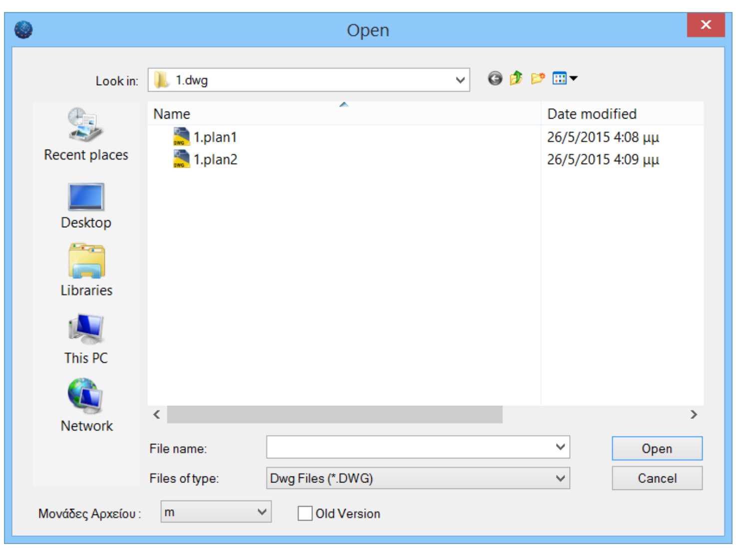
Select the icon 

and in the dialog box:

Define the name and the path of the file. If you wish, you can add some information related to your project, inside the "Info" field and click OK.

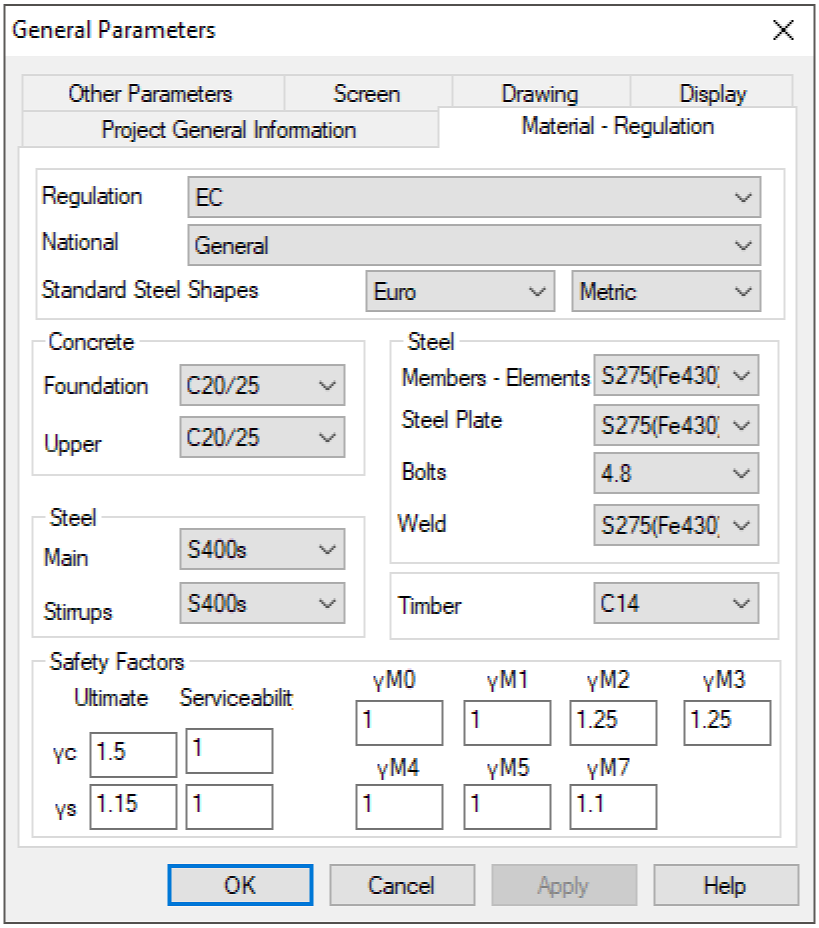


On the next window, select the dwg file and click Open.



⚠ In the case of operators without a typical floor, or more typical floors, or with structures with different floor plan per level, there is a need for introducing more help files. SCADA enables the engineerr to import as many dwg / dxf files as he wishes. These are stored in the study file and can be used to create the static model, combining the fully automatic way with semi-automatic and manual way.

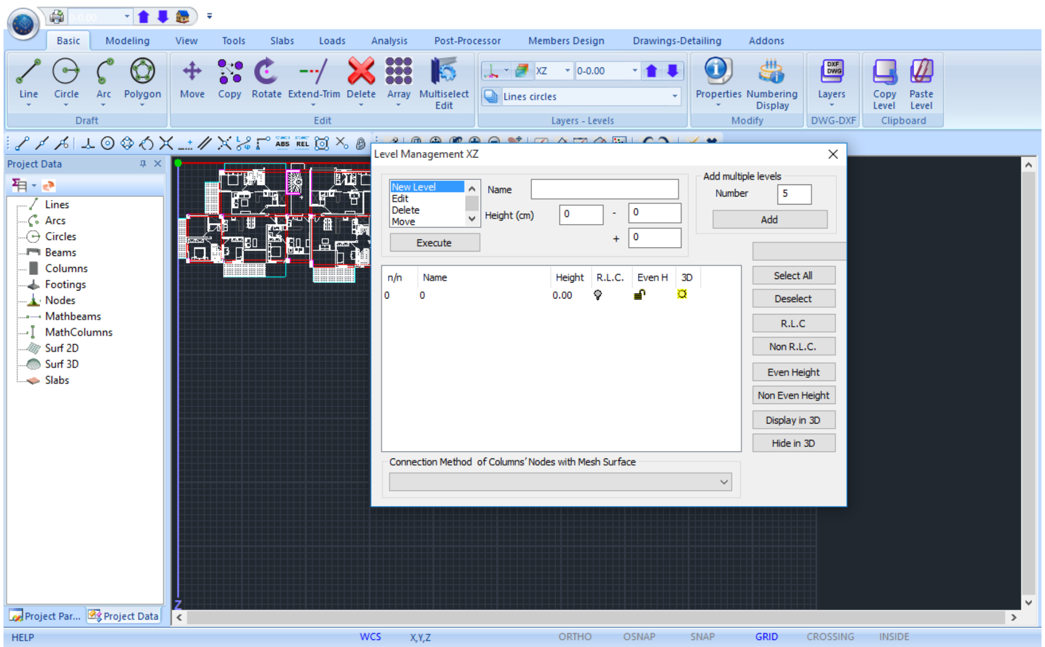
In each new file you create, the General Parameters window appears in the interface, where you can define from the start the Materials and the Regulation that you will use, as well as General Project Data and other parameters.



ATTENTION: The materials must be defined according to the selected regulation and the data entry, as well as all cross sections mast have the right types (C for new regulations, B for the old ones).

⚠ * Predefined scenarios are created according to the Regulation and Attachment option you make at the beginning, within the General Configuration window that opens automatically immediately after the file name is defined.

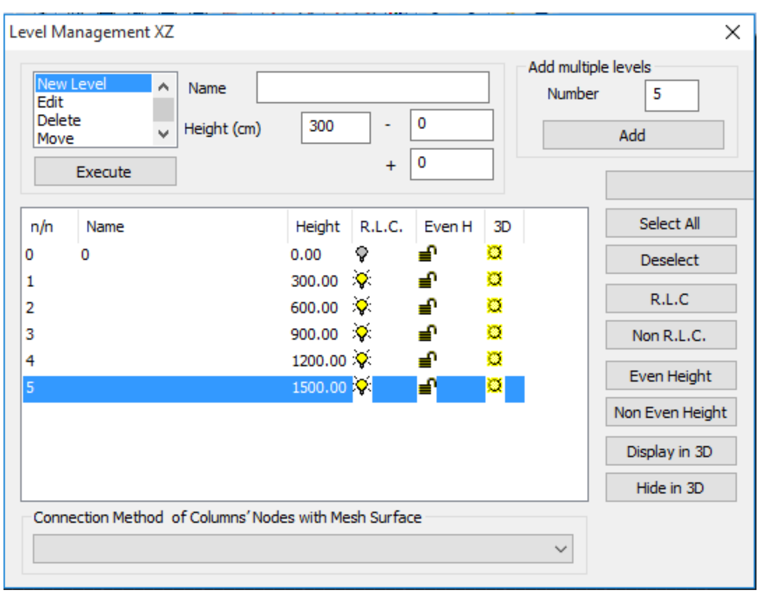
Click OK and then the project opens automatically in SCADA environment, with all its designed elements.

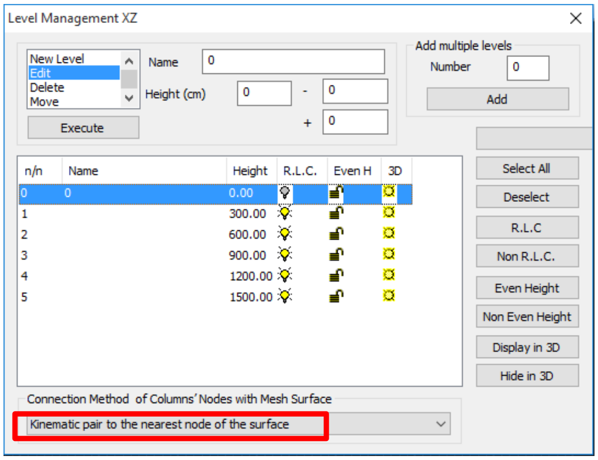


The drawing opens inside the Scada environment, with all of its elements.
At the same time, the "**Level Management XZ"** window, in which you define all the levels of the model, opens.

First modify the level 0 (default). Select "**Edit**" and select the level 0 from the list (blue color indicates the selected level). Now you can change the name and the height value. Select "**New Level**" and then type a name and a height. Complete the range "–" and "+", in case of uneven height, slopes, or vertical mesh elements, to make them belong to the current level (for mass distribution) and visualize them on the current level, too.
Also you can automatically create multiple levels with "Add multiple levels" command.

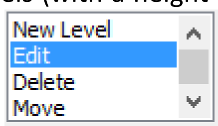
Set the number of the levels that will be created and click "**Add**":





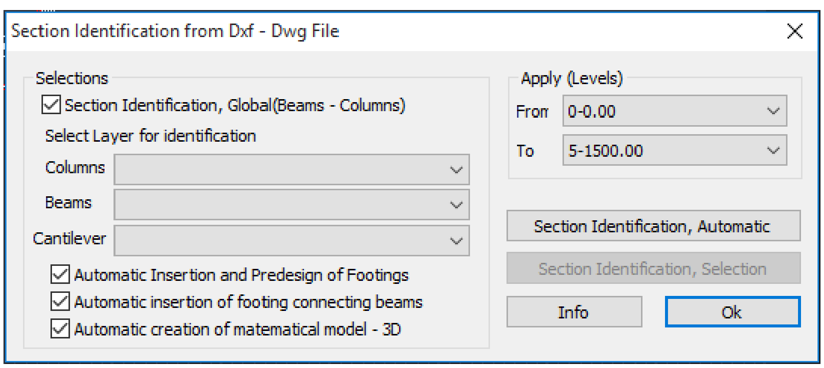
IMPORTANT NOTE:

⚠ *Make sure that you define* ***'Kinematic pair to the nearest node of the surface'*** *at the level 0, so the nodes of the members of the columns can automatically depend on the nodes of the plate that will be created on the foundation.*

The list is updated with the levels (with a height difference of 3m (300cm)), which are editable through the "Edit" command 

(for further information view the respective chapter of the Manual)

Close the window and the "**Section Identification from Dxf – Dwg file**" form automatically opens.



It regards an automation that recognizes the beams and columns of any shape (T, Π, Γ), slabs and cantilevers, footings and connection footings beams, while it automatically creates the mathematical model of the whole model as well.

The lists next to the element types "**Section Identification**" contain the layers of the .dwg help file.

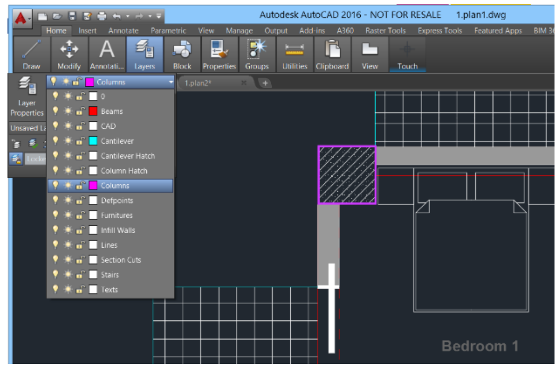
NOTE:

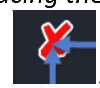
⚠ *To make sure that the automatic procedure of the sections recognition will be performed, some preconditions must be taken into accound.*

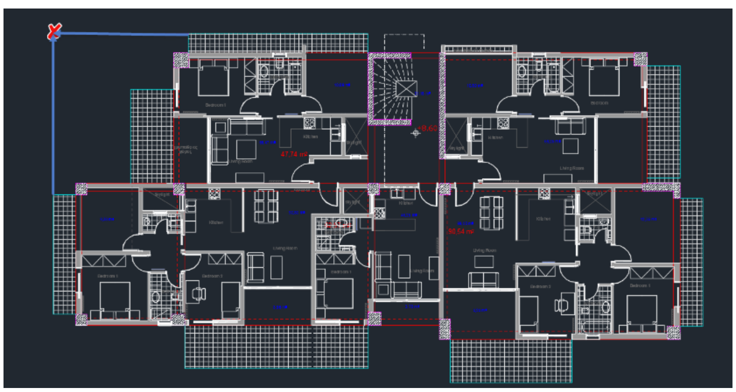
PRECONDITIONS:

1. *Each plan that will be used as a help dwg file must be in a separate file that includes only this plan (without any other drawing entities).*

2. *The lines (or/and polylines) that define the columns, the beams and the cantilevers must belong on their one layer without any other type of element or/and drawing entity inside this layer.*

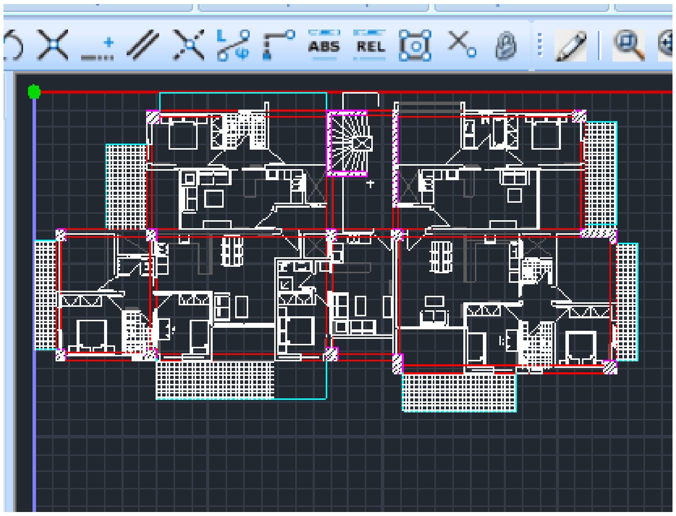


3. *The help file is imported to the SCADA environment at the active (current) level by placing the upper-left corner of the drawing to the (0,0) point of the SCADA coordinate system* . During the import of the help files be aware of the floor plans inserting position (by their reference point) so that the correct placement for all the plans is achieved.

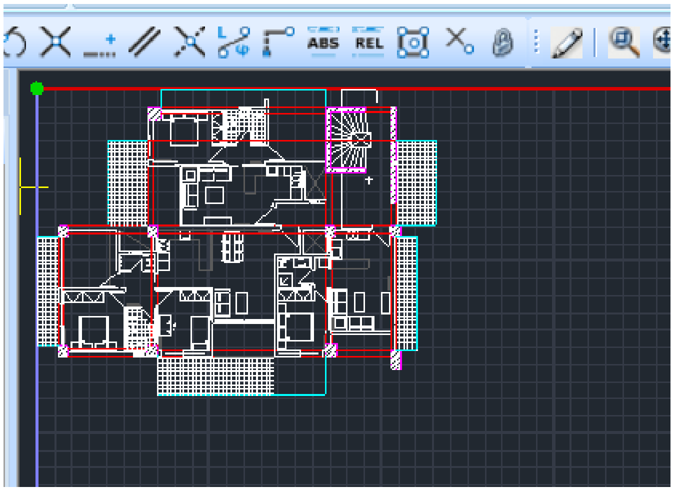


Floor plan 1 (dwg)

Floor plan 2 (dwg)



Floor Plan 1 (Scada)



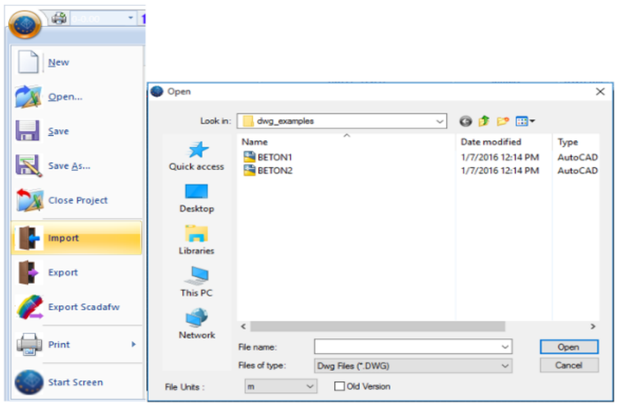
Floor Plan 2 (Scada)

NOTES:

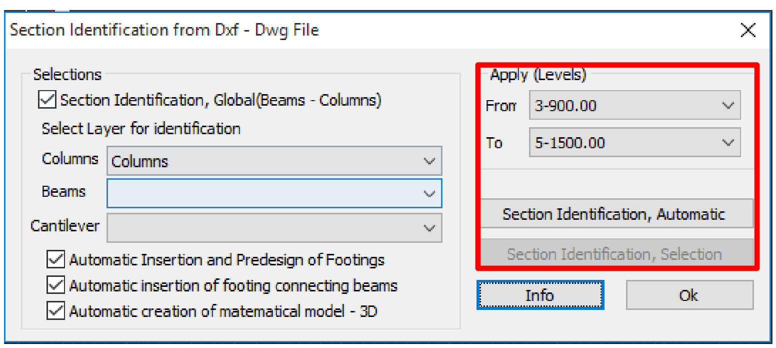
- *Through the* 

command that appears on the initialization window, you can import a help file and perform the automatic modeling with one single command.

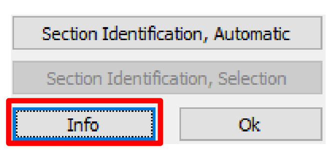
- *For each help file of the same project, use the “***Import***” command and with the considered floor plan XZ activated import the drawing.*

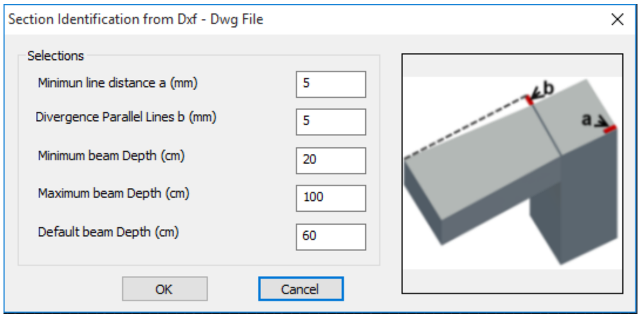


- T*o perform the automatic modeling move to the “***Modeling***” unit and select the “****Elements Creation****” command.*
  Next, select Columns – Beams – Foundation Beams, for the <u>automatic</u> or <u>selective</u> elements creation on the selected levels.



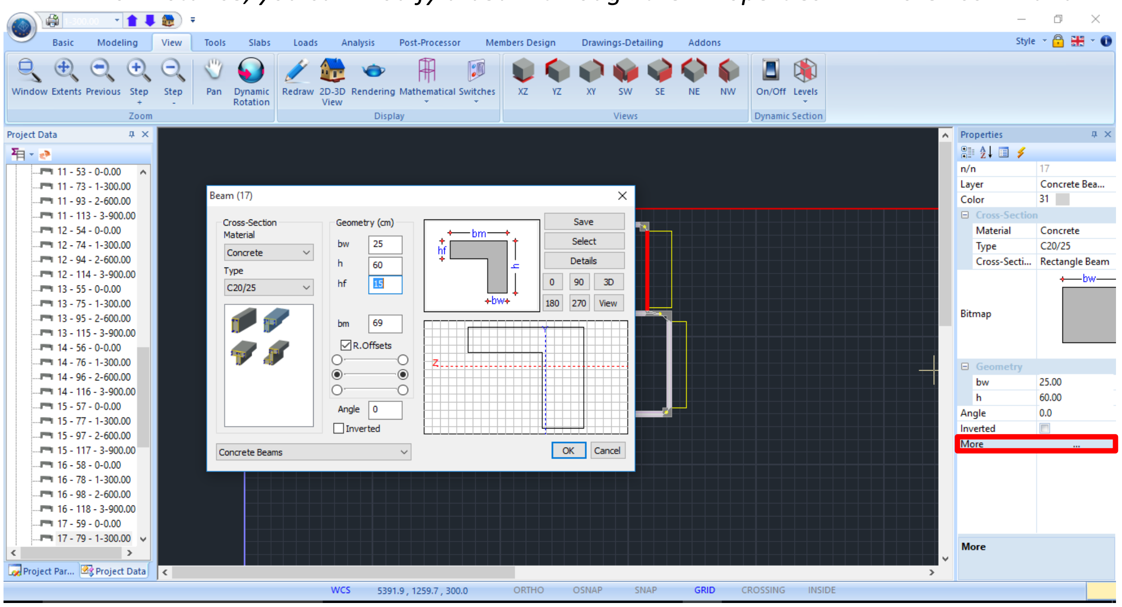
- *The “***Info***” command offers the ability to handle some drawing imperfections and limits so that they will be ignored during the automatic creation. For the beam creation, you define the divergence parallel lines distance, the minimum line length and the t beam depth which is rectangular defaulted by the program*

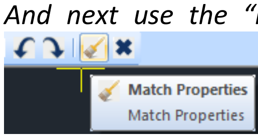




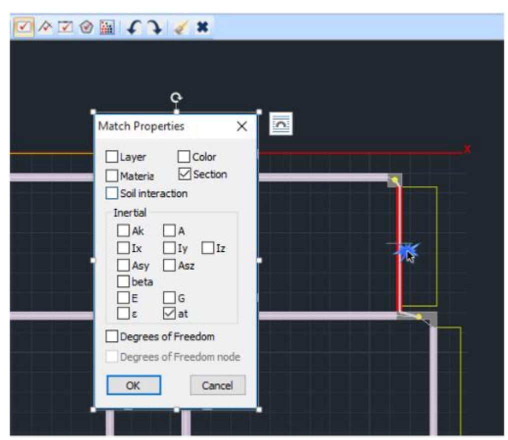
- *The modification of the beam cross section after the import can be performed in several ways.*

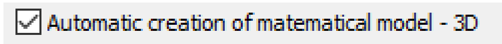
⚠ *For instance, you can modify a beam through the "Properties" > "More" command*



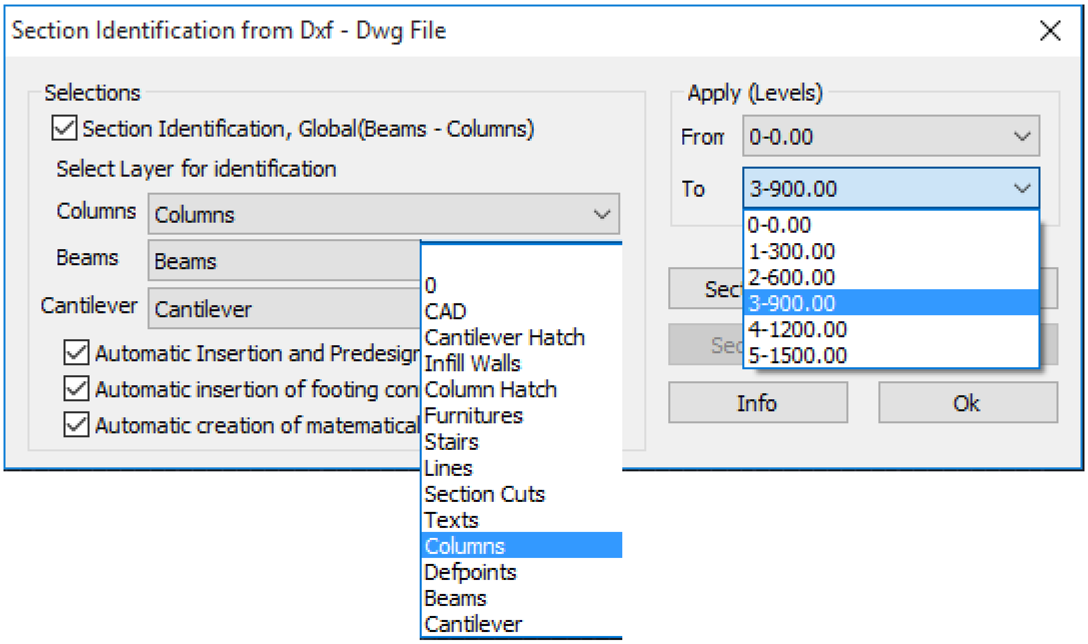
And next use the "Match Properties" command 

to modify the rest of the beams that share the same cross section by selecting the preferred elements with one of the available methods .



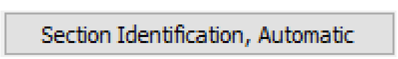


By activating the "Automatic creation of the mathematical model – 3D", the program not only does it recognize and insert the cross sections of the physical model, but it calculates the inertial properties and creates the mathematical model too.

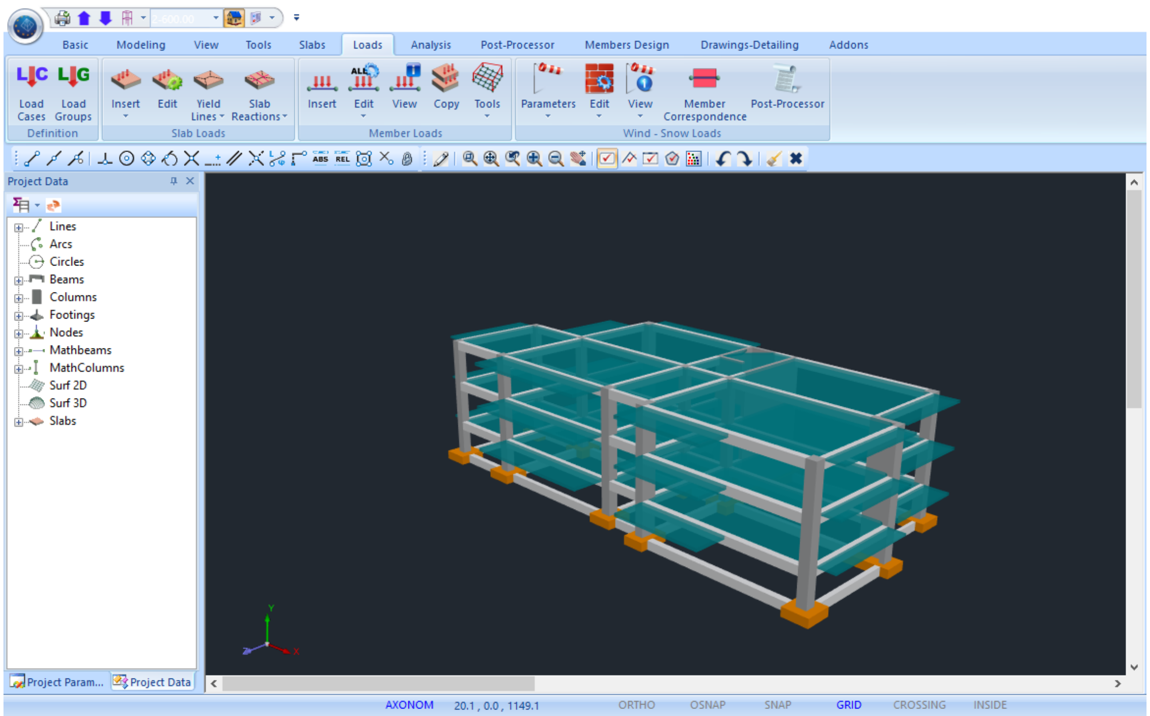


Precondition for the automatic creation of the slabs and the cantilevers is the selection of the columns and beams creation as well as the automatic creation of the mathematical model so that the members that surround the slabs exist.

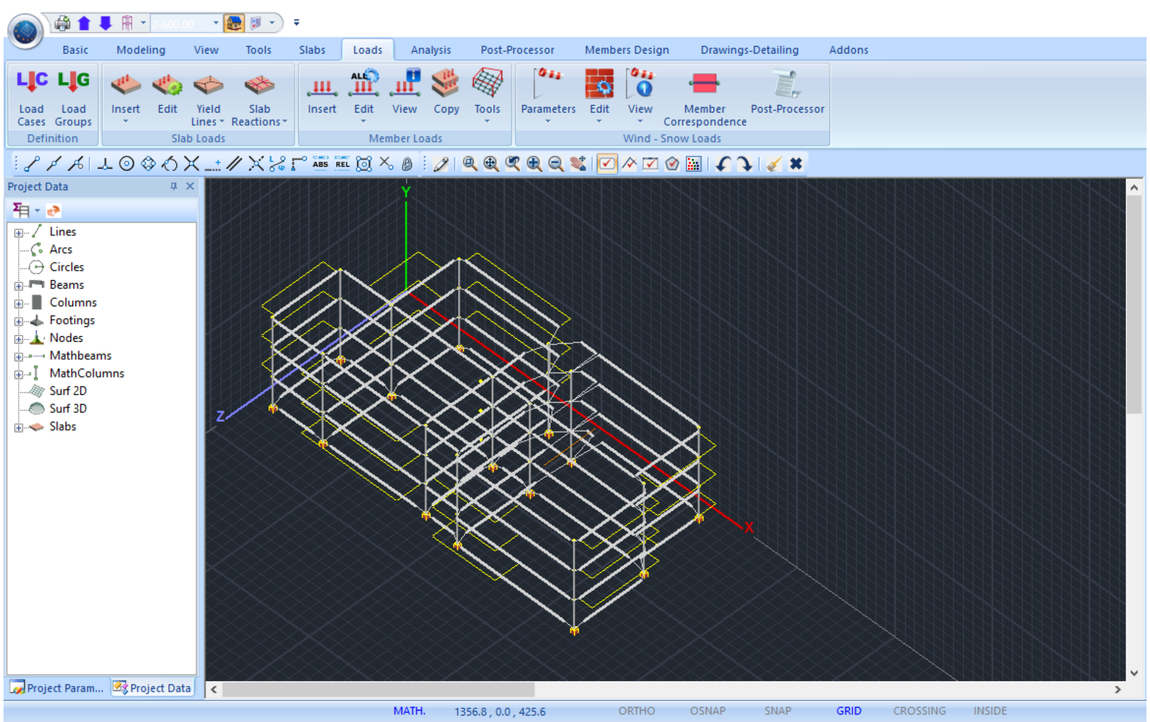
The "**From - To**" commands define the levels that the drawing will be applied.



Select the "**Section identification, Automatic**" to view the Rendered representation of the model.



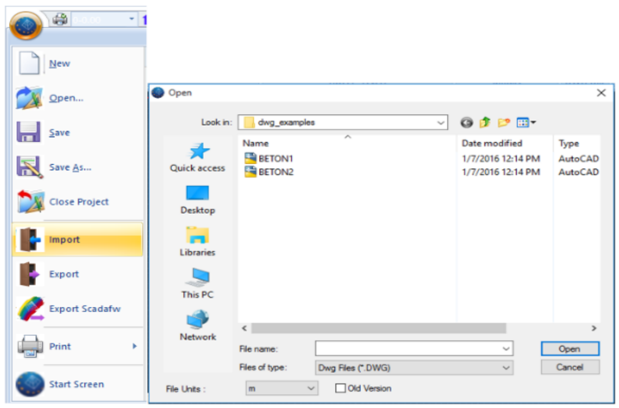
Deactivate the rendering to view the mathematical model

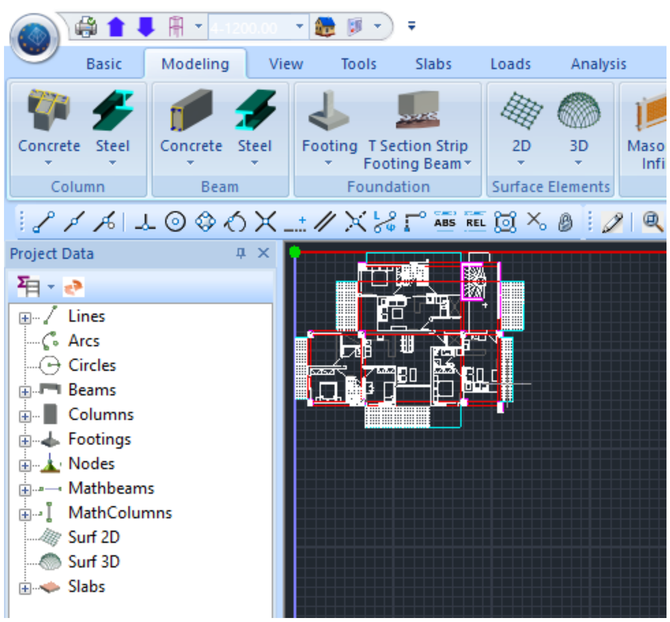


2.3 Import of a new floor plan (new dwg file) at the existing model for the creation of the rest floor levels:

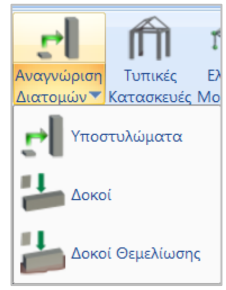
After creating the first floor plan (*plan1.dwg*) for levels 0 to 3, the levels 4 and 5 of the model do not include any elements. For the creation of the elements that belong to the second floor plan (*plan2.dwg* - levels 4 & 5) follow the automatic process described below:

➢ "**Import**" the plan2.dwg at the current empty XZ level of the SCADA (level 4)

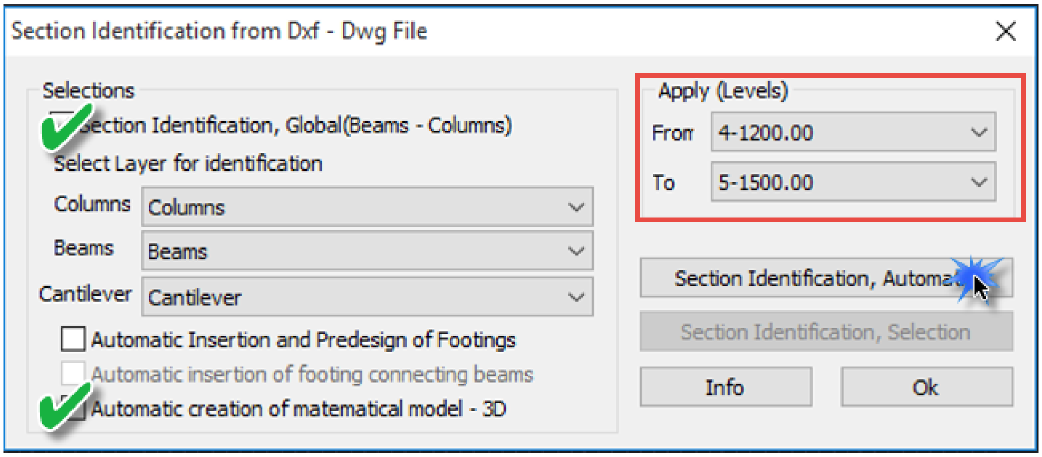




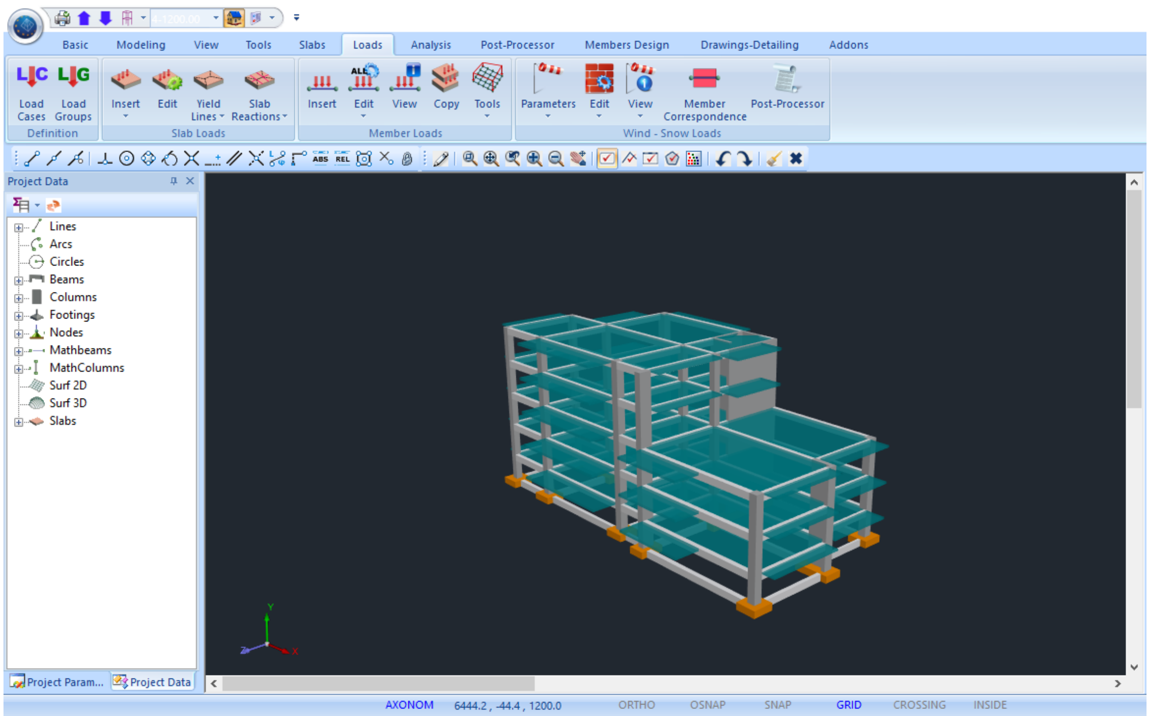
➢ Use the automatic Elements Creation through the "**Modeling**" menu.

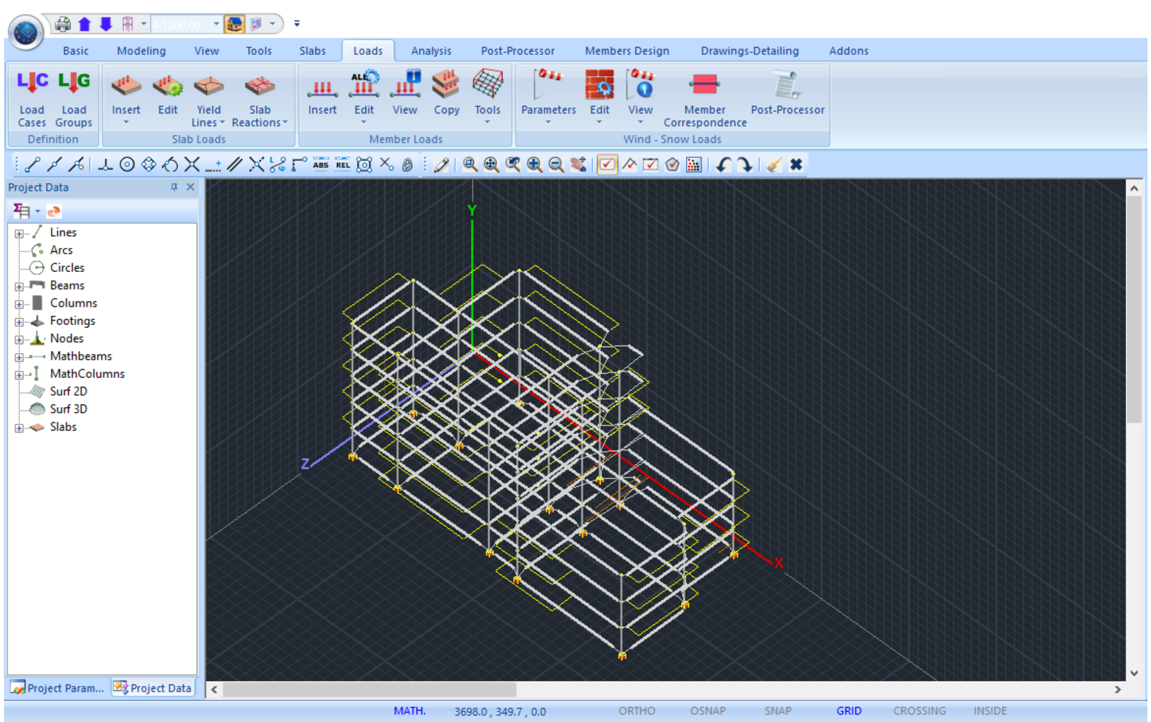


Select Columns or Beams and in the dialog box activate:



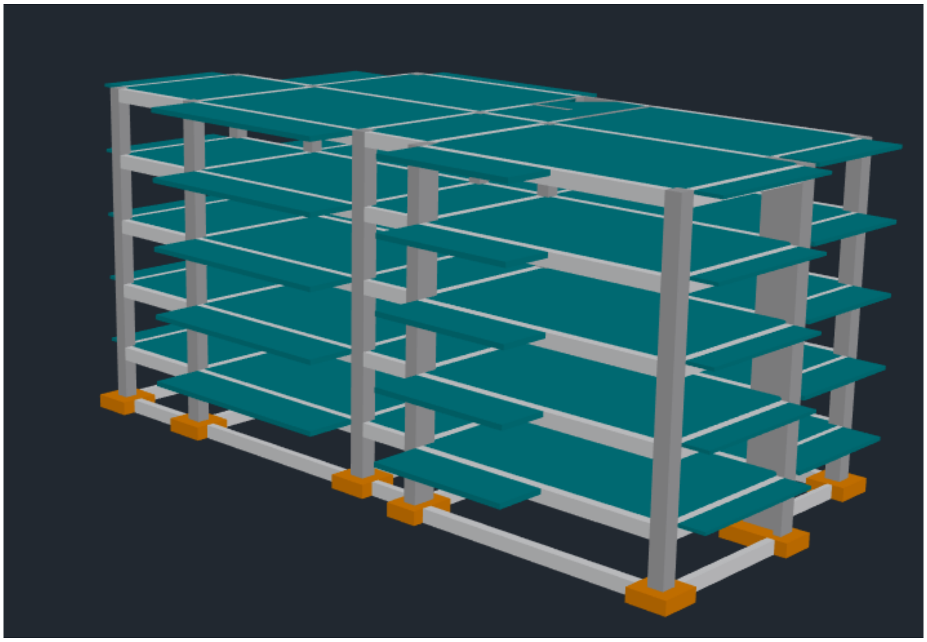
- The "Section identification, Global (Beams - Columns)" that activates the corresponding fields for the selection of the respective layers for the creation of Beams, Columns and Cantilevers by selecting their corresponding layer.
- The "Automatic Creation of Mathematical Model – 3D"
- The "Application" at levels 4&5
- The "Section Identification, Global"


Basic
Modeling
View
Tools
Slabs
Loads
Analysis
Post-Processor
Members Design
Drawings-Detailing
Addons
Load Cases
Load Groups
Definition
Insert
Edit
Yield Lines
Slab Reactions
Slab Loads
Insert
Edit
View
Copy
Tools
Member Loads
Parameters
Edit
View
Member Correspondence
Post-Processor
Wind - Snow Loads
Project Data
Lines
Arcs
Circles
Beams
Columns
Footings
Nodes
Mathbeams
MathColumns
Surf 2D
Surf 3D
Slabs
Project Param...
Project Data
AXONOM
6444.2 , -44.4 , 1200.0
ORTHO
OSNAP
SNAP
GRID
CROSSING
INSIDE


Basic
Modeling
View
Tools
Slabs
Loads
Analysis
Post-Processor
Members Design
Drawings-Detailing
Addons
Load Cases
Load Groups
Definition
Insert
Edit
Yield Lines
Slab Reactions
Slab Loads
Insert
Edit
View
Copy
Tools
Member Loads
Parameters
Edit
View
Member Correspondence
Post-Processor
Wind - Snow Loads
Project Data
Lines
Arcs
Circles
Beams
Columns
Footings
Nodes
Mathbeams
MathColumns
Surf 2D
Surf 3D
Slabs
Project Param...
Project Data
Y
Z
X
MATH.
3698.0 , 349.7 , 0.0
ORTHO
OSNAP
SNAP
GRID
CROSSING
INSIDE

2.4 Typical Floor Modification:

Alternatively, we can follow the 2nd way, i.e. import only one floor plan (*plan1.dwg*) and copy it to the rest of the levels, and perform the desired modifications, to the rest of the levels.

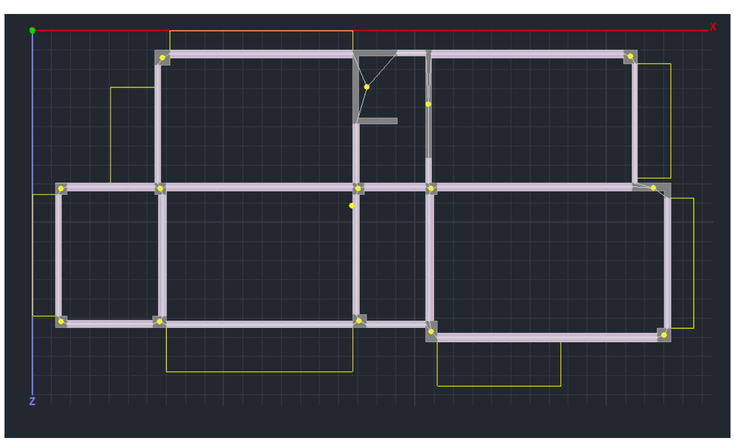


IMPORTANT NOTE:

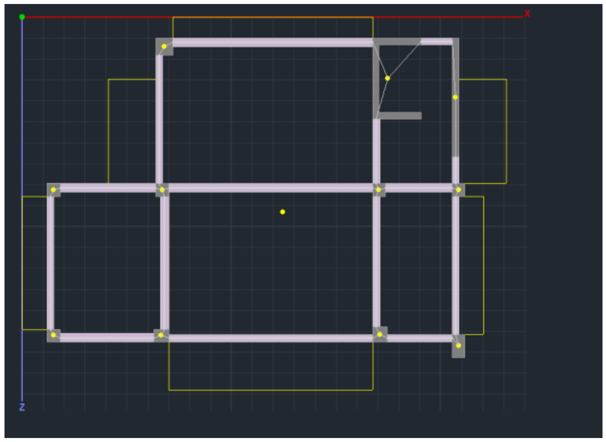
⚠ *To modify the physical model, the mathematical model must not exist, but since the* ☑ Automatic creation of matematical model - 3D *command created the mathematical model, you must delete the mathematical model of the desired levels that you wish to modify. In case that you know that you will perform modifications you may deactivate the "Automatic creation of mathematical model" option, so that you won't have to delete it later.*

Level 4

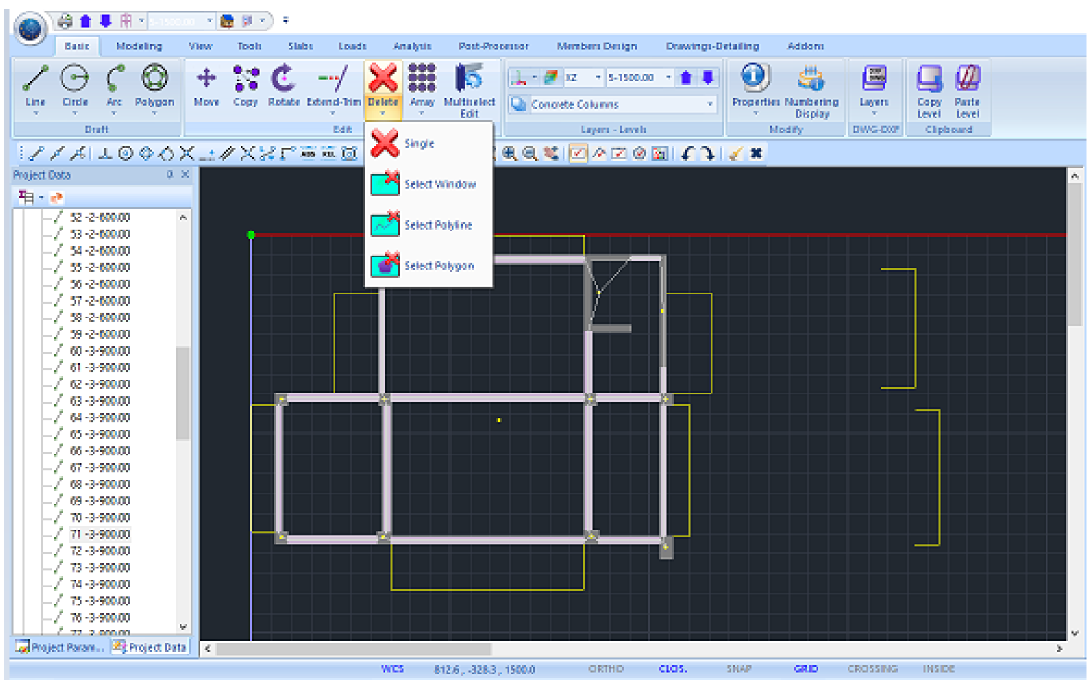
Initial floor plan

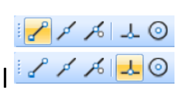


Floor plan after the modifications



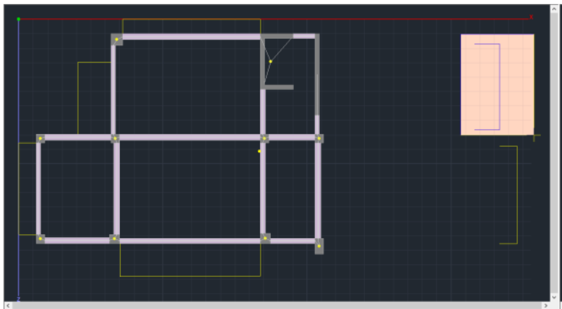
Delete the Mathematical Model of the current level and then delete the elements that do not exist at this level (level 4).

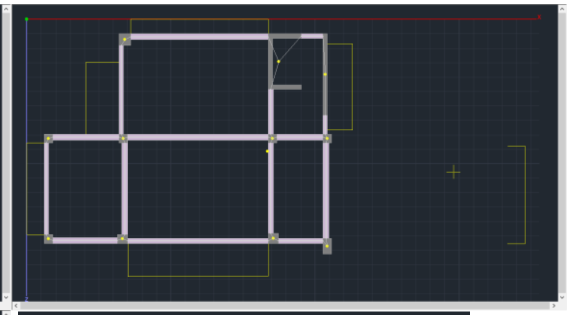


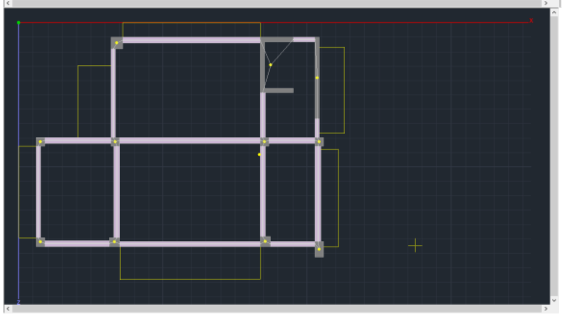
Using the  selection option, and by activating the Endpoint and vertical  snaps, use the 

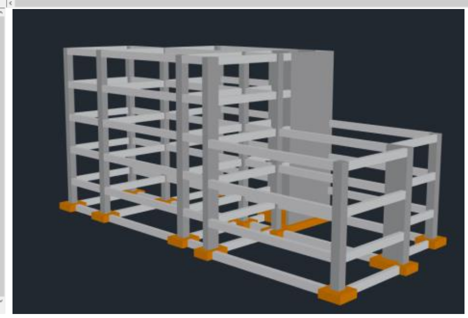
command to move the cantilevers at the correct position. Activate the Move command and select with a window the lines of the upper cantilever. Next, right click the mouse to end the selection. With a left click, show an end point of one of the selected lines (activated ) and left click to the destination point with the snap  activated.

Follow the same procedure for the Level 5.

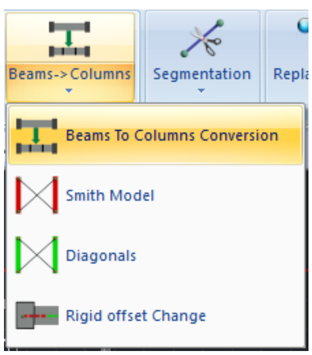




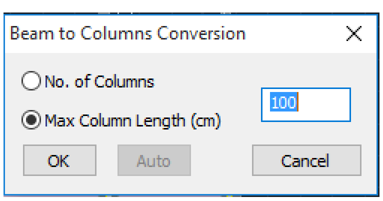




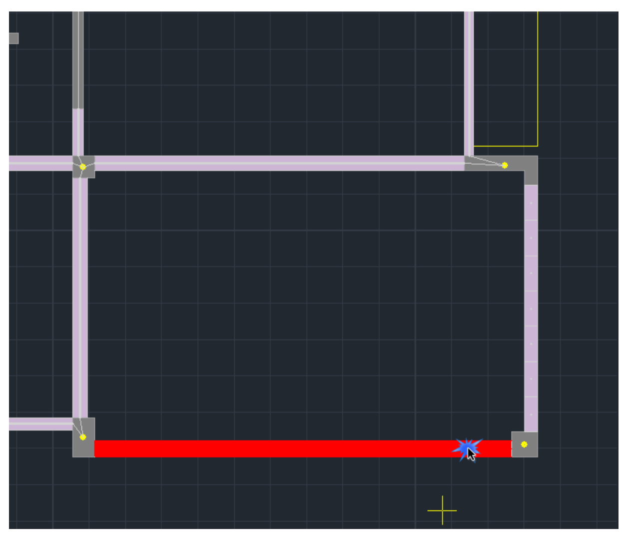
2.5 How to simulate the basement walls:



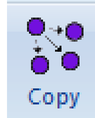
There are several ways to simulate the basement walls. In this example the "Beams to Column Modification" method was used. For the first level (ground floor level), through the "**Tools**" >> "**Model**" unit, select the "**Beams to Columns Conversion**" command. At the dialog box that appears, select one of the two choices and insert the appropriate number.



Level 1:



Left click to select the beams of the level 1 that will be automatically converted into consecutive columns (after you delete their mathematical representative).

You can repeat the same process for Level 0 or copy the new columns that were created at level 1, to level 0 using the "Copy" 

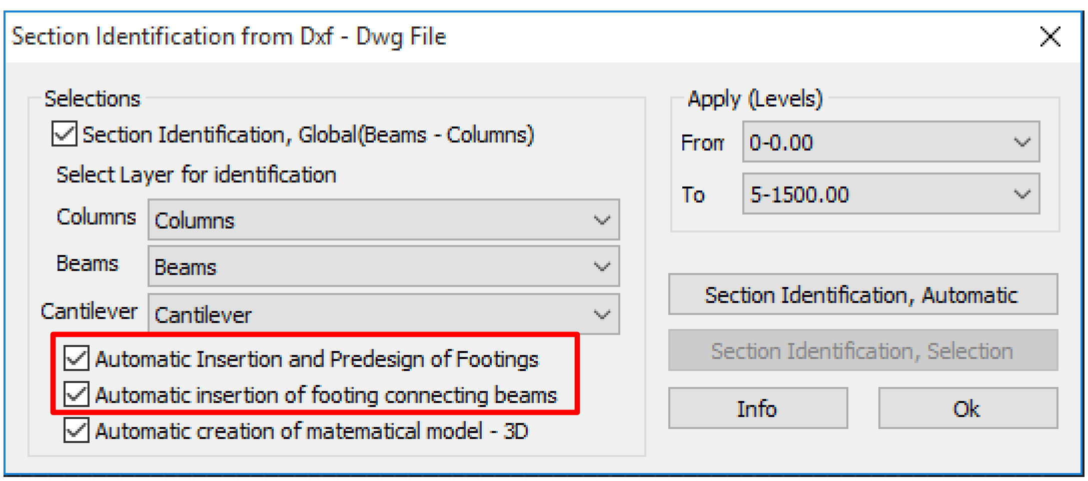
command.

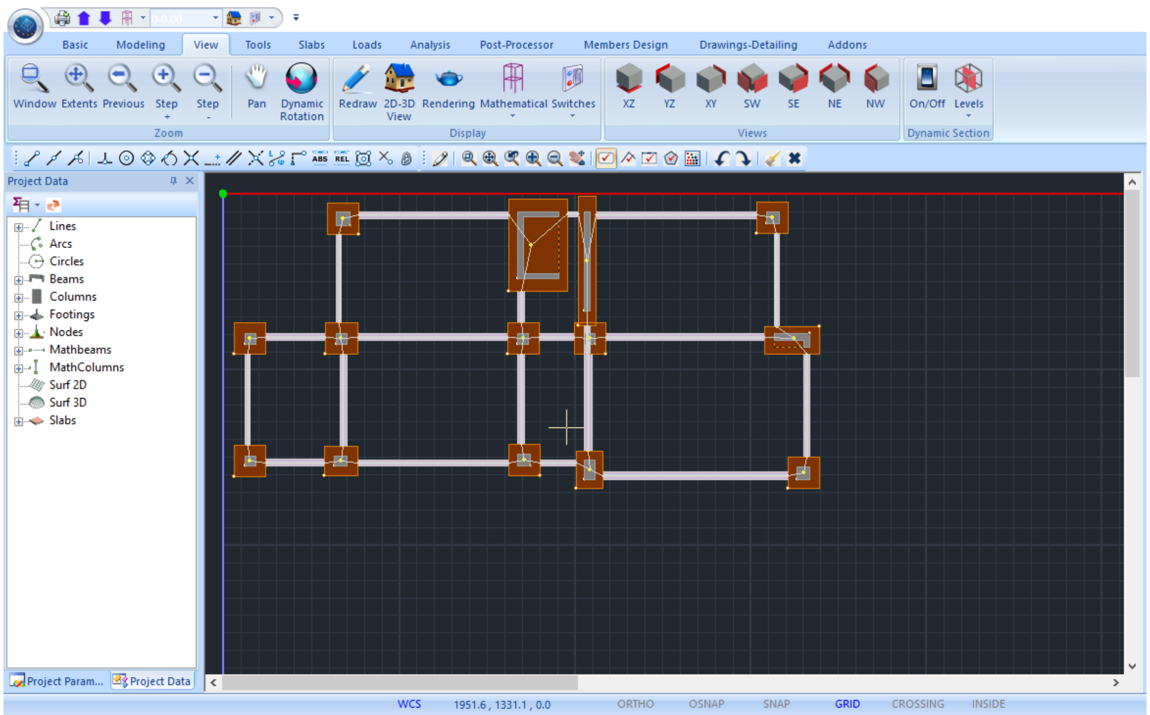
First, delete the Footing Connection Beams from level 0. Next, call the command and select the elements that you want to copy. The selection can be performed either individually or by window, polygon etc. Next, right click to end the selection and define the characteristic point (end point of a line, column, beam etc). Move to level 0 and define the respective point for the selected elements copy.

The modeling is completed with the creation of the mathematical model and the connection between the nodes of the columns above with high rigidity beam members (see §2.10).

2.6 Automatic import of Footings and Footing Connection Beams at the foundation level:

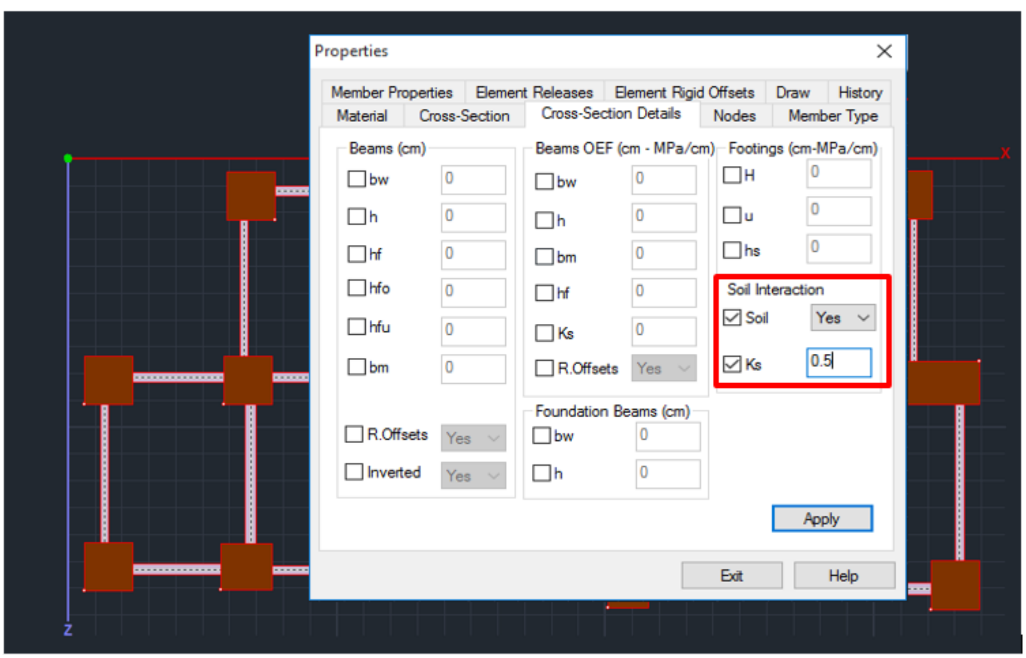
The new version of SCADA Pro, enables the user to **Automatically Insert and Predesign the Footings,** and insert **Footing Connection Beams** as well, during the **Section Identification procedure**.





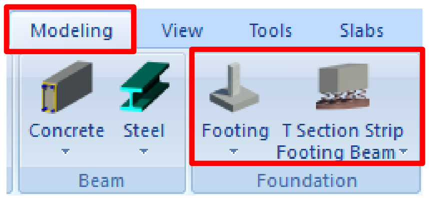
Level 0

The dimensions of the footings, derive from a predesign, and are imported as fixed (with a Ks=0). The user is to select all the footings through the "Multi Select Edit" command and set a value for the Ks variable by the soil.

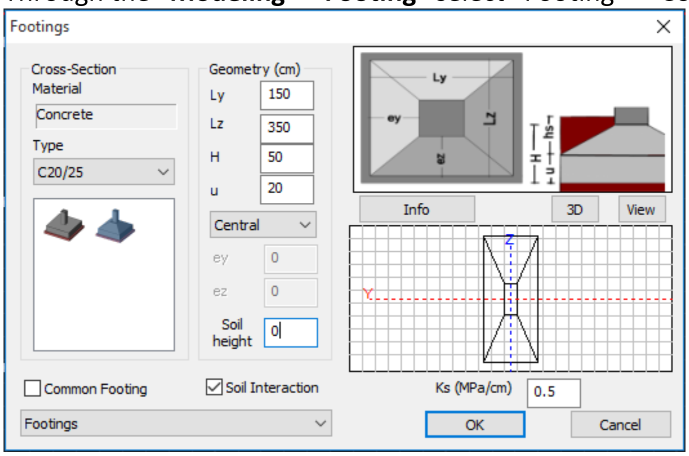


For completeness, in this example, the manual way of inserting footings and footing connection beams is presented as well.

2.6.1 Footings



Through the "**Modeling**">"**Footing**" select "Footing"> "Cone shape":



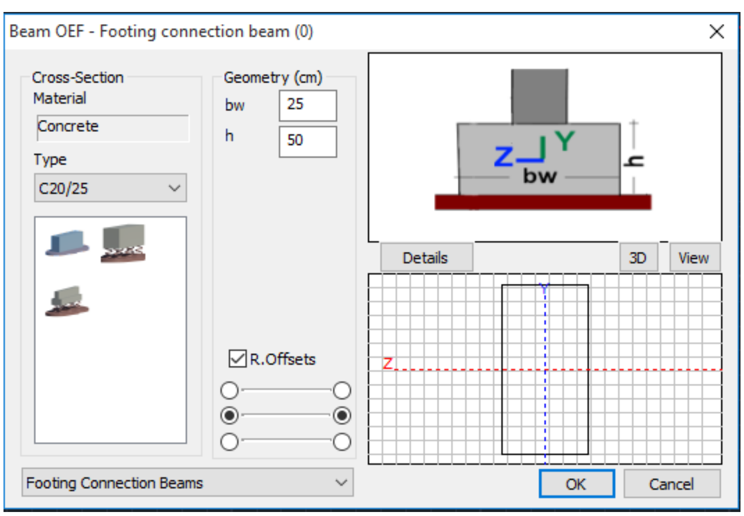
In the dialog box define the geometry and material properties of the footing.

Click "**OK**" and place the footing to the model by left clicking on one of the upper structure columns edge.

Repeat the same process to place the rest of the footings as well.

2.6.2 Footing Connection Beams

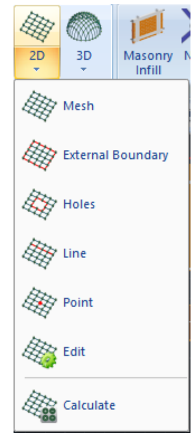
Select "Footing Beam"> "Footing Connection Beam":



In the dialog box, define the properties of the material and the geometry as well as the edge reference of the beam(*).
Click "**OK**" and place the beam by left clicking at the endpoints. Repeat the same process to place the rest of the footing connection beams.

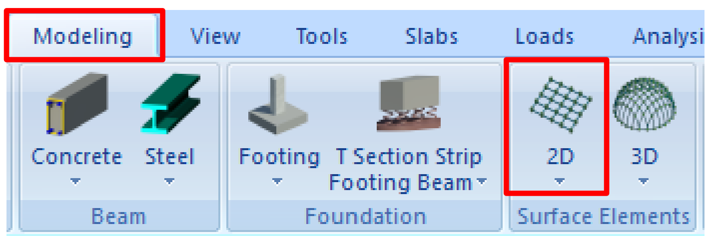
(*)*During the placement of a beam you can alternate the endpoints reference edges of the beam using the TAB and SHIFT keys.*

2.7 How to define a raft:

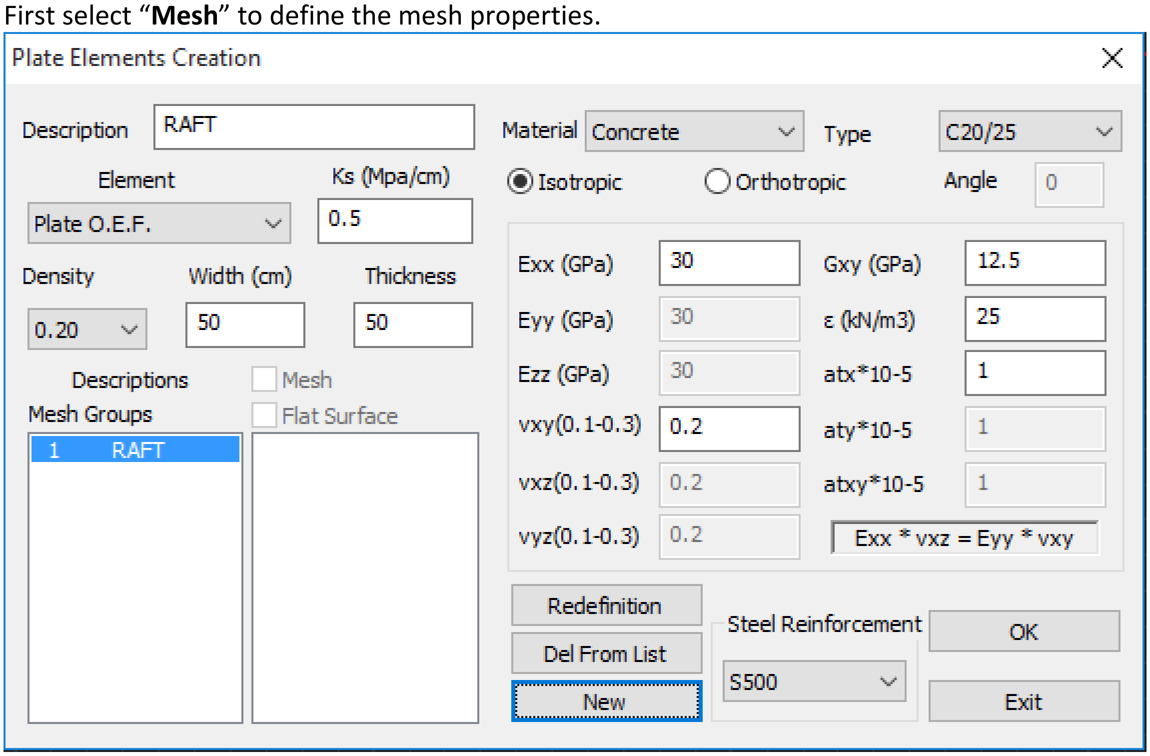


⚠ *For completeness, in this example, a part of the foundation will be replaced with raft, so that every foundation type can be presented.*

To model a raft, use the 2D Plate Elements.

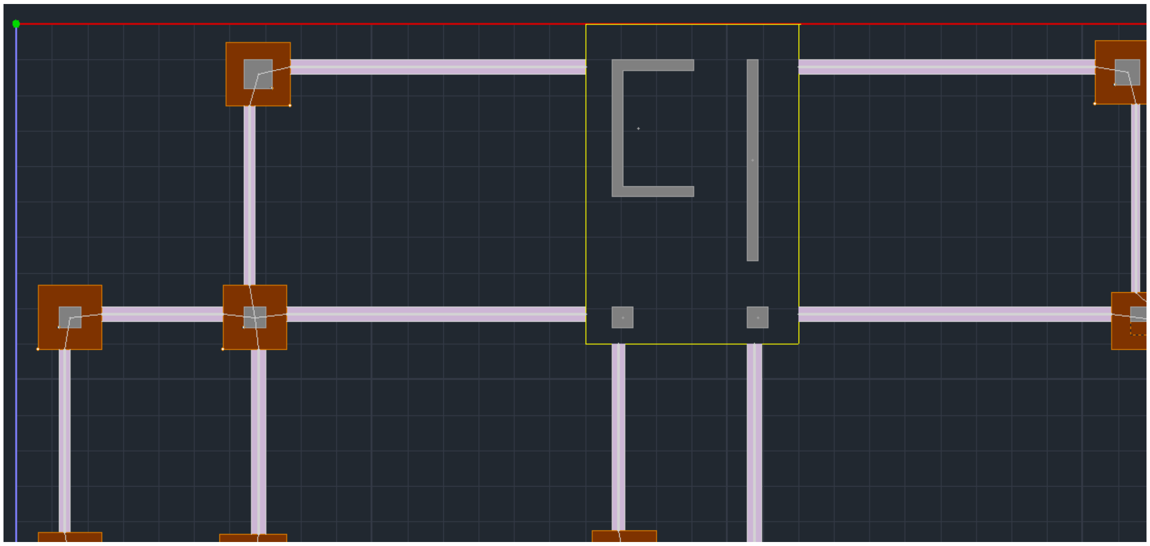


First select "**Mesh**" to define the mesh properties.

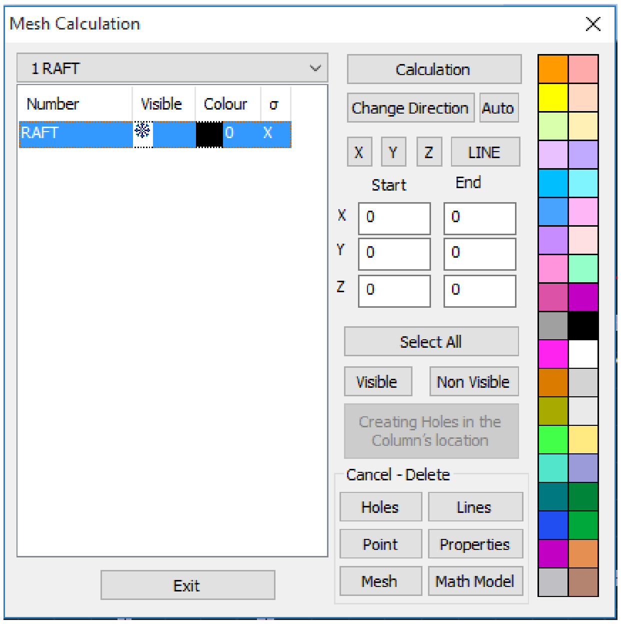


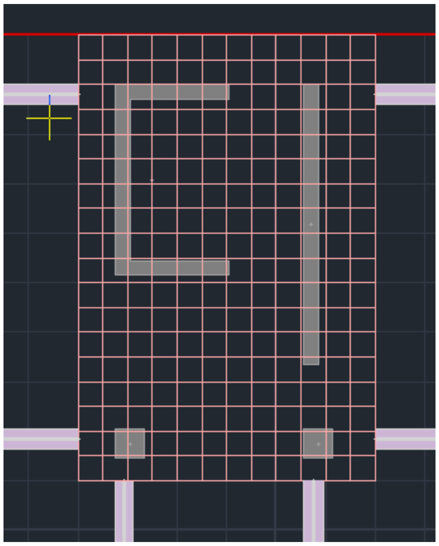
In the dialog box, set a description, the material, the quality, the type of the element, the Ks value, the density and the dimensions of the mesh as well as the quality of the steel reinforcement. Click "**New**" and then "**OK**".
Next select "**External Boundary**".

Define the perimeter of the raft by left-clicking on the corners of the boundary. Right click to complete the selection.



Finally perform "**Calculation**".
In the dialog box that appears select the mesh, so that it turns blue and then click "Calculation".
The mesh is automatically created. Click "**Exit**" and the mesh is created.

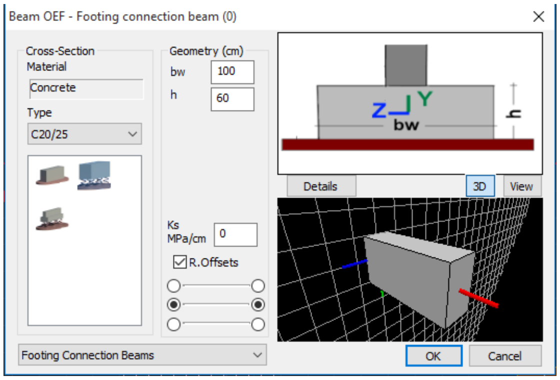


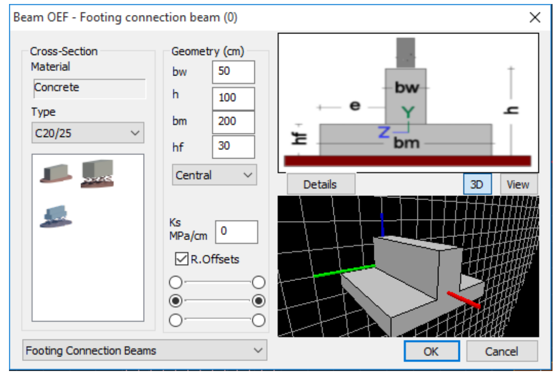


We move on to the rest of the foundation elements and we'll get back to mesh elements later. After completing the import of the physical elements, the mathematical model of the respective physical elements are to be created.

2.8 How to insert Footing Beams at the basement walls:

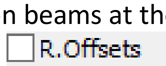
At the “**Modeling**” > “**Foundation**” unit select “Footing Beam” > “Rectangular” or “T” section.

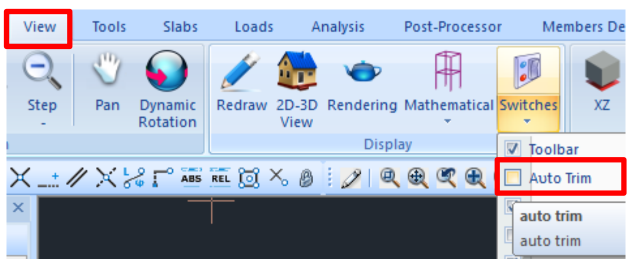




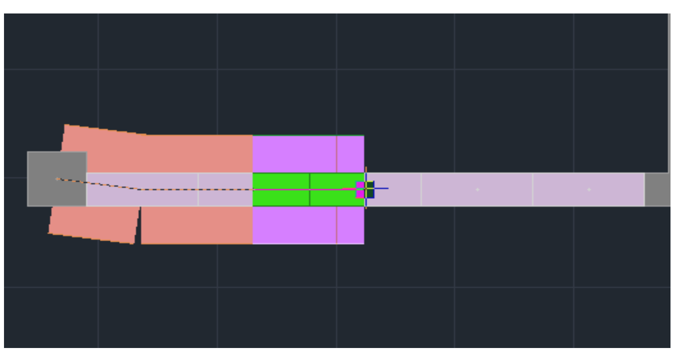
In the dialog box, define the parameters, the material and the geometry of the foundation beam.

To insert foundation beams at the basement walls, first of all, deactivate:

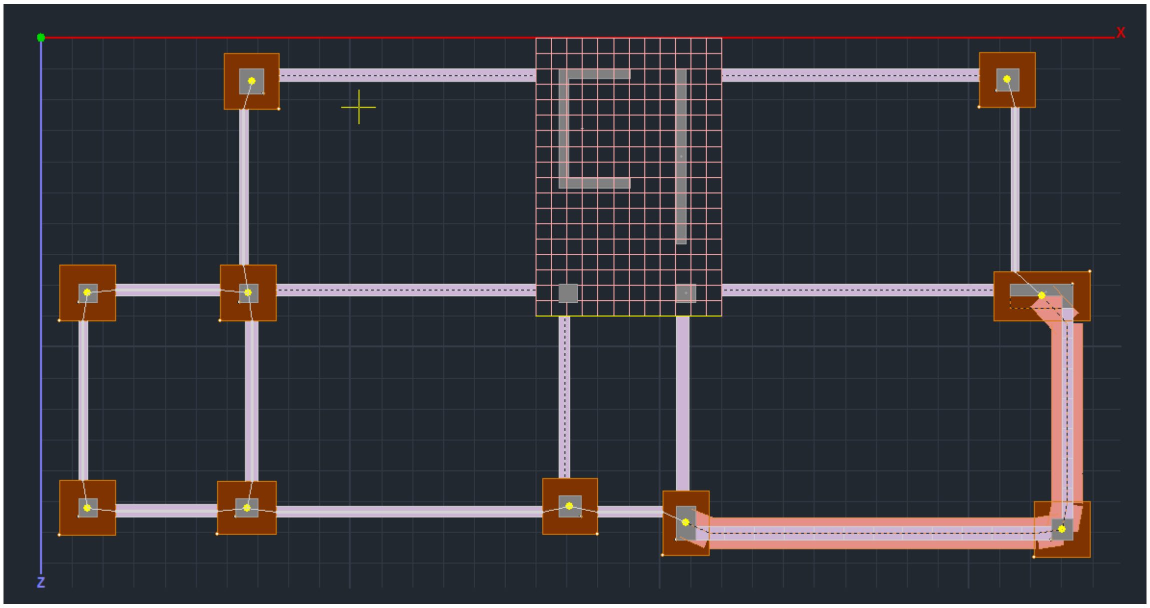
- “R.Offsets”  (in the dialog box)
- “Autotrim”  (in the View>Switches command group)



Next, insert footing beams at the basement walls, using the snaps, from center to center.



Complete the insertion of all the foundation above elements, so that level 0 looks like the following image:

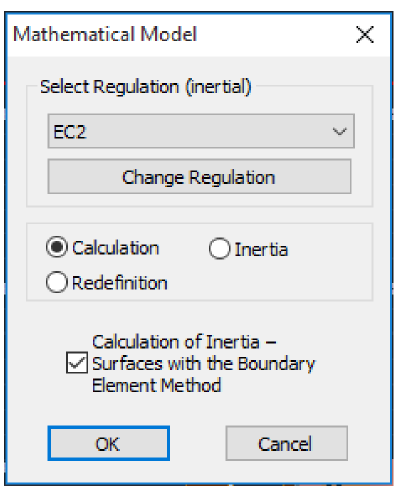


2.9 Mathematical Model Creation:

As soon as you complete the modifications of the physical model (copy-delete elements) and the elements insertion, you move on to the creation of the mathematical model.

With the “Calculation” command, the program calculates and produces the mathematical model of the project (nodes and beams). This means that all the physical elements obtain their corresponding mathematical representative.

Select the command and the dialog box opens:



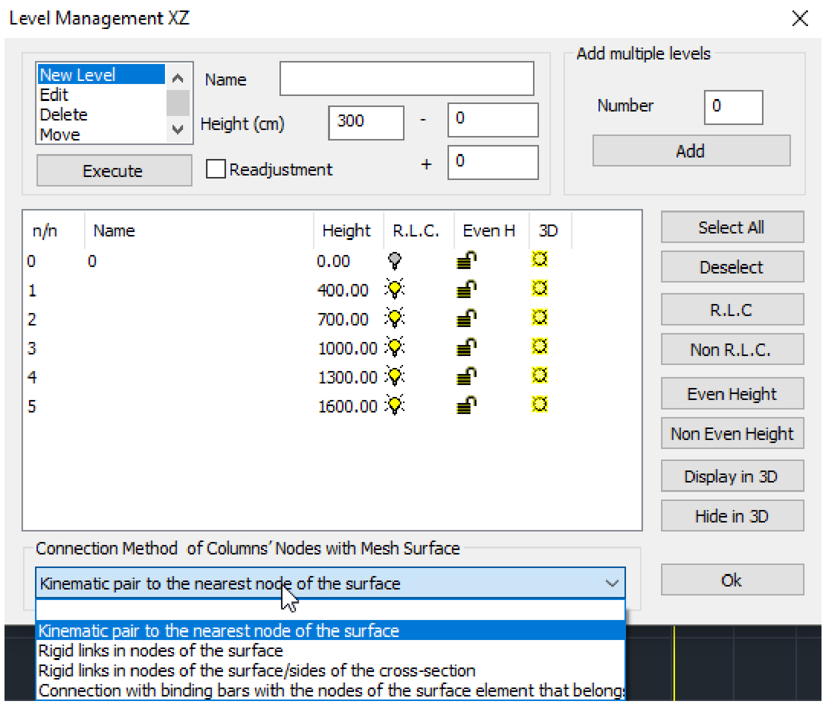
Select a regulation to calculate the modulus of elasticity accordingly.

⚠ *In case that you want to change the regulation after the creation of the mathematical model, select the new regulation and click “Change Regulation” to update the modulus of elasticity.*

⚠ *SCADA Pro enables the connection between beam and plate elements in the same modeling environment. The connection between beam elements and the corresponding plate element node is performed automatically with the above mentioned command.*

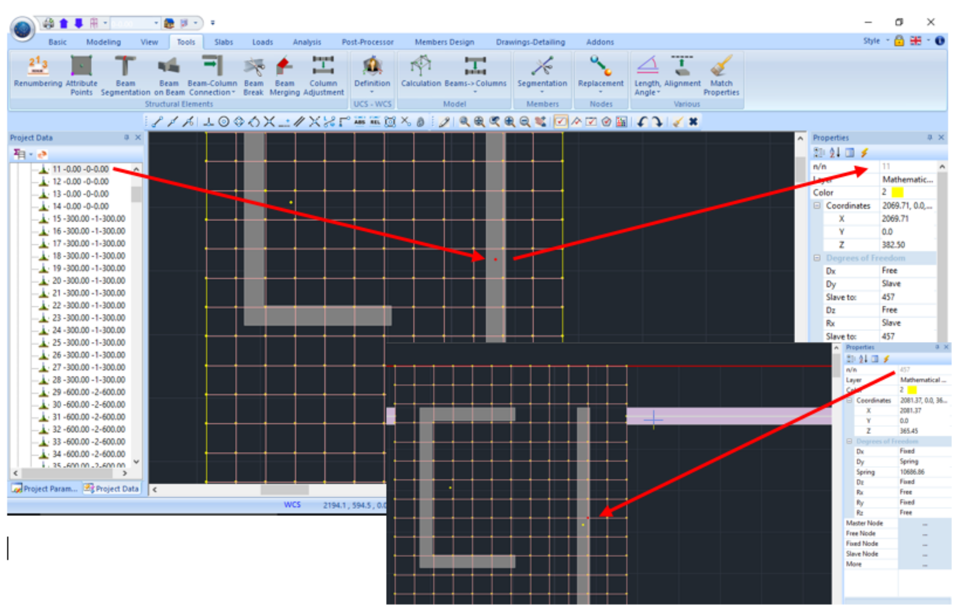
Connection between column nodes with mesh surface

SCADA Pro enables the connection between beam and plate elements in the same modeling environment. Therefore, there is an need for their enchain.



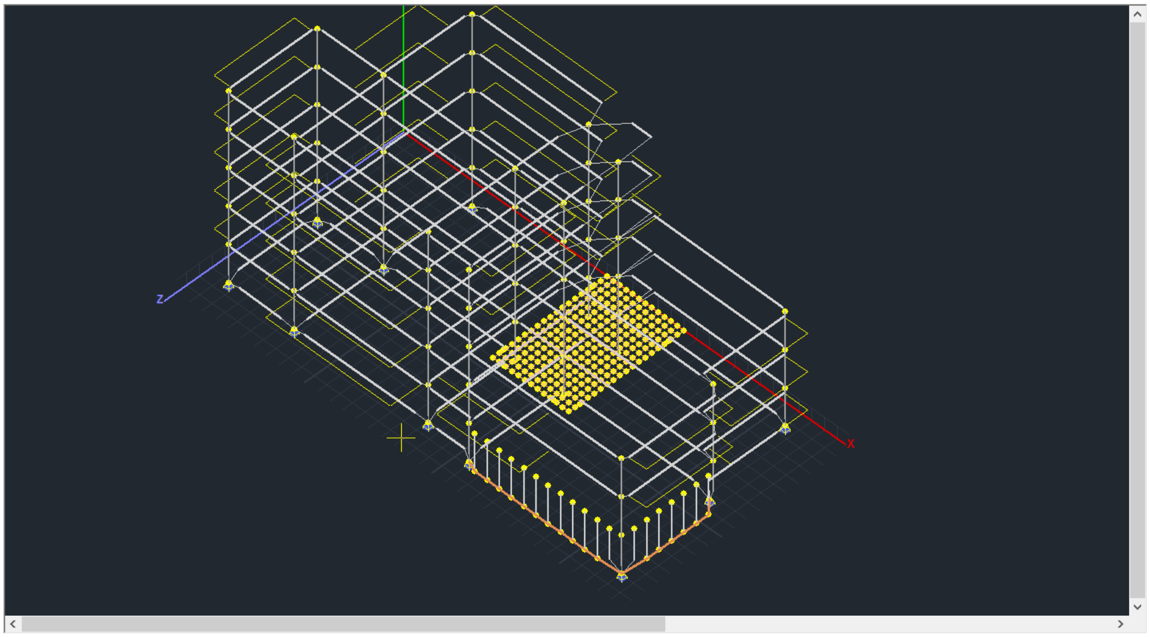
In the bottom side of the window there is an option of the way to connect the column nodes with the mesh surface elements for the selected level, by choosing one of the following ways, that is, connecting the nodes either with a kinematic pair or with binding bars.

Select the column base node inside the raft area (11 node) and notice the automatic slave relation from the nearest surface node (457 node).



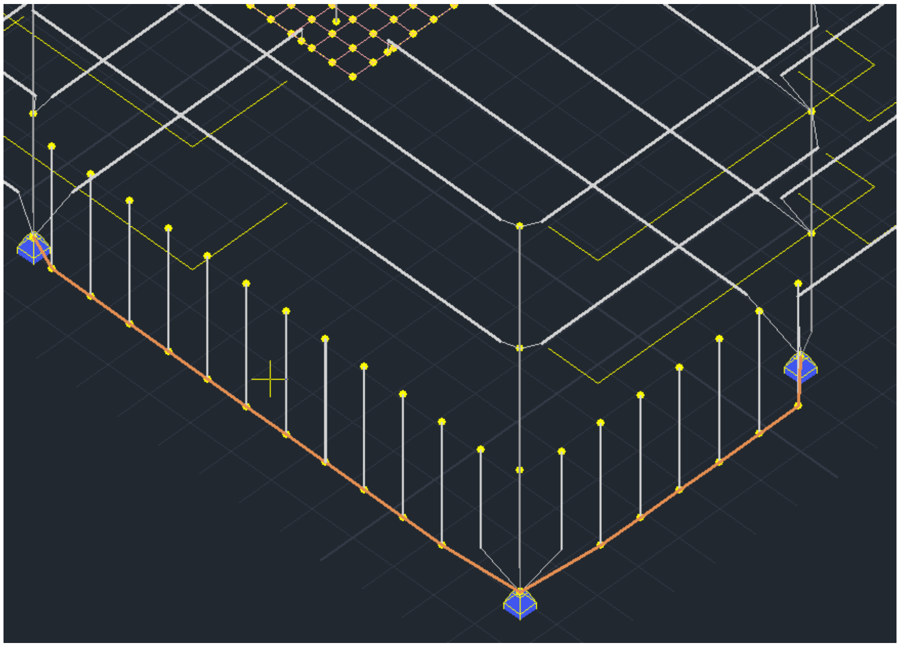
2.10 3D Representation:

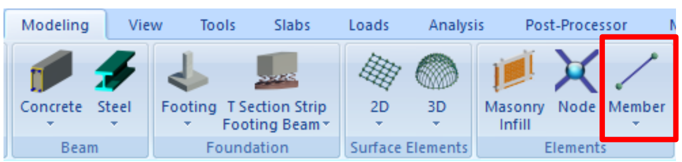
After creating the mathematical model, you are allowed to view the 3D representation of the model as well as use the rendering options.
In this way you can modify in real time the mathematical elements, the nodes etc. For instance, you can create a slope and insert mathematical members to connect the unconnected nodes of the basement walls.

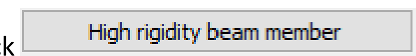


2.11 Basement walls nodes connection – High rigidity beam member:

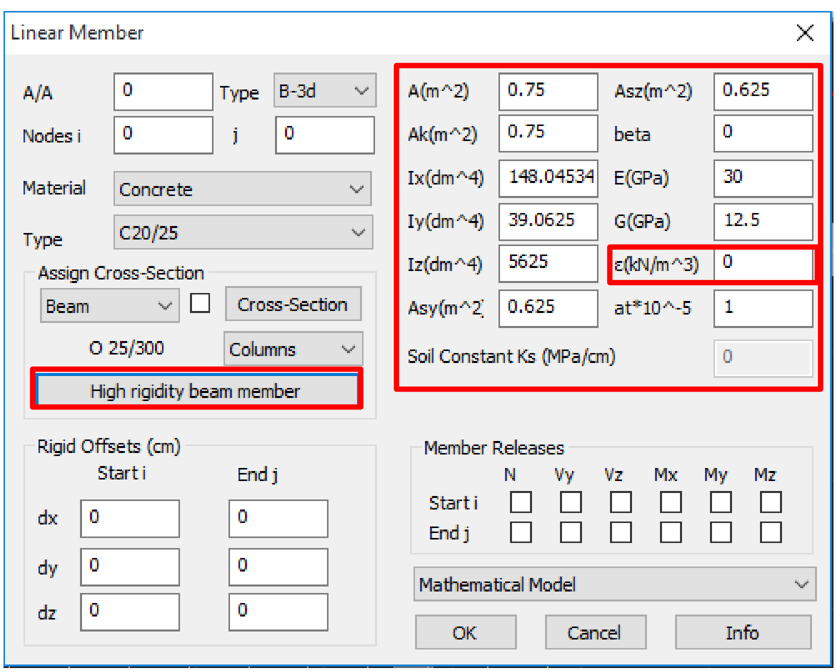
The basement walls simulation through the "Beams to Columns" command is completed with the connection of the column nodes in level 1 (in level 0 the connection is already performed through the insertion of the footing beams).



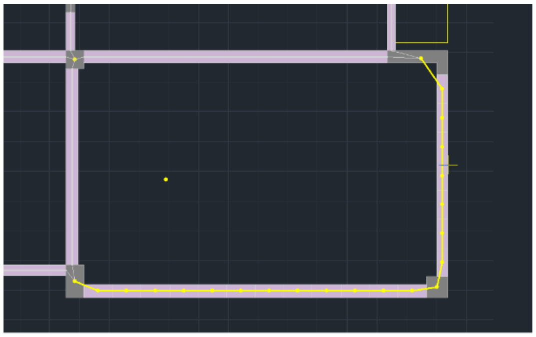


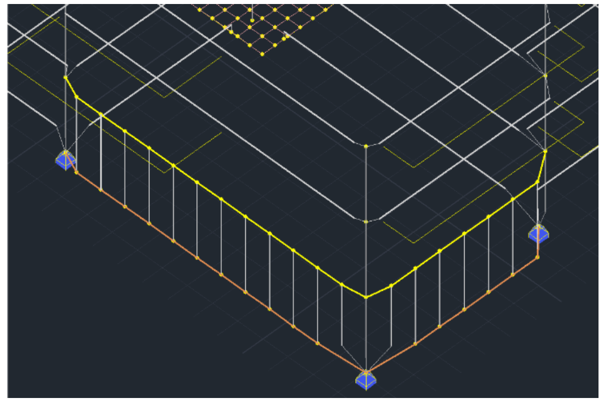
Select the "Member" command and in the dialog box click 

. The parameters' fields are automaticaly completed with the characteristics of a high rigidity cross-section; zero specific weight and without an assignment of a physical cross-section. This element is necessary for the connection of the basement wall column elements.



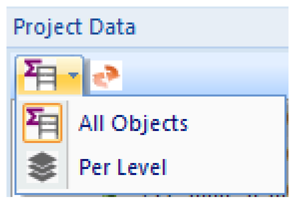
Select “OK” and insert from node to node (or with the help of the window selection method) the members:



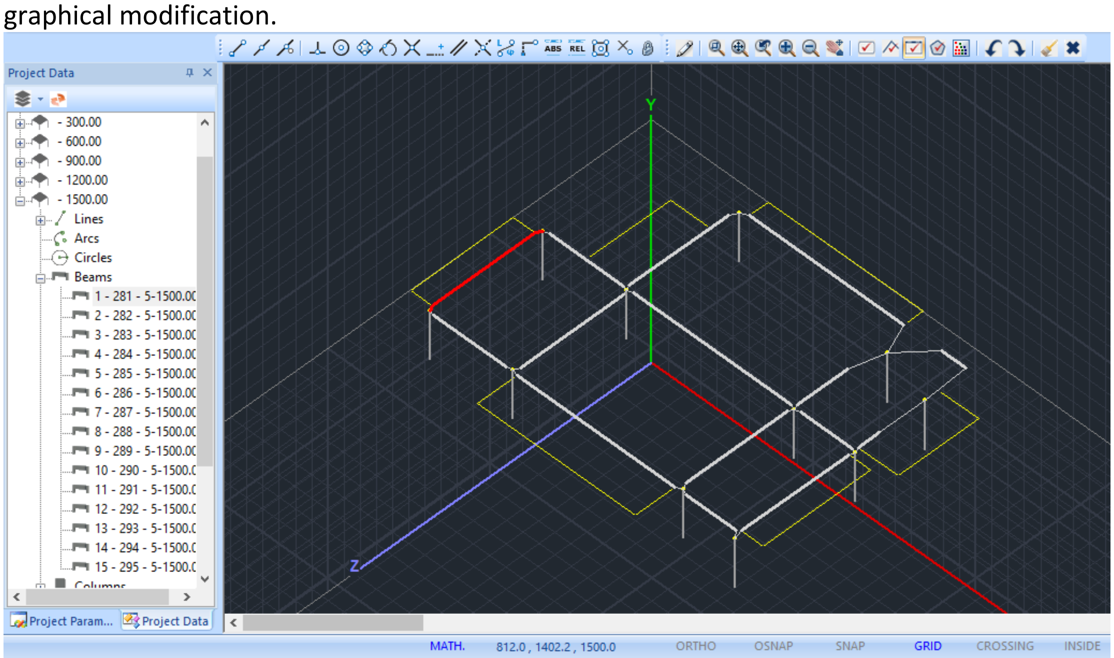


2.12 How to create a slope:

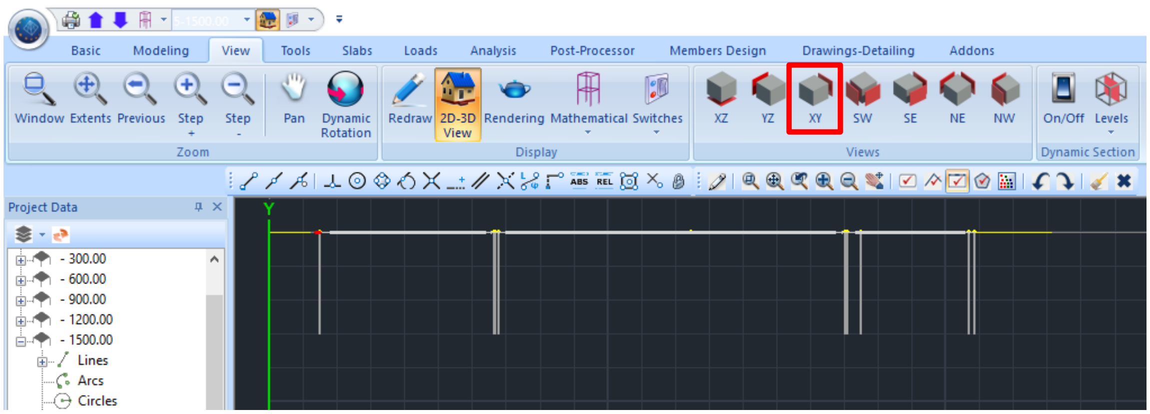
To create a slope simply, take advantage of the 3D representation of the model by selecting the command .

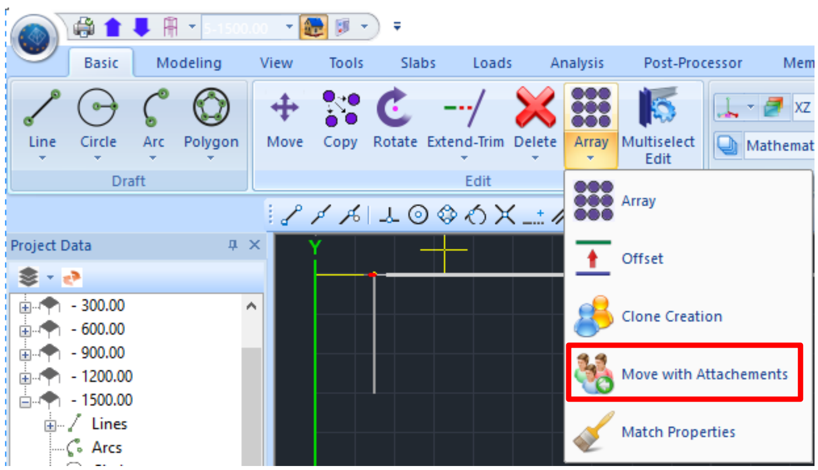
Also, on the left select  “per level”

Open the group Level 3 and the subgroup Beam Members.
Select the member whose slope will be modified:
The member is colored red while level 5 is isolated helping in this way the localization and the graphical modification.



Through the "View" unit activate the "XY" view:

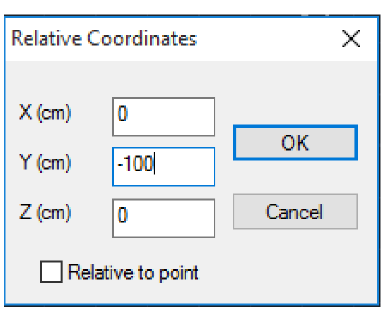




Through the "Basic" unit select the "Move with attachments" command, activate the select "with window" option enclose the node and right click.

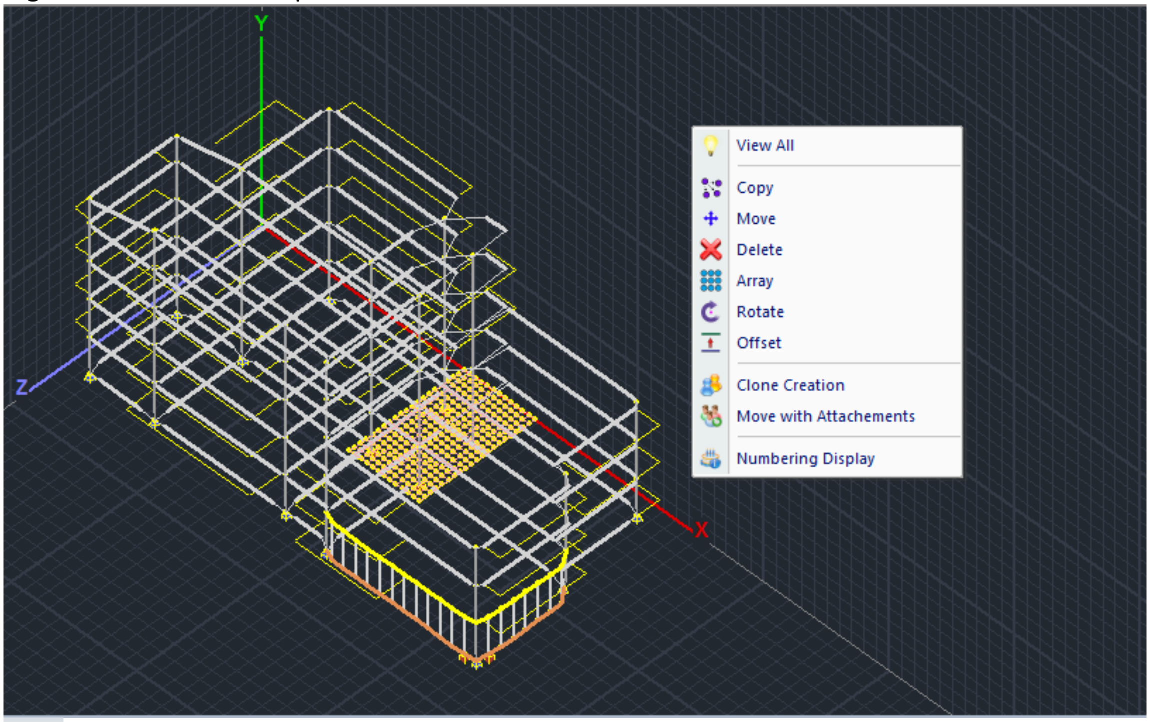
In this way select all the nodes that are located behind the selected node on the XY plane.

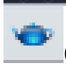
Left click on the node and select the command .

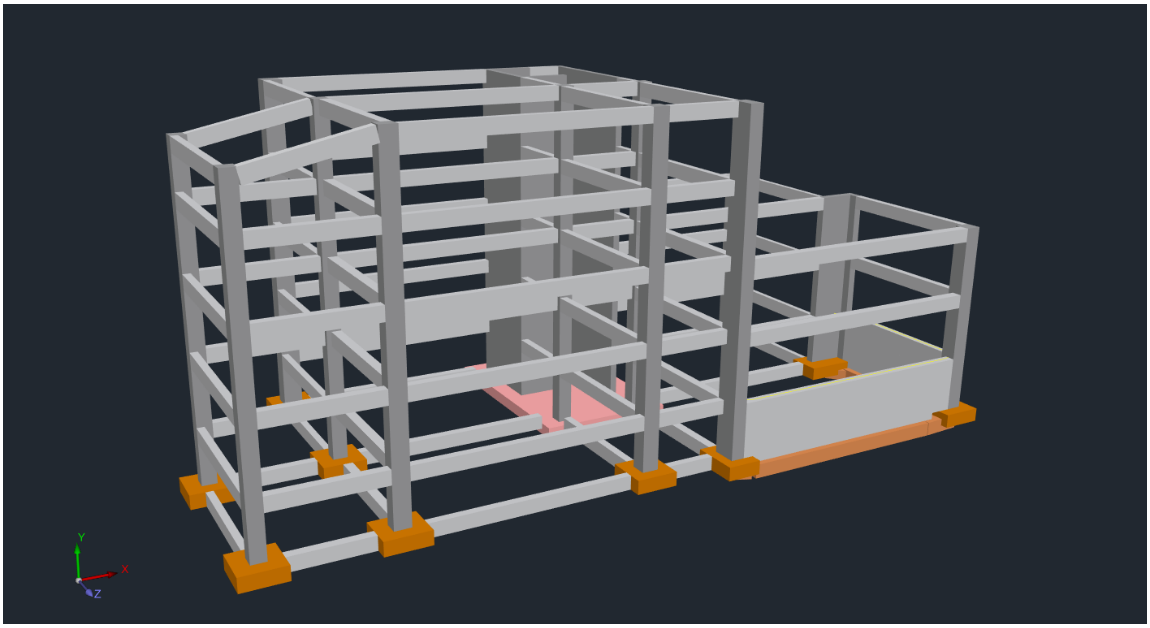


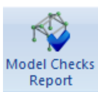
In the "Relative Coordinates" window, fill in cm the relative displacement and click "**OK**". The nodes descend automatically and the slope is created.

Right click on the desktop and “View All” to view the whole model.

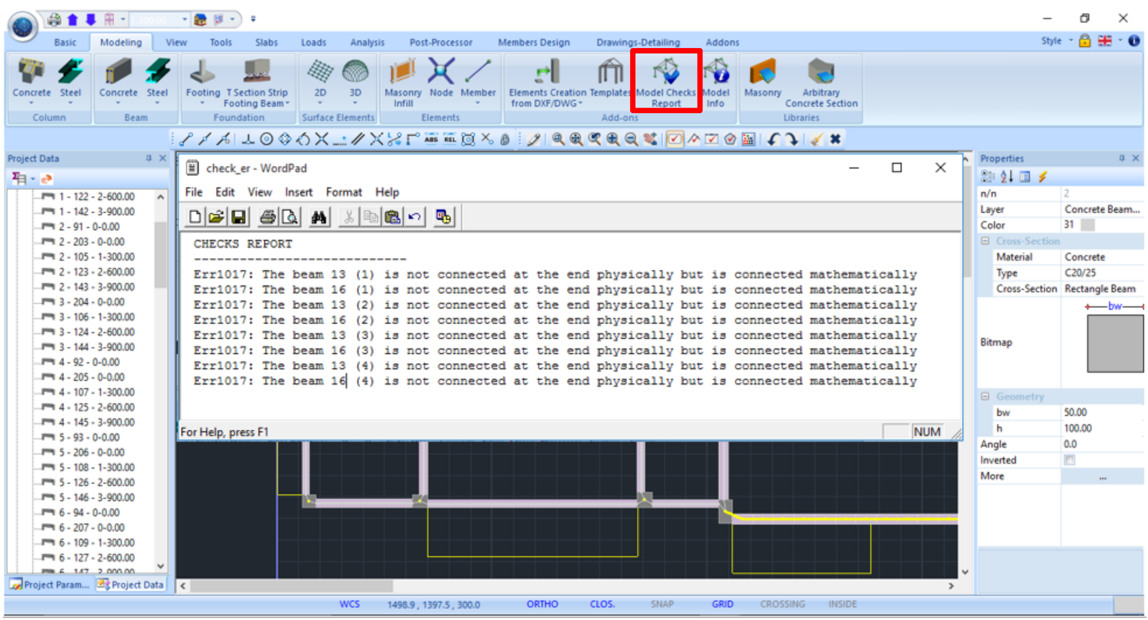


Open the rendering to view your model in final state:





Select the “Model Check Report” to locate any errors created during the modeling process:

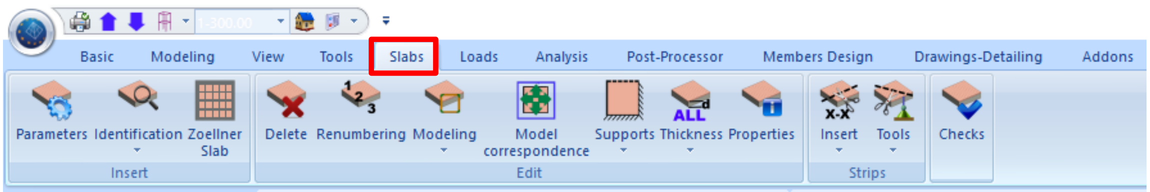


The window that opens, displays for each error, a sort description, the number-ids of the members that concern the error and possible warnings.

⚠ *The Error1017 of the current example refers to the “Beam on Beam” support and it is not an error. In case that the model has errors, correct them using the program tools before you move to the next step.*

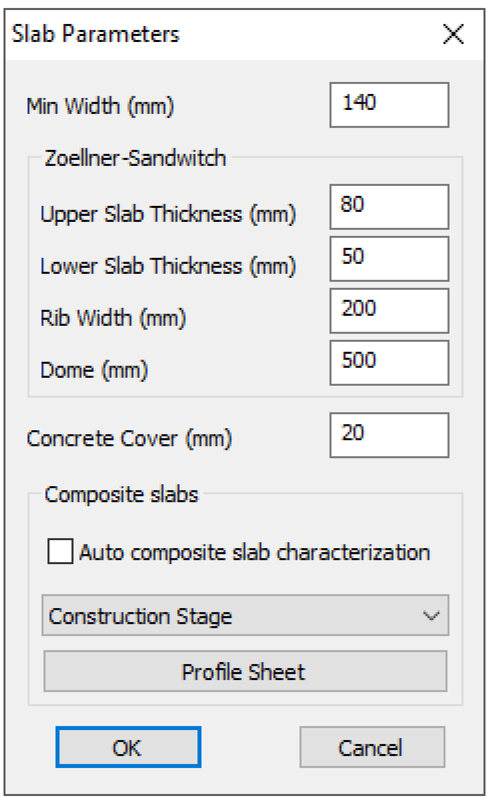
3. SLABS

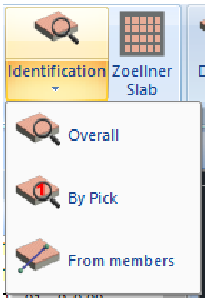
The  command, models the slabs as well but in case that you want to modify the auto-generated model you must first delete the mathematical model, thus the slabs will be deleted as well. To redefine the deleted slabs (or even to define new ones) you must use the commands of the "Slabs" unit.



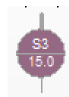
3.1 How to define solid slabs:

To define the slabs set in 2D view each level and:
In the "Insert" field, select "Parameters" and fill in the values of the minimum width and concrete cover in mm.





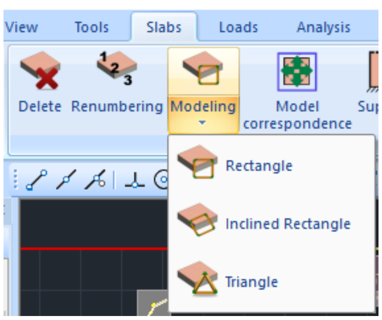
The command “**Identification**”>“**Overall**” identifies all the closed contours that exist in the current level and automatically inserts all the slabs.

After inserting a slab, the circular symbol 

with the corresponding information is displayed; the number and the thickness in cm (the greater value between the minimum you set and the one resulted from the bending resistance check), in a circle. Around the circle, lines are displayed representing the slab’s support conditions:

- Thick Line: slab continuity →fixed.
- Thin Line: slab discontinuity → joint.
- No Line: free end (case of balconies).

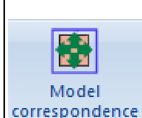
- *The sign “**?**”* in the symbol of the slab, indicates that the slab has not rendered correctly and needs "Modeling". For this purpose you need to define a new slab, the shape of which can be rectangular, rectangulal with a slope, triangular or equivalent to the original.



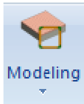
Inside the “**Modeling**” command group select one of the three commands.
With a left click inside the area of an arbitrary shape slab, you select it and define the equivalent slab:

- For rectangle shapes: Left click on the first top, move the mouse diagonally up to the second top drawing a rectangle and left click again.
- For inclined rectangle shapes: Left click on a slab side to define the direction of the equivalent inclined rectangular slab. Left click on the first top, move the mouse diagonally up to the second top drawing a rectangle and left click again.
- For triangle shapes: Left click on the three sides of the equivalent triangular slab.

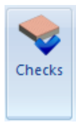
Last, define the correspondence between the sides and the tops of the equivalent slab and the ones of the real slab. The members and the lengths of the sides of the physical model match the ones of the mathematical model with this process.



Select the “**Model Correspondence**” command and click on the slab. The rectangle or triangle that appears is the mathematical model of the equivalent slab.



Select one side of the mathematical model of the slab. An X appears on it. Left click on the corresponding physical model member (in the middle of the member a circle that takes the color of the corresponding mathematical member is displayed). Right click to complete and left click again to continue with the rest sides of the mathematical model of the slab. Finally assign to each vertex of the equivalent slab (symbolized with a triangle) the corresponding physical point to make the reduction of the length of the sides of the physical to the mathematical model. Consequently, the loads of the equivalent slab will be distributed to the real lengths of the physical members. For the assignment first select the top of the mathematical model and then left click on the new location. Repeat the process for the remaining three vertices of the mathematical model without using the right mouse button.

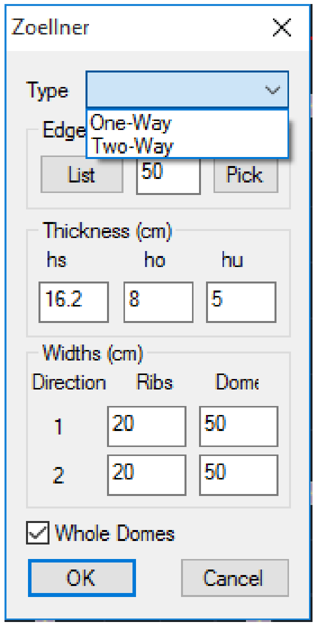
With the 

command a TXT file that contains the results of the slab's design checks for all slabs of the current level opens. *In case of errors you must correct them before moving on. Repeat the command after the load assignment as well, to be sure that no errors were located*.

3.2 How to create a Zoellner Slab:

From the "**Insert**" field, select "**Zoellner Slab**" and click on the slab.

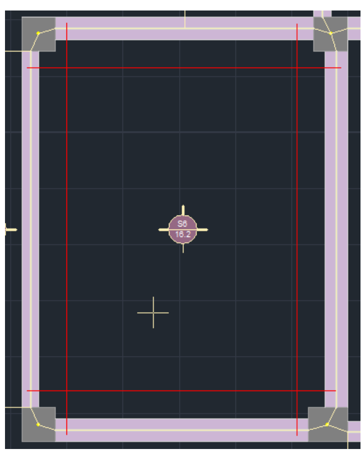
In the drop-down list "Type", select if the slab is connected in one or two directions and define solid zones' widths (cm). Click the "Pick" button and left click on the side of the beam considered as an outline of the slab. Then, the boundary of the solid zone will be placed in parallel with the beam at a distance equal to the width that was defined previously. The line is drawn (boundary of the solid zone) and with left click the direction is indicated. Repeat the same procedure for every solid zone.



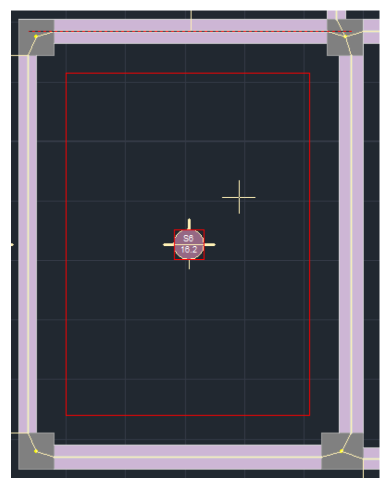
To define a solid zone of different width, first right click to reopen the dialog box. Modify the width and continue as previously described to place the very last of the solid zones.
Right click to open the window and complete it by typing thickness, ribs and domes widths, and click "**OK**". (*6)

(*5) hs: type the slab's total thickness (cm).
ho: type the up side solid slab's thickness (cm).
hu: type the down side solid slab's thickness (cm) for Sandwich slabs. Otherwise type 0.

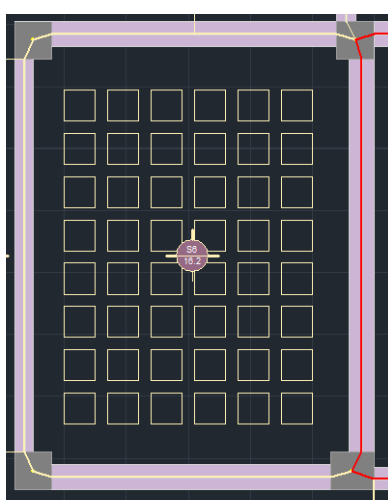
(*6) Select the checkbox next to "**Whole Domes**" to receive only whole domes.



Click "OK" to display the mathematical model of the selected slab. Then the program asks you to define Direction 1 (the side of the slab, which will be parallel to the beam of the first direction). Select the side of the slab's model and the gap with the defined geometry is placed automatically in the center.

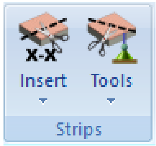


To define where to start putting the domes, first click on a dome's vertex and then on a slab's vertex. The solid slab is automatically converted to a Zoellner slab.



The image on the left, shows the result of the procedure above.

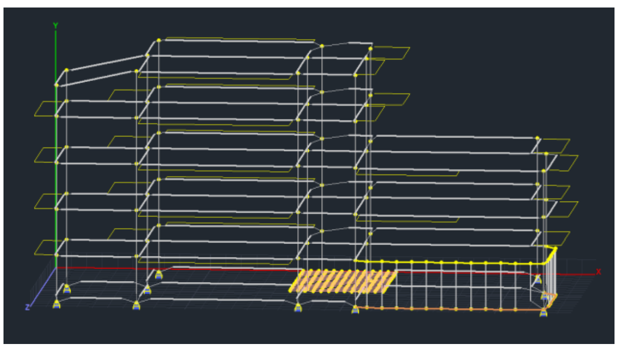
3.3 Define Slab Strips:

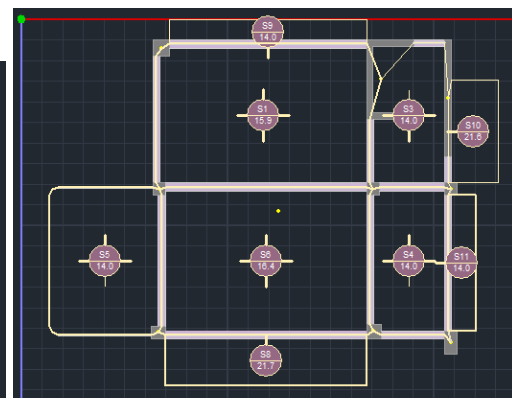


Strips' input is essential to the analysis and design slabs' steel reinforcement. They are the "guides" for the design of the steel reinforcement and the diagrams display.
From the "**Slabs**" unit, "**Strips**" command group select "**Insert**" along **X** or/and **Z** and define the strips with a left click. The direction of the strip identifies the steel reinforcement main bars direction.

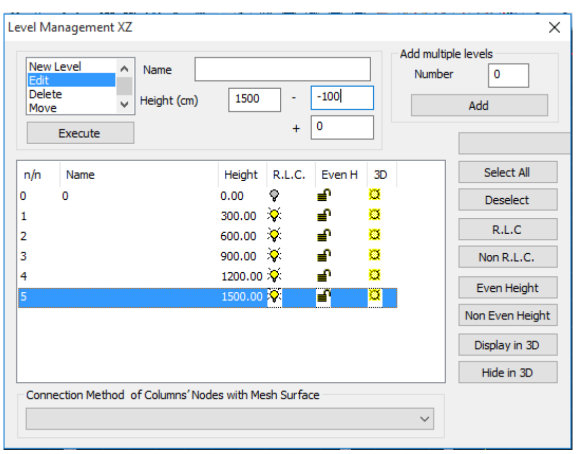
3.4 In case of inclined slabs:

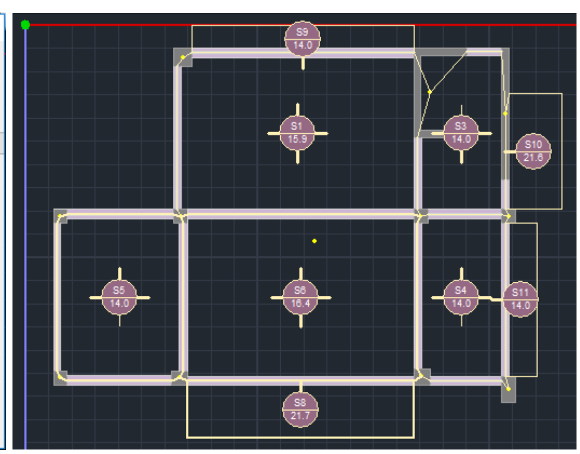
As in the current example, in case of inclined slabs, to achieve correct modeling, specific steps must be followed:





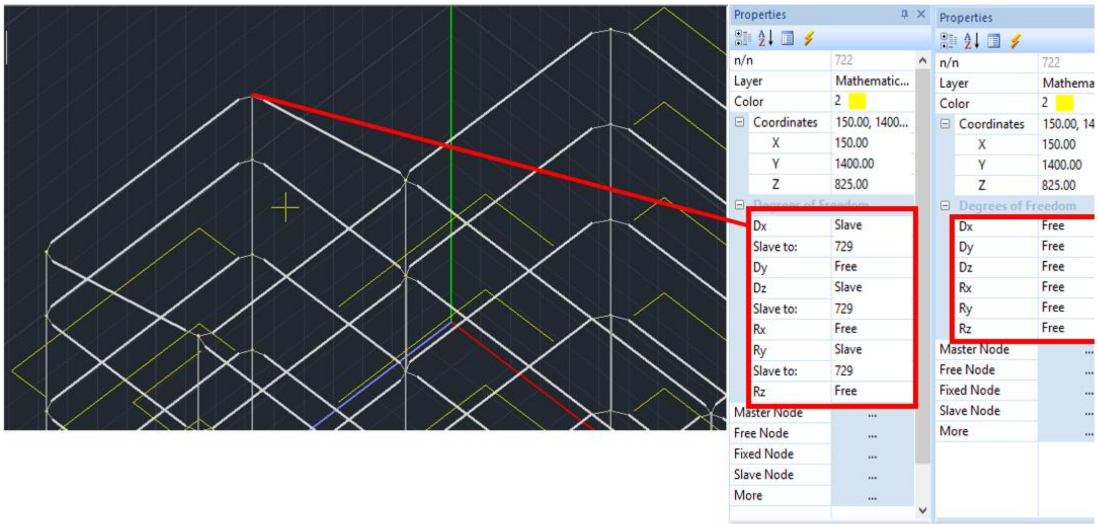
1. To insert slabs strips, all the elements that are connected to the slab, must belong to the same level. Thus, in case of inclined slabs you must define the uneven height value of the considered level:



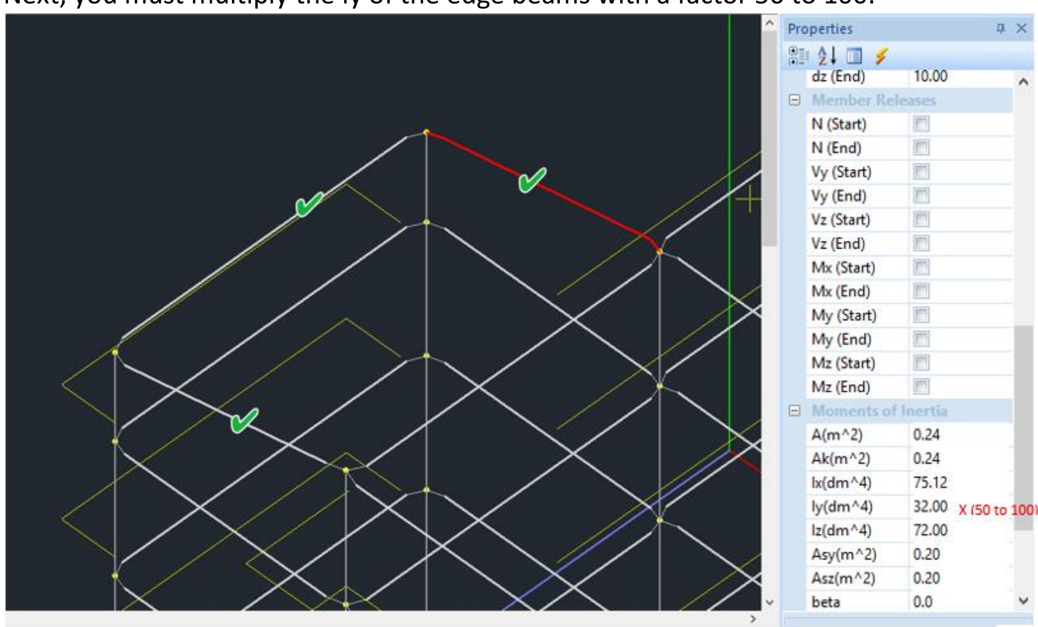


Declare the height value (for example - 100) > Execute > Exit.

2. The nodes of the inclined slab, must be excluded from the diaphragm. Select the nodes one by one and release all of their constraints related to the diaphragm node.

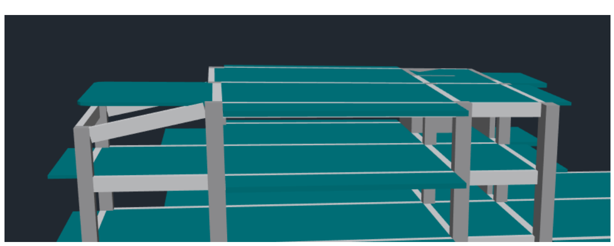


3. Next, you must multiply the Iy of the edge beams with a factor 50 to 100.



⚠ **NOTE**

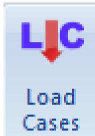
The rendering displays the slab as if it wasn't inclined. In any case the slabs are considered as if they were horizontal. Only the beams are inclined.

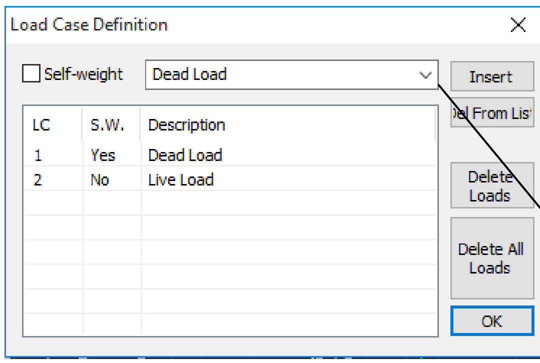


4. LOADS

4.1 How to define the loads:

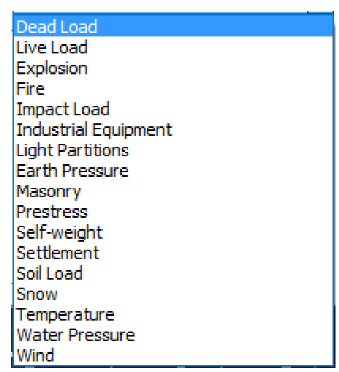
To insert loads, you must first define the load cases.
Open the "***Loads***" unit and select the "**Load Cases**" command.



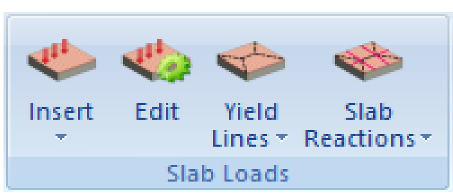


The dialog window contains by default the two basic load cases Dead and Live. The user can delete ("Delete") or define new load cases by selecting from the list any load or by typing a new name and clicking "Insert".

- Check the checkbox next to "'Self-weight" if the specific load case includes the self-weight and a ''Yes" is written in the self weight column. Instead, a "No" is written. (The self-weight can be included only in one load case, usually at dead loads)
- "*LC*" means "Load Case" i.e. "Loads".
- Click "*OK*" to save and exit.

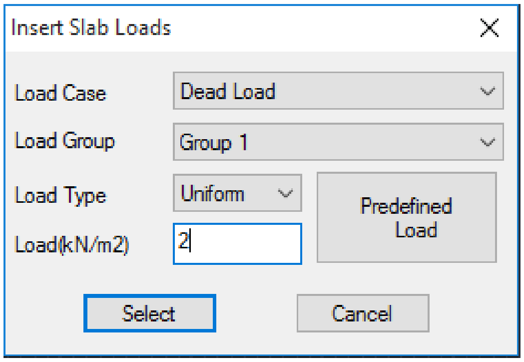


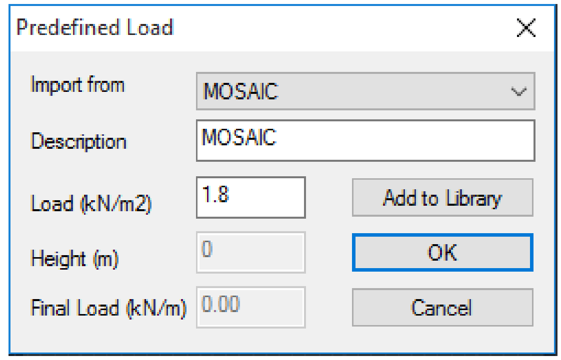
4.2 How to insert Slab Loads:



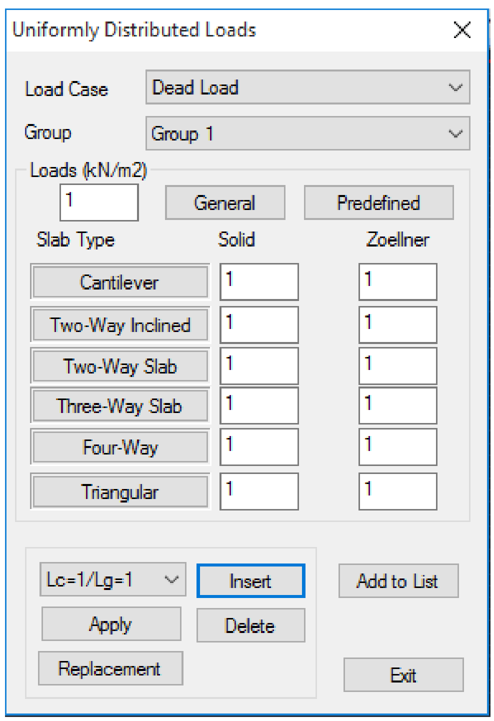
From the "**Slab Loads**" command group, select "**Insert**". You can insert loads either "Overall" for all the slabs of the current level, or "By Pick" by selecting each slab at a time.

In the dialog box define the Load Case and Group and then type the load value for each type of slab (KN/m^2). "Select" and click on the slab.





The "Predefined Load" button, contains a library with coating materials that automatically updates the assigned load value. The user can update or/and enrich the library with new materials by defining the corresponding load values.



In the dialog window that appears, fill in a load value (KN/m^2) and click the "General" button to assign this value to every slab type.

The "Insert" button, creates the loads but it won't be applied until you click "Apply". Select another Load Case and repeat the same process.

To apply the loads that you just defined click "Apply". The loads are automatically distributed uniformly on the slabs area of the current level.

The first time that you insert a load (for example dead load) after the "Insert" command you select "Apply". Next, if you want to add live load you define it and then click "Add to List".

In this example the load values that were considered are 1KN/m2 for dead loads and 2KN/m2 for live loads for all slabs.

4.3 How to assign the slab loads to the members:

After inserting slab loads, select:

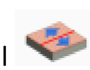
"Yield Lines": Load areas' calculation resulting from geometric partitioning of the slab, and then used to calculate the design forces for beams (slab loads which will be imposed on beams),

Calculation is automatically made by the program according to the support conditions, either Overall or By Pick.

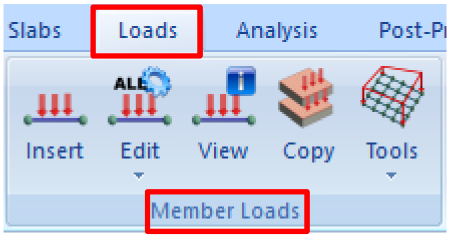
and

"Slab Reaction": To assign slab loads on beam members as reaction - Load distribution from slabs on beams and columns, based on the geometric partitioning done previously (Yield lines).

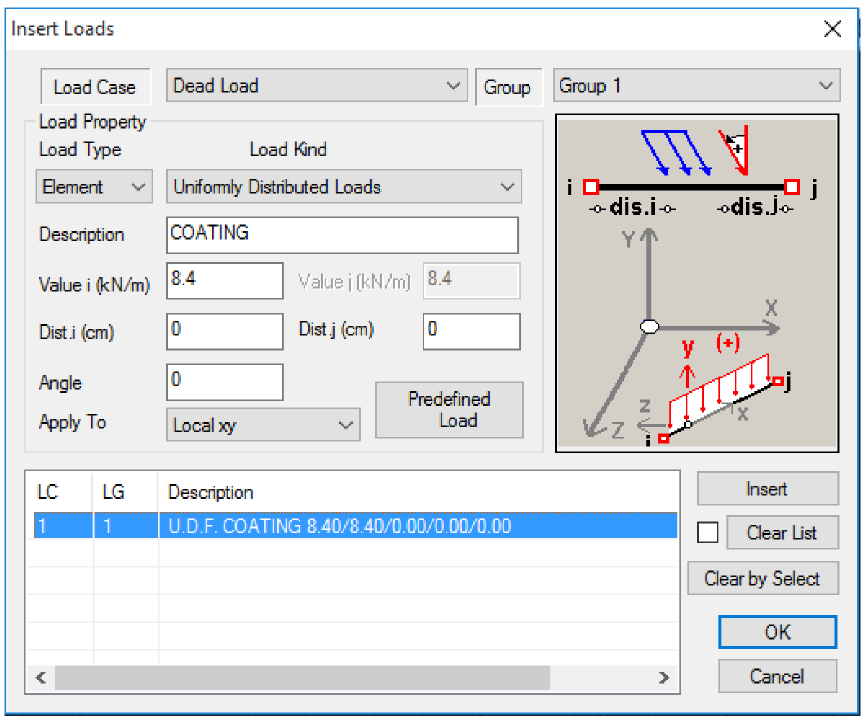
Overall: select the command (Load distribution from all the current level slabs).
By Pick: select the command and then left click inside one or more slabs (Load distribution from the selected slabs).

Equivalent: With this command, you can assign (Overall  or By Pick , respectively) the slab loads on the connected members, without considering the yield lines evaluation (rectangular and triangular areas). Instead the assignment is implemented by the convertion of the entire area corresponding to the member, in an equivalent rectangle.

4.4 How to assign loads in members:

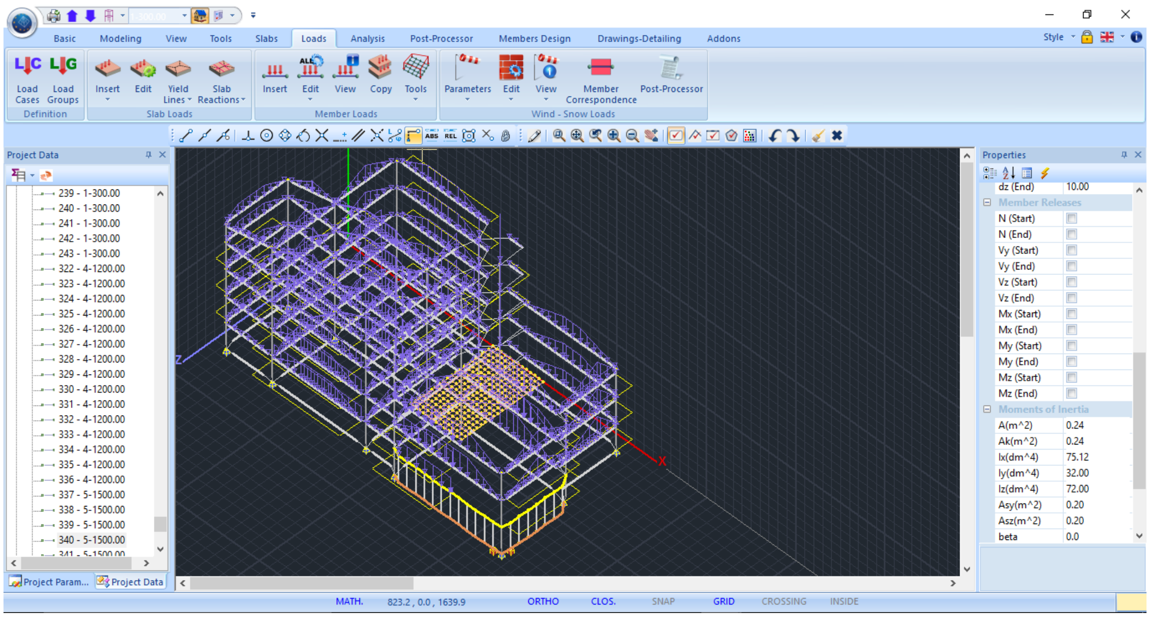


From the "**Member Load**" field, select "**Insert**" and define the elements (member, node and surface) to assign the loads to. For elements selection use . Complete selection by pressing the right mouse button and then the following dialog box appears

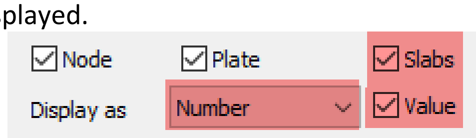


Click the "Insert" button to insert the defined load to the table and "OK" to apply the loads to the selected members.

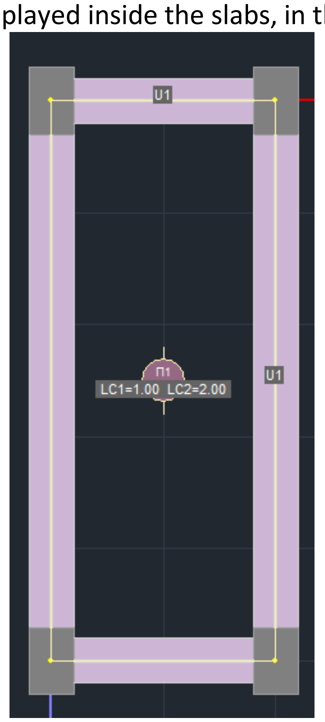
⚠ Click the  command to display the loads for all elements, in 3D view as vectors, with or without values, or in 2D view as number.



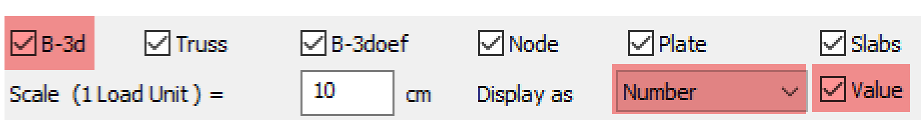
You can choose a vector or a number to appear. The vector appears only in the three-dimensional mathematical model. If you ticked the "Value" option, then graphical values of the loads with the vectors are also displayed.

Also, by choosing 

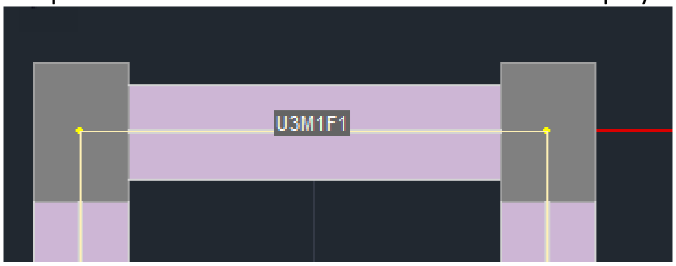
, the values of the slab loads are displayed inside the slabs, in the 2D display.

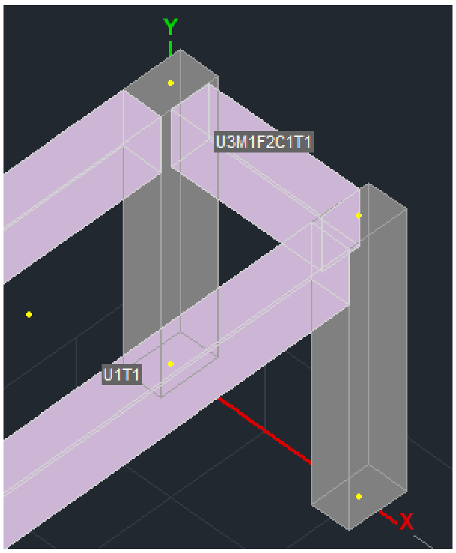


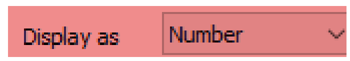
Respectively for members, with B-3d selected,



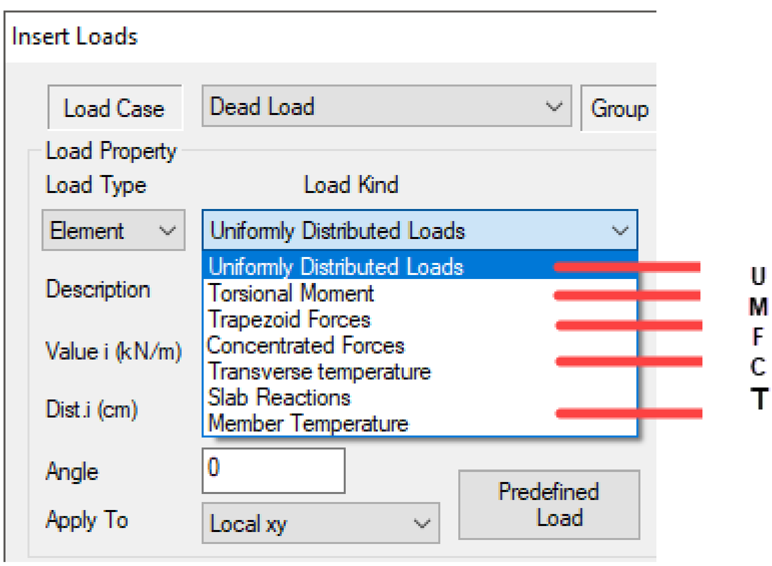
the presence of loads in letters and numbers is displayed on the member,



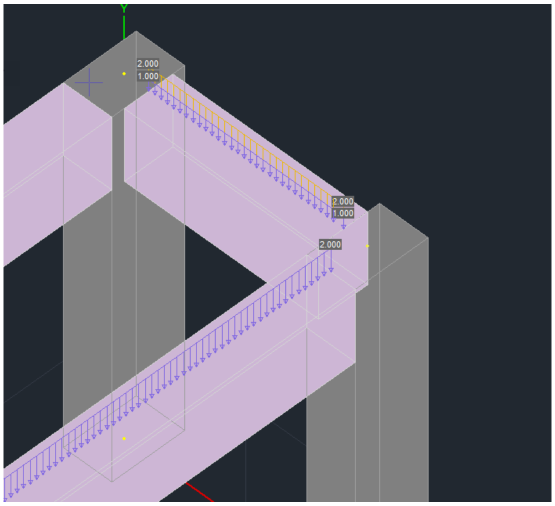


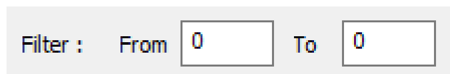


Depending on the type of the load (U,M,F,C,T) :



And the number indicating how many loads of that type exist



Finally, in the option Filter 

you can define a value range for the loads you wish to appear.

In this example a value of 8,4KN/m for dead loads was defined for every beam of the perimeter in 1,2,3 and four levels.

5. ANALYSIS

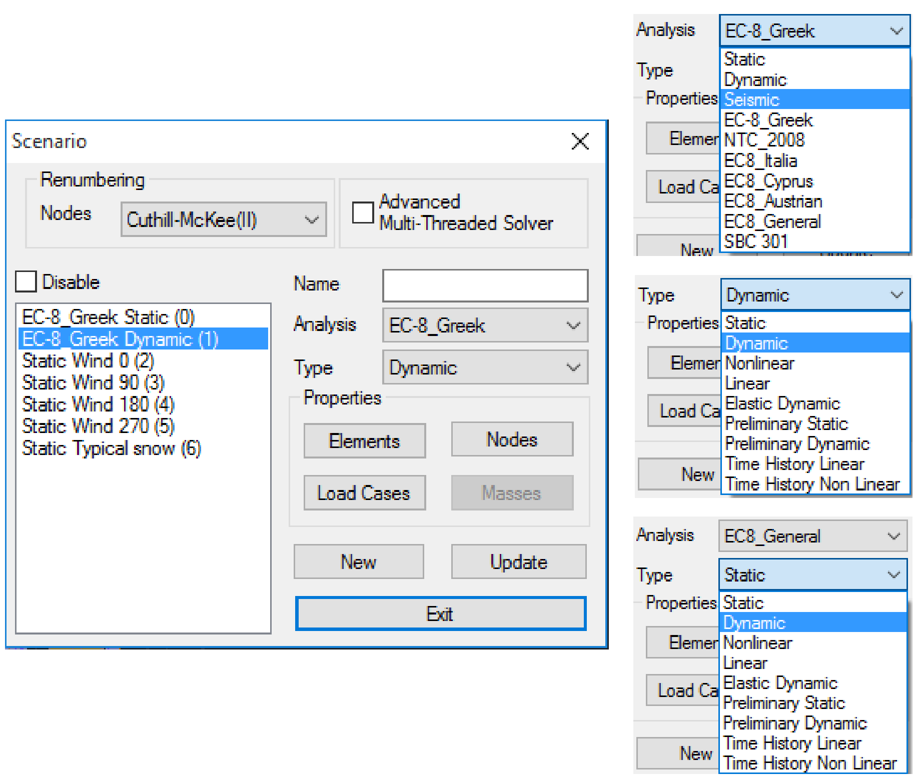
As soon as modeling and loads' assignment have been completed, the "Analysis" of the structural member, for the design of the structure follows, based on the provisions of the current design codes, to get the results, the loads combination and the final checks.

5.1 How to create a new analysis scenario:

The command group "Scenario" allows sceneries creation (choosing regulation and type of analysis) and implementation.

Press "New" and in the dialog box you can create analysis scenarios by choosing different design regulations and methods of analysis. By default there are two scenarios based on the selected "language" codes (including local Annex if there are any, or "EC-General" if there are not)

⚠ *Predefined scenarios are created according to the Rule and Attachment option you made at the beginning, within the General Parameters window that opens automatically immediately after the file name is defined.



Select the design code from the "Analysis" list and the analysis method from the "Type" list and click New to create a new analysis scenario. Optionally, type a name.

Select among the possible scenarios provided in SCADA Pro:

For Greece:

LINEAR – NON LINEAR METHODS

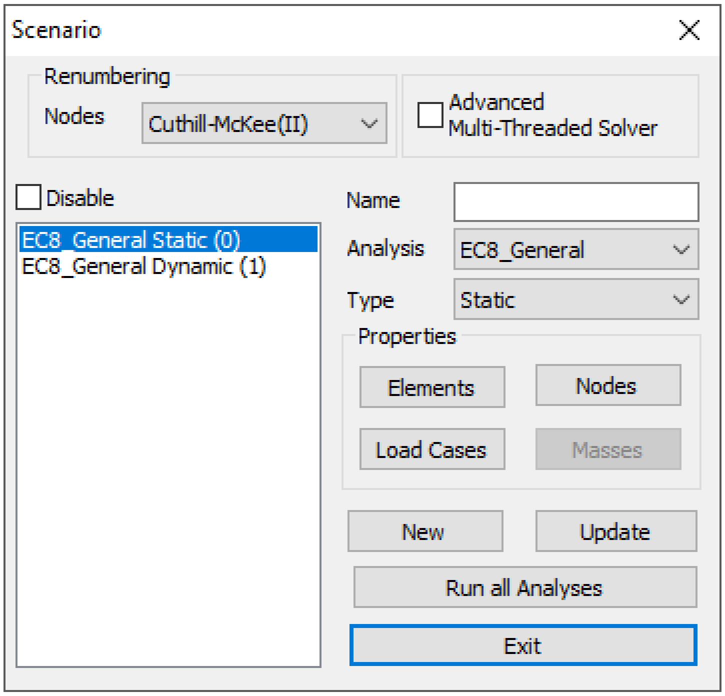
- EAK Static	Simplified spectral analysis according to EAK
- EAK Dynamic-eti	Dynamic spectral analysis according to EAK
- EAK Dynamic	Dynamic spectral analysis (masses displacement) according to EAK
- Old 1959-84	Seismic analysis according to 1959 Regulation
- Old 1984-93	Seismic analysis according to 1984 Regulation
- Static	Static Analysis without seismic actions
- EC 8 Greek static	Static analysis according to Eurocode 8 and the Greek Appendix
- EC8 Greek dynamic	Dynamic analysis according to Eurocode 8 and the Greek Appendix
- EC 8 Greek Preliminary Static	Static Preliminary analysis according to KANEPE
- EC8 Greek Preliminary Dynamic	Dynamic Preliminary analysis according to KANEPE
- EC 8 Greek Time History Linear	Static analysis according to Eurocode 8
- EC 8 Greek Time History Non Linear	Dynamic analysis according to Eurocode 8
- EC 8 Greek NonLinear	Nonlinear analysis according to Eurocode 8 & KANEPE.

For other countries:

LINEAR – NON LINEAR METHODS

- NTC 2008	Seismic analysis according to the Italian Regulation 2008
- EC8 Italia	Seismic analysis according to Eurocode 8 and the Italian Appendix
- EC8 Cyprus	Seismic analysis according to Eurocode 8 and the Cyprus Appendix
- EC8 Austrian	Seismic analysis according to Eurocode 8 and the Austrian Appendix
- EC8 General	Seismic analysis according to Eurocode 8 with no Appendix (enabled typing values and coefficients)
- EC 8 General Non Linear	Nonlinear analysis according to Eurocode 8
- SBC 301	Seismic analysis according to Saudi Arabia code (SBC 301)

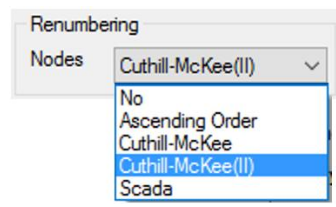
For this example a Dynamic Scenario by the Eurocode 8 will be used.

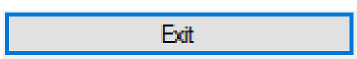


The **"Renumbering"** field includes a drop-down list with multiple options:

The choice of each option affects the computational time.

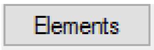
- ✓ Default choice: "Cuthill-Mckee(II)".
- ✓ "Cuthill-Mckee" and "Ascending Order" take more time to complete the analysis, while choosing "No" is not recommended.

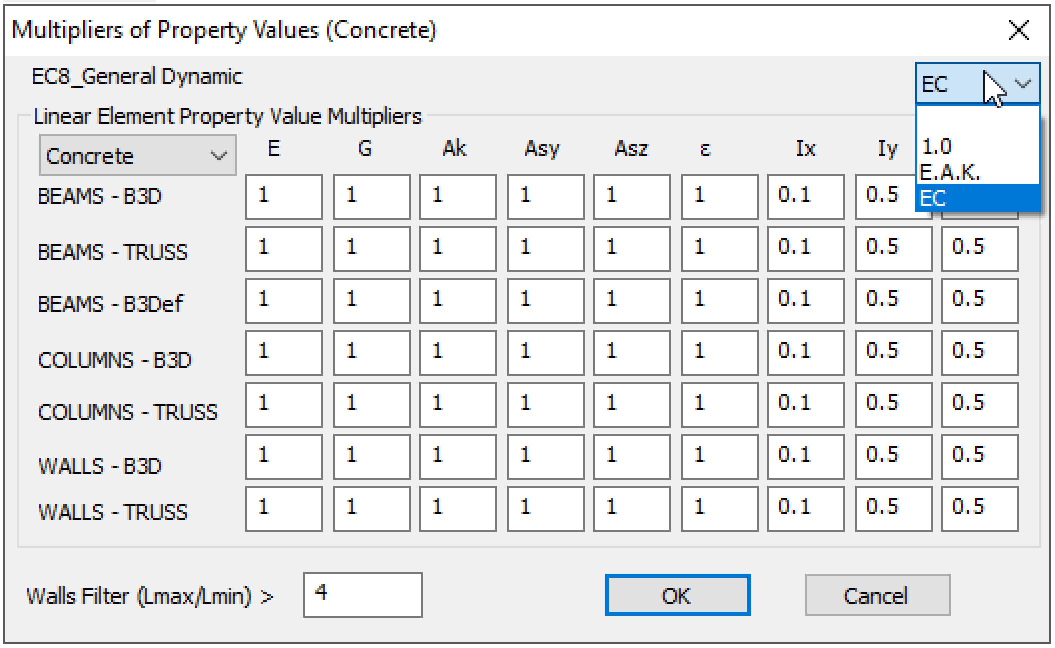


Select 

to save the scenarios and move on to the analysis.

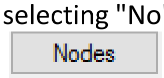
Click "Elements" to open the dialog box that contains the multipliers of the characteristic properties of the linear element, considered for the analysis:

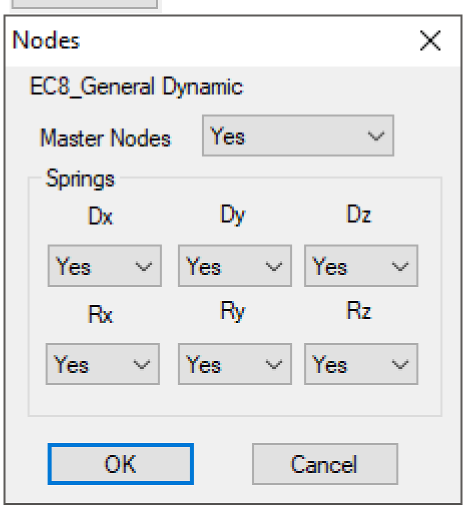




By default, the values of the multipliers are defined according to the design code, while any modification is acceptable.
If for example, you select "EC" the values of the multipliers will automatically be updated by the Eurocode provisions.

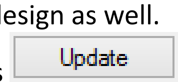
Click the "Nodes" button to open the following dialog box:
Select whether to consider slab's Master Node (FSR) by selecting "Yes" (default) or not by selecting "No"



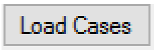


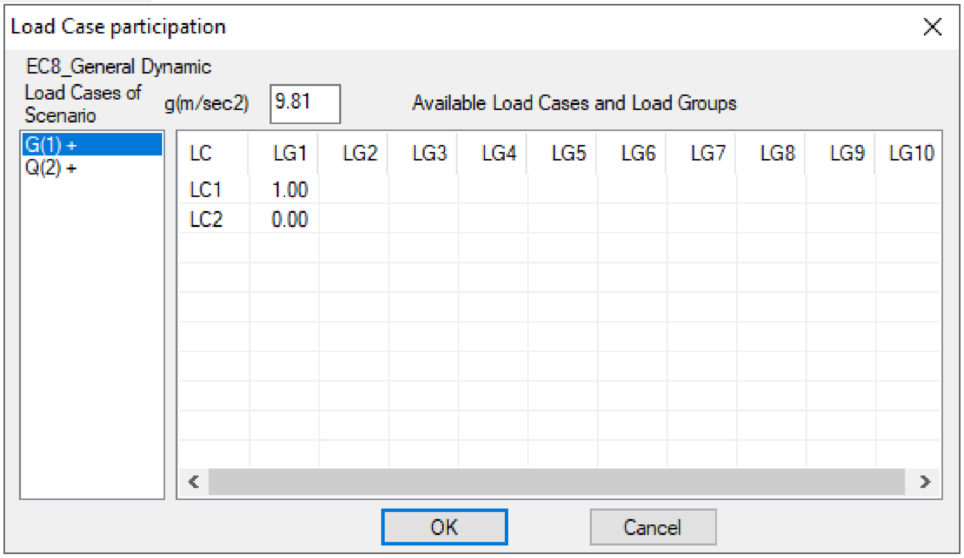
Moreover, you can choose whether to allow the corresponding displacement or rotation of the foundation's springs or not (fixed support conditions).

⚠ In cases that a Dynamic Analysis is required, if you select "Nodes" and you activate the springs ("Yes"), then you will be able to use the combinations of the dynamic analysis for the footing design as well.

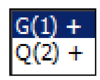
Press  to update the scenario with the performed changes.

Select "Load Cases" to open the following dialog box:



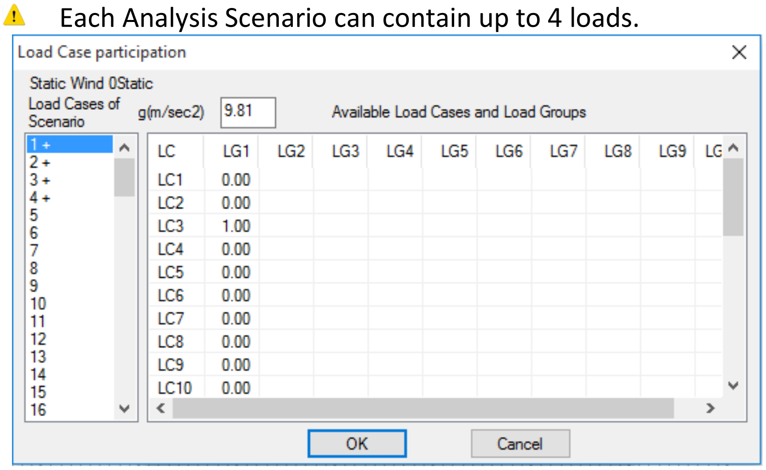


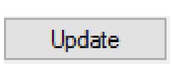
- For scenarios **considering the seismic actions**,
  - select "Dead Loads" – G(1) and type 1.00 next to LC1, under LG1 or LG2 or both (it depends on your choice to consider all dead loads together or not).
  - Select "Live Loads" – Q(2), and type 1.00 next to LC2, under LG1 or LG2 or both (it depends on your choice to consider all live loads together or not).

⚠ "+" sign located next to the load category  indicates that there is an indicative multiplier for the participation of the specific load.

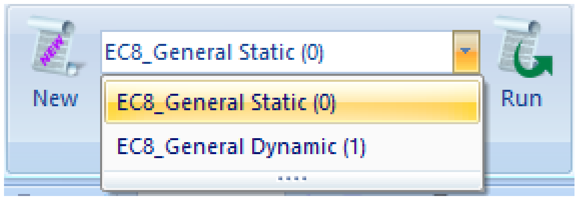
- Scenarios **without considering seismic actions** (simple static method),
  - each load case ("Load Cases of Scenario") is displayed with a number (i.e. LC1) and contains a load and its groups (i.e LG1). The load group is taken into consideration when the value in the corresponding cell is set to a value different than 0.00.

⚠ Each Analysis Scenario can contain up to 4 loads.



Click  to apply any performed modifications.

5.2 How to run an analysis:

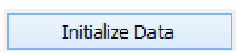


select from the scenario list the considered scenario, i.e. the scenario that will be used for the analysis.

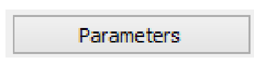
In the scenarios list, apart from the two predetermined, all the previously created scenarios are created. Choose one scenario at a time and continue with the definition of the parameters of the corresponding analysis

Click the "Run" button to open the parameters of the current analysis window which differs for:

- ✓ **EAK** Scenarios
- ✓ **Eurocodes** Scenarios
- ✓ **Non-Linear** Analysis Scenarios

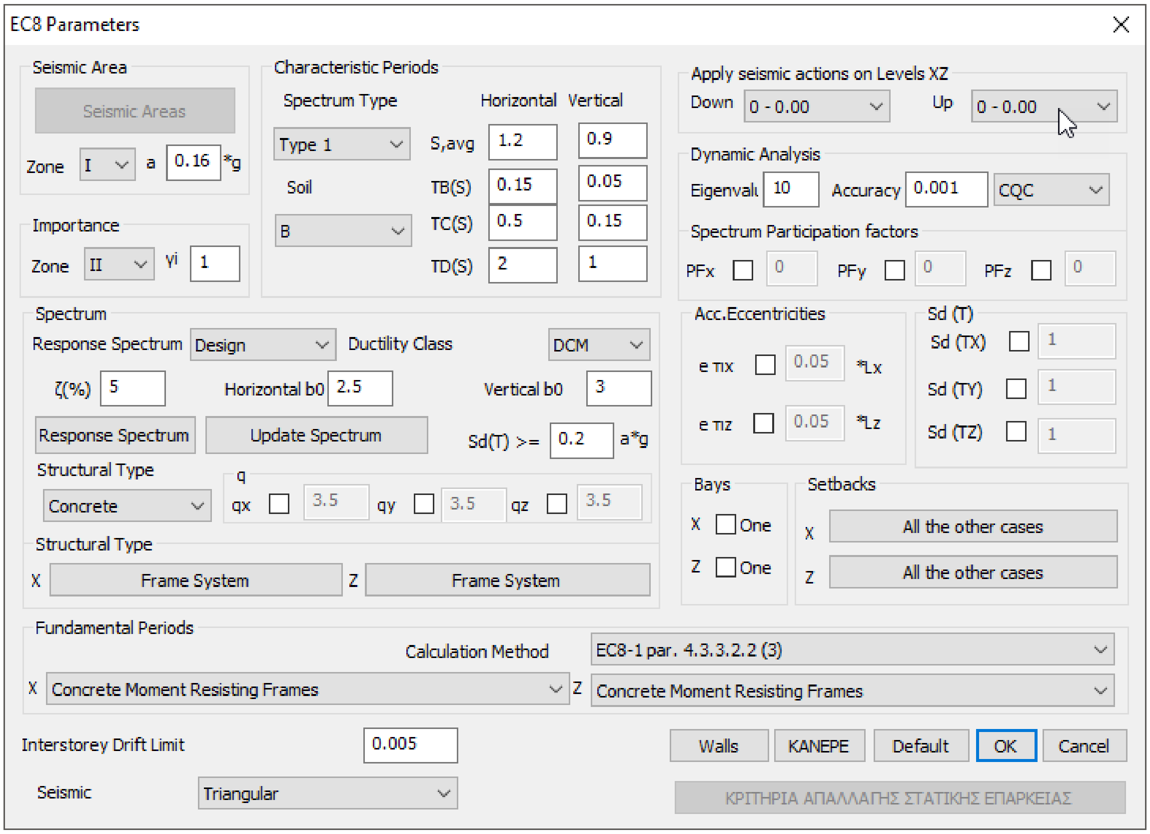
First of all, press 

to update the parameters of the current scenario.

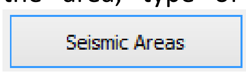
Then press 

to define the parameters of the project.

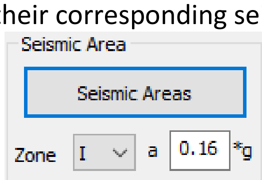
⚠ *Based on the selected scenario, the parameters dialog box differs accordingly. In this example, having selected the Eurocode 8 scenario, the dialog box will have the following format:*



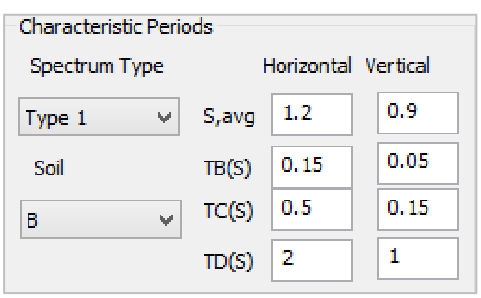
Special parameters for a specific analysis are determined in this dialog box (level of seismicity of the area, type of soil, importance of the structure etc.). By clicking "**Seismic areas**"



a file that contains a list taken by the national annex, with the places and their corresponding seismicity zone, pops up.

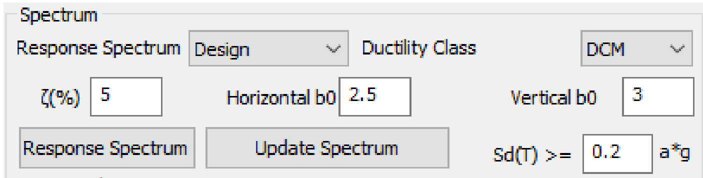


Select the considered seismic zone and the coefficient "a" will be filled in automatically.

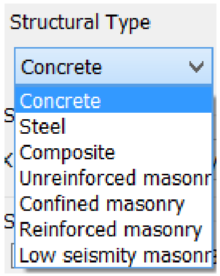


Define the Spectrum Type (for Greece Type 1) and the Soil Type so that all the coefficients for both horizontal and vertical spectrums are filled in

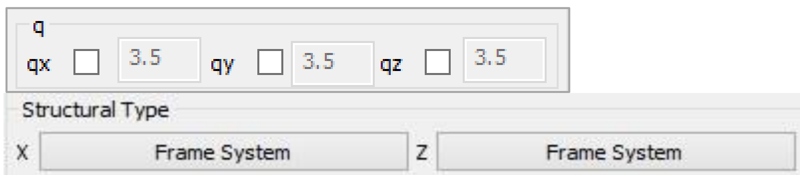
Choose the type of "**Response spectrum**" and "**Ductility class**" to suit your analysis



Choose the “**Structural Type**



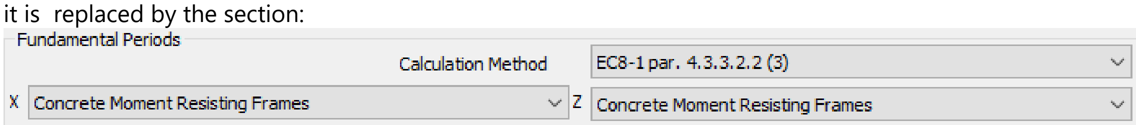
*The “**Behavior factor q**” of the structure is a result of a computation procedure. Additionally, the “**Structure type**” follows certain criteria*



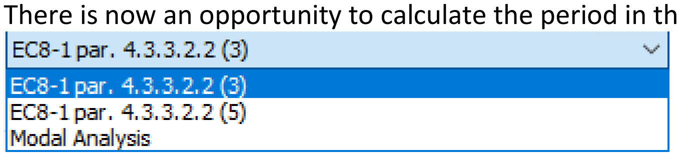
ScadaPro gives the engineer the opportunity to get rid of them and follow the procedure described in the next chapter: "How to calculate the behavior factor q

In the field ***Structure periods:***

In previous versions there was the **Structure Type** X and Z field to calculate the fundamental period. Now it is replaced by the section:

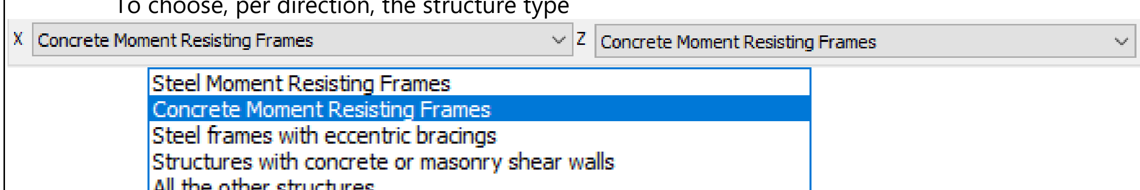


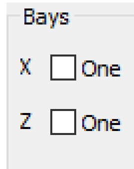
There is now an opportunity to calculate the period in three ways.



The first two are the approximate methods of EC8-1.

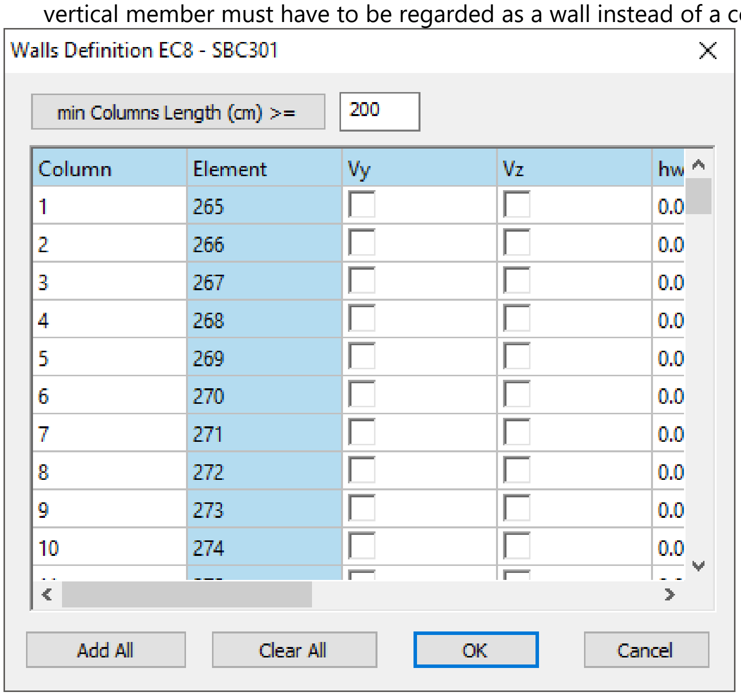
1. In the first one EC8-1 par. 4.3.3.2.2 (3) it is necessary:
   To choose, per direction, the structure type



(in case that in X or/and Z direction, the structure only consists of one frame you activate the checkbox in the field “Bays” 

)

Afterwards, choose the command “Walls” Walls to assign a value for the minimum length that a vertical member must have to be regarded as a wall instead of a column

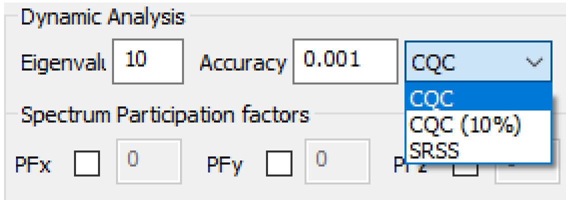


.Type the min wall length (cm) and Click the min Wall Length (cm) >= button, and automatically, all the walls are checked in each direction, so as T1 is calculated according to paragraph 4.3.3.2.2

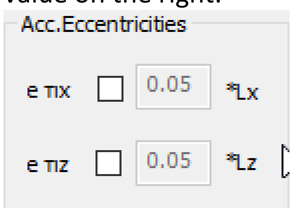
2. For the second approximate method EC8-1 par. 4.3.3.2.2 (5), there is no need to do any further action as long as it is selected.
3. The third method includes a Modal Analysis to calculate the periods.

The program takes into consideration the period which corresponds to the dominant modal in each direction. (the modal which has the biggest percentage of the activated mass)

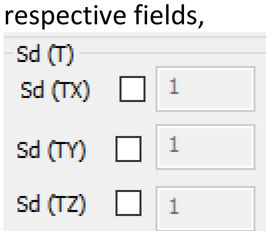
The user can increase or decrease the number of eigenvalues in case of dynamic or static analysis, as long as the calculation of the eigenvalues with Modal Analysis and the percentage of accuracy are chosen.



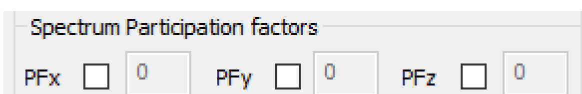
Also, there is also the opportunity to choose the method of combining the modal responses according to Complete Quadratic Combination CQC and CQC (10%)(3.6 EAK), or the square root of the sum of squared (SRSS) method.
Moreover, the results of the modal analysis for the static scenarios are included in the results of seismic action.
To modify the coefficients of the eccentricities, select the respective checkbox and type the new value on the right.

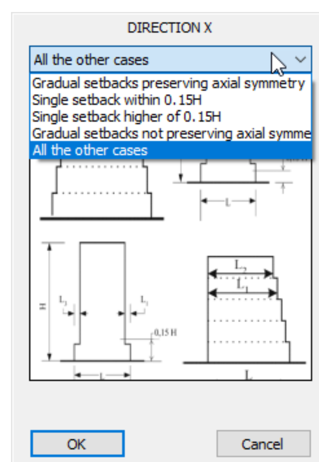


In the same way, the engineer can modify the X, Y, and Z spectrums by typing his values in the respective fields,

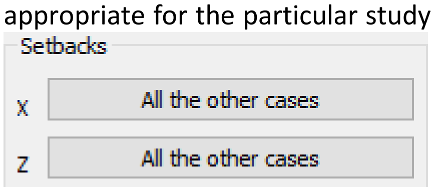


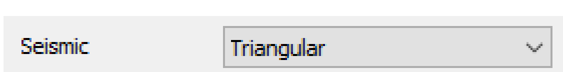
as well as the spectrum participation factors.



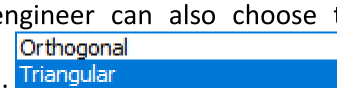


In the **Indents field**, select for each direction the case that is appropriate for the particular study and is defined by the Eurocode.





The engineer can also choose the ***Type of Distributionς*** of the seismic force between two options.

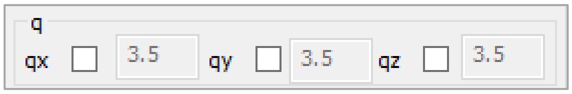


Method of calculating the behavior factor q

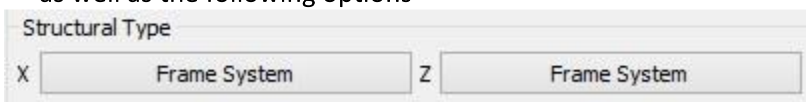
According to the Eurocode *the "**Behavior factor q**" of the structure is a result of a computation procedure. Additionally, the "**Structure type**" follows certain criteria*.

⚠ SCADA Pro calculates automatically the q factor and the type of the structure. To apply the automatic process, you must follow the procedure described bellow:

❖ After having completed all the previously mentioned values, leave the following boxes blank

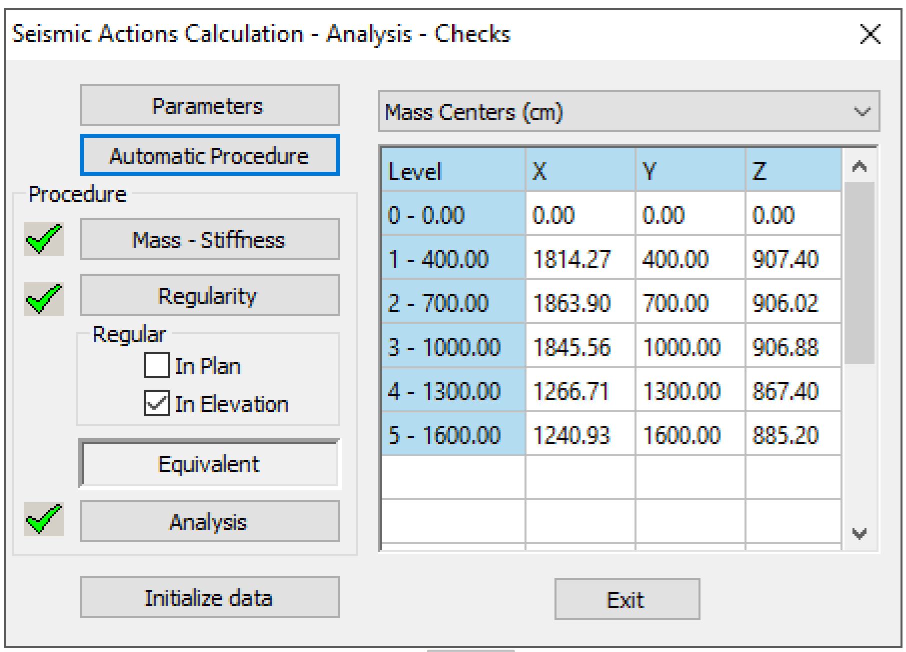


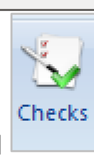
as well as the following options



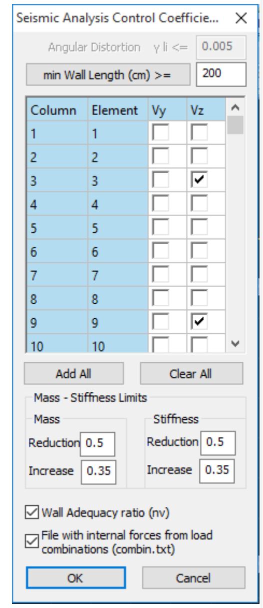
without any changes.

❖ Choose "**Ok**" and use the "**Automatic procedure**" to run an initial analysis.



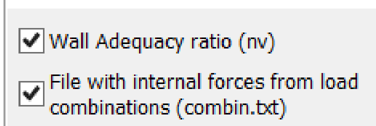
❖ Choose the "**Checks**" command 

and in the dialog box choose choose **"OK".**



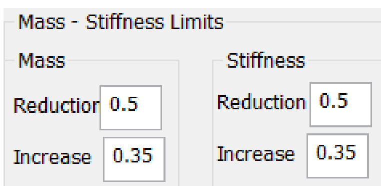
In the dialog box "**Seismic analysis control coefficients**" you are asked to assign a value for the minimum length that a vertical member must have to be regarded as a wall instead of a column. Click the

button, and automatically, all the walls are checked in each direction.



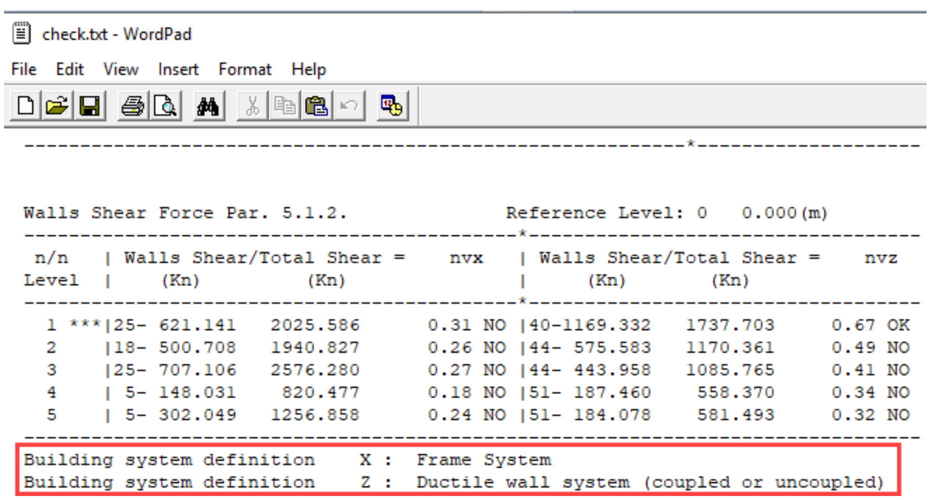
Additionally, by checking the boxes next to the two last options, two .txt files will be created and saved to the folder of the project, ready to be viewed or printed afterwards.

As far as the "**Wall adequacy**" is concerned, the relevant .txt file contains the computation of the shear acting to each wall, at each level of the structure and for all the load combinations considered.



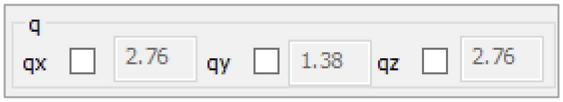
The "**Mass – Stiffness limits**" area, since no specific limitation is prescribed by EC8 (in contrast with EAK – Greek antiseismic regulation), modifications may be incorporated to those limits. Consequently, the building's regularity state in elevation will be altered, too.

In the "Checks" file, the program "defines" the structural type by the base shear undertaken by the walls.

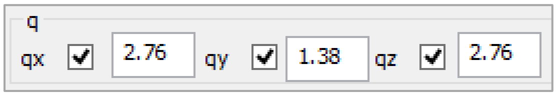



```
 ----------------------------------------------------------------*--------------------


 Walls Shear Force Par. 5.1.2.               Reference Level: 0    0.000(m)
 ---------------------------------------------*-------------------------------------
  n/n    | Walls Shear/Total Shear =     nvx    | Walls Shear/Total Shear =     nvz
 Level   |     (Kn)           (Kn)              |     (Kn)           (Kn)
 ---------------------------------------------*-------------------------------------
   1 ***|25- 621.141    2025.586       0.31 NO |40-1169.332    1737.703      0.67 OK
   2     |18- 500.708    1940.827       0.26 NO |44- 575.583    1170.361      0.49 NO
   3     |25- 707.106    2576.280       0.27 NO |44- 443.958    1085.765      0.41 NO
   4     | 5- 148.031     820.477       0.18 NO |51- 187.460     558.370      0.34 NO
   5     | 5- 302.049    1256.858       0.24 NO |51- 184.078     581.493      0.32 NO
 -----------------------------------------------------------------------------------
 Building system definition     X :  Frame System
 Building system definition     Z :  Ductile wall system (coupled or uncoupled)
```

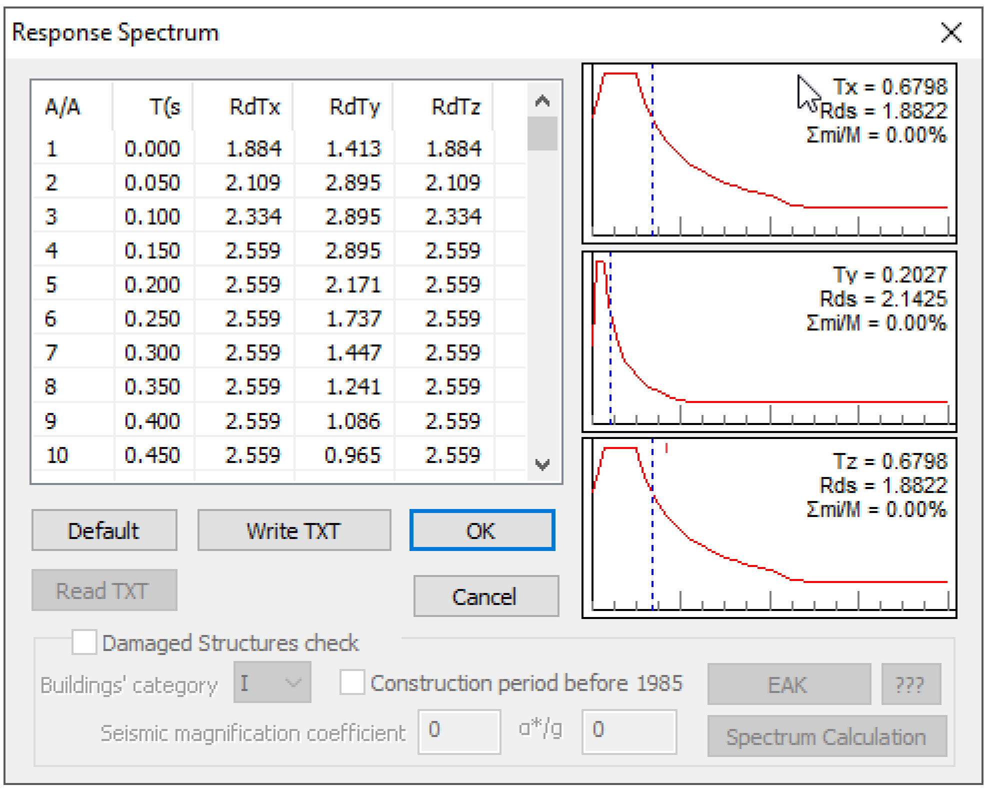
Since the **"Building system definition"** has been determined, it should be included in the **"Parameters"** dialog box. With these changes, analyze for a second time. Now, the proposed values for the **"Behavior coefficient q"** can be found in the "Parameters" dialog box. For the example considered, in the **"q"** area, one can read.



The proposed values may be kept or altered (the latter one is an option that could be utilized from the beginning of the procedure, however, in this occasion the software would not propose any values, at all).



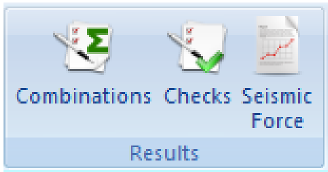
Click [Update Spectrum] to update the spectrum by the new values of the q factor and click [Response Spectrum] to see it.



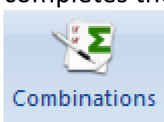
Click "**Ok**" and conduct the analysis one more time, for the **q values to be accounted**.

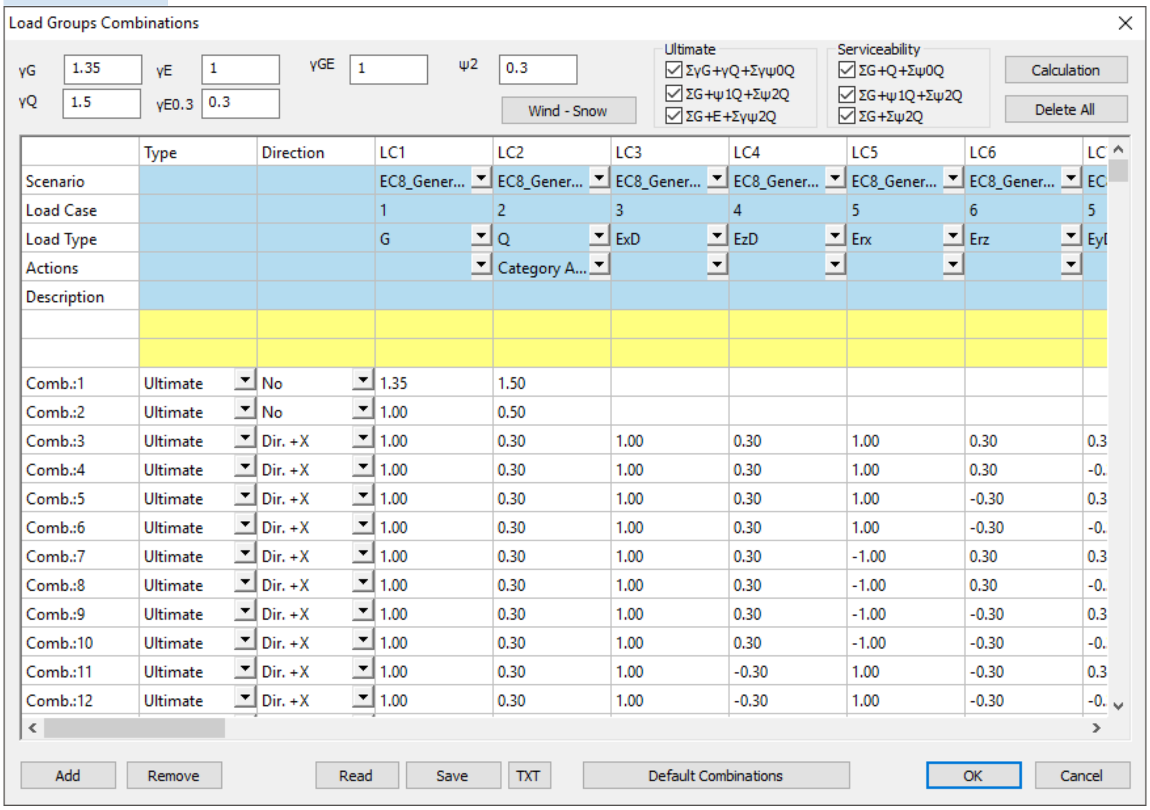
5.3 How to check the analysis results and create the combination file

Right after running the selected scenario analysis, use the “Results” command group to create the combinations (to apply the EC8 checks and design) and view the results of the analysis:



Click “Combinations” to open the “Load Group Combinations” where you can define your combinations or use the results derived from the "Default Combination" button, which completes the table with the combinations of the active scenario analysis.





After running a scenario analysis, combinations are automatically generated by the program. "Combinations" opens the table with the combinations of the active scenarios.

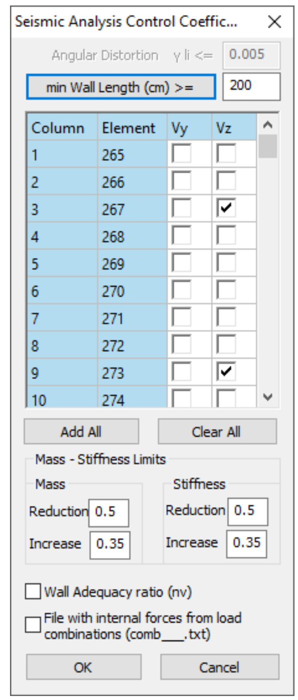
- The same results are derived from the "Default Combination" button, which completes the table with the combinations of the active scenario analysis.

- The default combinations of the executed analysis, are automatically saved by the program.

❖ You can create your combinations without using the "Default", or add more loads of other scenarios and calculate the new combinations either by modifying the defaults, or deleting all "Delete All" and typing other coefficients. Furthermore you can type the factors and select the combinations and then press 'Calculation" to complete the table. The tool "Laod Groups Combinations" works like an Excel file offering possibilities like copy, delete using Ctrl+C, Ctrl+V, Shift and right click.

❖ Predefined combinations concerning seismic scenarios. To create combinations of scenarios without seismic loads you can use both **automatic** and **manual** mode.

5.4 Checks:



Press "Checks" and in the dialog box:

- ✓ Type in the minimum length for defining the walls and click the corresponding button,
- ✓ set limits on the mass and the stiffness considering the regularity conditions of the building,
- ✓ Activate the creation of the two .txt files
- ✓ "OK"

A TXT file that contains design check's results according to the "active scenarios", opens automatically:

- Regularity
- Second Order effects
- Interstory Drift Limitation
- Interstory Drift sensitivity coefficient θ
- Walls Shear Force ratio nv,z
- Seismic joint's calculation
- Torsional sensitivity

check.txt - WordPad

File Edit View Insert Format Help

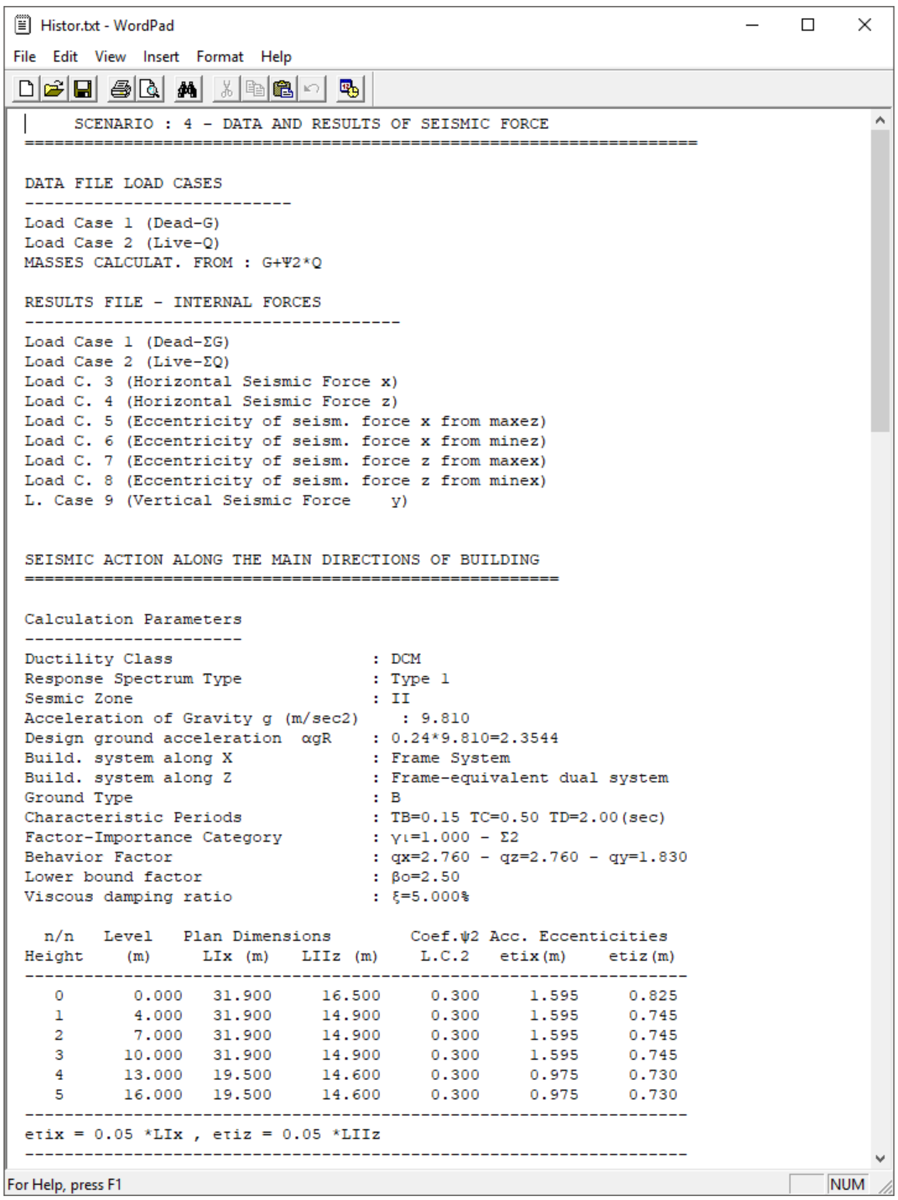
```
                              CHECKS REPORT
              DYNAMIC RESPONSE SPECTRUM ANALYSIS WITH PAIR MOMENTS  (EC8)
 =================================================================

    Check for mass and stiff.differences per build.level (par.4.2.3.3.)
 -------------------------*---------------------*-----------------------------
  n/n     Total    Tot.Mass | Total Stifness       |    Differeces Mass - Stifness
 Level    Heig(M)    KN/g   |     Ki*10^3(KNM)      |   (Mi+1-Mi)/Mi - (Ki+1-Ki)/Ki
 -------------------------*--(Ki-X)--*--(Ki-Z)--*--(ΔMi)--*-(ΔKi-X)-*--(ΔKi-Z)---
   1       4.000     544.053|  9422.231| 54136.261|         |         |
   2       7.000     555.476| 12183.331| 69941.855|inc. 0.02|inc. 0.29|inc. 0.29
   3      10.000     548.121| 12183.331| 69941.855|red  0.01|inc. 0.00|inc. 0.00
   4      13.000     334.994|  9420.903| 69439.631|red  0.38|red  0.22|inc. 0.00
   5      16.000     304.206|  9420.903| 69439.631|red  0.09|inc. 0.00|inc. 0.00
 -------------------------*---------------------*-----------------------------
 Masses    : The increase must be <=0.35 - The reduction must be  <=0.50
 Stifness : Increase must be <=0.35 - Reduction must be <=0.50
  --------------------------------------------------------------------------------
 Check satisfy the regular.in elevation criteria


  Center Weight - Center of Stiff
 ----------------*-------------------------*-------------------------*----------
   n/n    Total    |     CENTER WEIGHT         |     CENTER OF STIFF      | Distance
 Level Height(m)   | X Coor.(m)  Z Coor.(m) | X Coor.(m)  Z Coor.(m)  | C.W-C.S(m)
 ----------------*-------------------------*-------------------------*----------
   1       4.000   |   18.1427     9.0740   |   18.9978     7.5303   |  1.7648
   2       7.000   |   18.6390     9.0602   |   17.8566     7.6059   |  1.6514
   3      10.000   |   18.4556     9.0688   |   17.1299     7.6826   |  1.9181
   4      13.000   |   12.6671     8.6740   |   16.4116     7.8626   |  3.8314
   5      16.000   |   12.4093     8.8520   |   16.6140     7.9362   |  4.3032
 ----------------------------------------------------------*--------------------


 Walls Shear Force Par. 5.1.2.              Reference Level: 0   0.000(m)
 -------------------------------------------*-----------------------------------
  n/n    | Walls Shear/Total Shear =    nvx   | Walls Shear/Total Shear =    nvz
 Level   |     (Kn)           (Kn)             |      (Kn)          (Kn)
 -------------------------------------------*-----------------------------------
   1 ***|25- 621.141    2025.586      0.31 NO |40-1169.332   1737.703     0.67 OK
   2    |18- 500.708    1940.827      0.26 NO |44- 575.583   1170.361     0.49 NO
   3    |25- 707.106    2576.280      0.27 NO |44- 443.958   1085.765     0.41 NO
   4    | 5- 148.031     820.477      0.18 NO |51- 187.460    558.370     0.34 NO
   5    | 5- 302.049    1256.858      0.24 NO |51- 184.078    581.493     0.32 NO
 -------------------------------------------------------------------------------
 Building system definition    X :  Frame System
 Building system definition    Z :  Ductile wall system (coupled or uncoupled)
 *** = Level check nv from regulation

 Regularity in plan check -        Par. 4.2.3.2                  Direction X
 ---------------------------------------------------------------------*---------
   n/n    Total     Coef.λ<4      Coef. r     >   Coef. ls    Eccentricity|Check
 Level    Heig(M)   Lmax/Lmin   sqrt(ΣKt/Σκ_)  sqrt(IO/mass)    ε_o(m)     |Regular
```

For Help, press F1 NUM

5.5 Seismic force:

Select the command and a TXT file which contains the parameters considered in the calculation of the seismic actions as well as the calculated results, automatically opens. Επιλέξτε την εντολή "Σεισμική Δράση" και αυτόματα ανοίγει ένα .txt αρχείο που περιλαμβάνει τις Παραμέτρους Υπολογισμού για τη σεισμική δράση, και τα αποτελέσματα του υπολογισμού για τα παρακάτω μεγέθη:

- ✓ Fundamental Periods
- ✓ Accidental Eccentricities
- ✓ Distribution of the equivalent static force along height (Shear-Moment) Response Spectrum values
- ✓ Response Spectrum values



Histor.txt - WordPad

File Edit View Insert Format Help

```
     SCENARIO : 4 - DATA AND RESULTS OF SEISMIC FORCE
====================================================================

DATA FILE LOAD CASES
---------------------------
Load Case 1 (Dead-G)
Load Case 2 (Live-Q)
MASSES CALCULAT. FROM : G+Ψ2*Q

RESULTS FILE - INTERNAL FORCES
--------------------------------------
Load Case 1 (Dead-ΣG)
Load Case 2 (Live-ΣQ)
Load C. 3 (Horizontal Seismic Force x)
Load C. 4 (Horizontal Seismic Force z)
Load C. 5 (Eccentricity of seism. force x from maxez)
Load C. 6 (Eccentricity of seism. force x from minez)
Load C. 7 (Eccentricity of seism. force z from maxex)
Load C. 8 (Eccentricity of seism. force z from minex)
L. Case 9 (Vertical Seismic Force    y)


SEISMIC ACTION ALONG THE MAIN DIRECTIONS OF BUILDING
======================================================

Calculation Parameters
----------------------
Ductility Class                    : DCM
Response Spectrum Type             : Type 1
Sesmic Zone                        : II
Acceleration of Gravity g (m/sec2)    : 9.810
Design ground acceleration  αgR    : 0.24*9.810=2.3544
Build. system along X              : Frame System
Build. system along Z              : Frame-equivalent dual system
Ground Type                        : B
Characteristic Periods             : TB=0.15 TC=0.50 TD=2.00(sec)
Factor-Importance Category         : γι=1.000 - Σ2
Behavior Factor                    : qx=2.760 - qz=2.760 - qy=1.830
Lower bound factor                 : βo=2.50
Viscous damping ratio              : ξ=5.000%

  n/n   Level   Plan Dimensions        Coef.ψ2 Acc. Eccenticities
Height   (m)      LIx (m)   LIIz (m)    L.C.2   etix(m)    etiz(m)
-------------------------------------------------------------------
   0       0.000   31.900     16.500     0.300     1.595     0.825
   1       4.000   31.900     14.900     0.300     1.595     0.745
   2       7.000   31.900     14.900     0.300     1.595     0.745
   3      10.000   31.900     14.900     0.300     1.595     0.745
   4      13.000   19.500     14.600     0.300     0.975     0.730
   5      16.000   19.500     14.600     0.300     0.975     0.730
-------------------------------------------------------------------
eτix = 0.05 *LIx , eτiz = 0.05 *LIIz
-------------------------------------------------------------------
```

For Help, press F1 NUM

When the scenario regards to a Dynamic Analysis, the following units are included to the exported data as well:

- ✓ Fundamental Periods derived from dynamic analysis
- ✓ Eigenvalues Participation Factors
- ✓ Masses participation factors / Direction
- ✓ Active Modal Masses

Fun.Periods (Modal Resp.Spect. analysis)

n/n Eigenvalue	Cyclic Frequency w (Rad/sec)	Frequency v (Cycles/sec)	Period T (sec)
1	7.2512E+000	1.1541E+000	8.6651E-001
2	7.9224E+000	1.2609E+000	7.9309E-001
3	1.4538E+001	2.3138E+000	4.3218E-001
4	2.6808E+001	4.2667E+000	2.3438E-001
5	2.9671E+001	4.7222E+000	2.1176E-001
6	3.1355E+001	4.9903E+000	2.0039E-001
7	3.4995E+001	5.5696E+000	1.7955E-001
8	3.7508E+001	5.9695E+000	1.6752E-001
9	3.8419E+001	6.1146E+000	1.6354E-001
10	4.2227E+001	6.7206E+000	1.4880E-001

Eigenvalues Participation Factors

n/n	Dir in reference to the global coord.system		
Eigenvalue	Dir X	Dir Y	Dir Z
1	2.1433E+001	-6.9810E-001	2.7847E+001
2	-3.7677E+001	-2.0058E-001	2.1469E+001
3	8.4483E+000	-1.6991E+000	2.5324E+001
4	8.8804E+000	2.9948E+000	2.1994E+000
5	-1.4702E-001	2.9863E+001	4.0690E-001
6	1.8737E-001	-1.5404E+001	-3.3619E+000
7	1.6543E+000	2.1082E+000	-2.0242E+000
8	9.7196E+000	-6.0046E+000	-1.8085E+000
9	-8.7209E+000	-1.2616E+000	2.2461E+000
10	3.3796E+000	1.5838E+001	-2.0670E+000

Masses participation factors / Direction

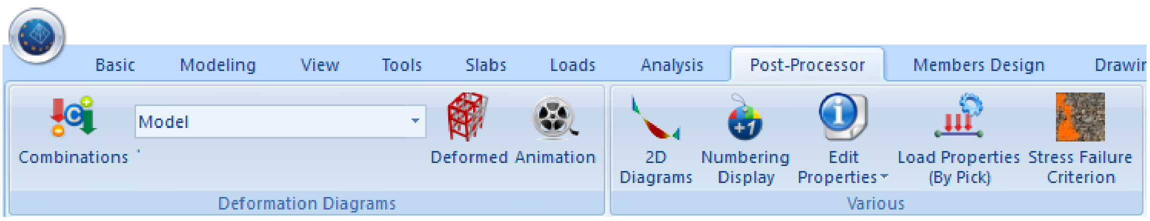
Dir X = 1	Dir Y = 1	Dir Z = 1

⚠ *For more details you can see the User Manual § 7. ANALYSIS*

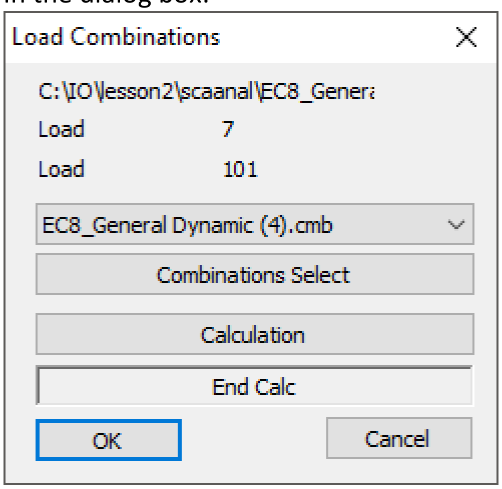
6. RESULTS

6.1 How to view the diagrams and the deformation results and the mesh areas steel reinforcement demand:

Move to "Results" Unit, to get a detailed observation of the internal forces, the diagrams (M, V, N) and the deformed shape of the model as a result of an individual load or load combination.



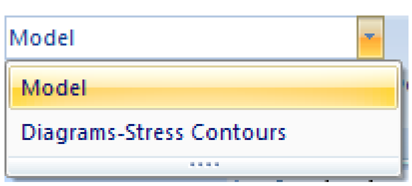
Select "Combinations" and load a combination's file, depending on the results you want to see. In the dialog box:



- Choose a combination from the list that includes the combinations of all the analyses that have been performed, and wait to complete the calculation automatically, or

- press "Combinations Select", select the combinations file from the correspondent folder and press "Calculation".

⚠ To see the deformed shape of the corresponding eigenvalues, choose a dynamic scenario .cmb file.

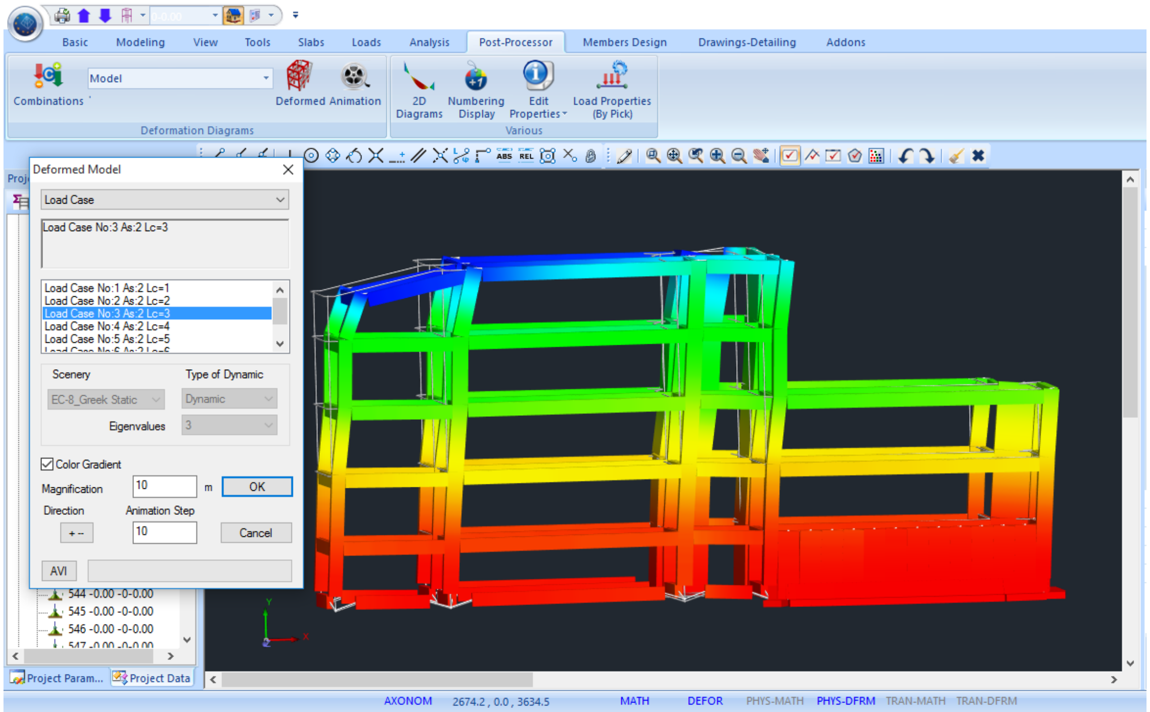


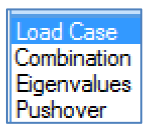
From the list on the right, based on the desired results select:

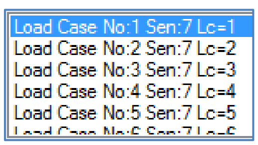
✓ Model or

✓ Diagrams – Stress Contours

6.1.1 Model + “Deformed Shape” :



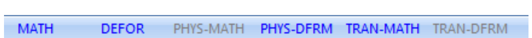
Choose from the list 

the general deformation cause and the next list, a general cause subcase. 

Activate 

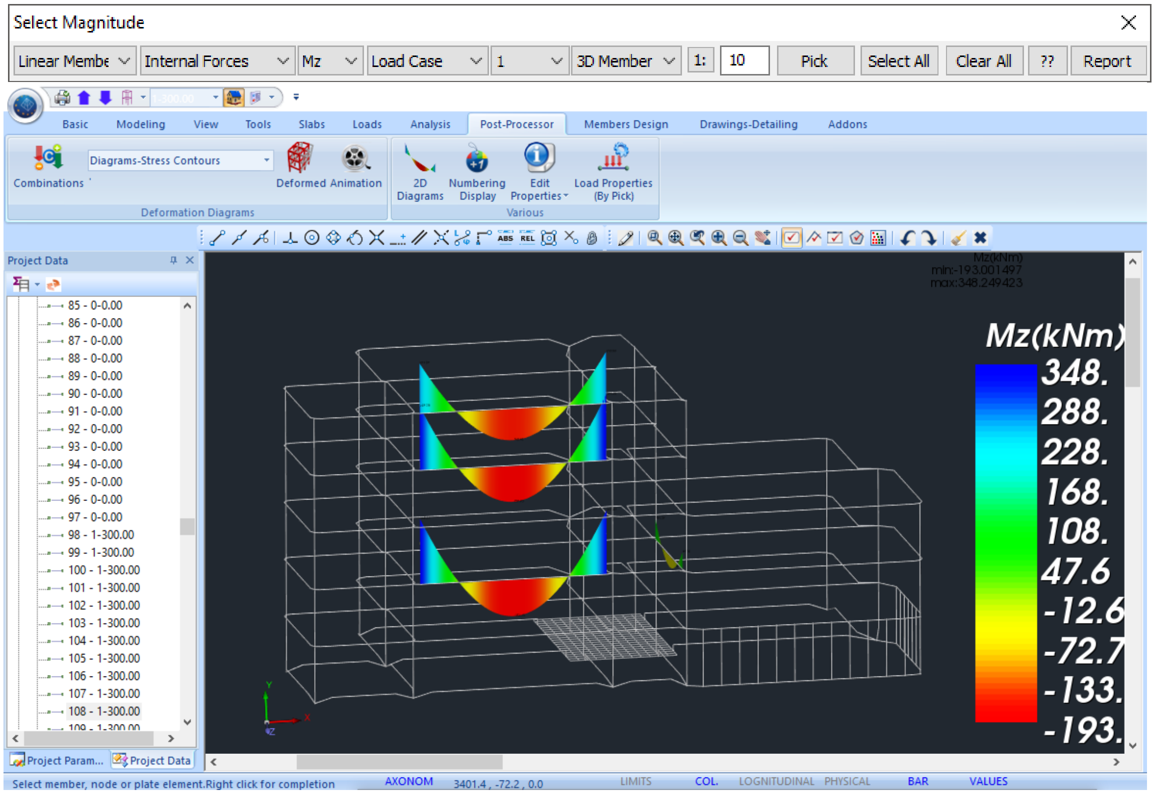
, modify “Magnification” and type in the value of the “Animation Step” to receive a better visualization.

On the “Status Bar” check (double click, blue=active, grey=inactive) the type of the visualization of the deformed model



“Animation” command is a button that activates and deactivates the deformed structure animation, according to the selections made in the “Deformed Model” dialog box.

6.1.2 Diagrams – Stress Contours:



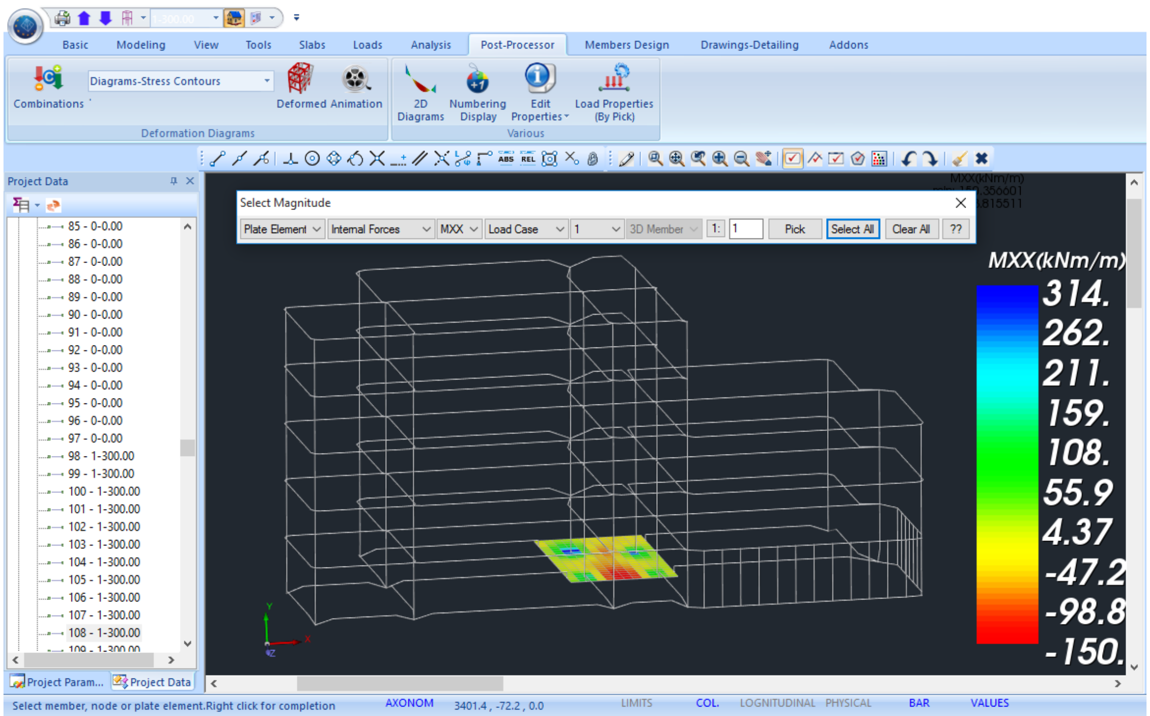
With this command group you can see the stress contours for beam and plate elements, and the calculated Steel Reinforcement for Plate Elements.

For Linear Members you can see:

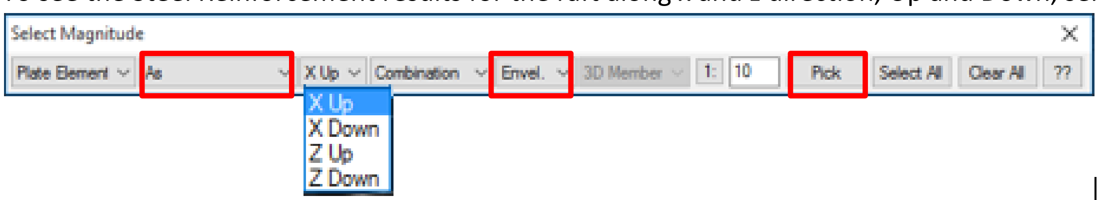
tensions [Mz / Vy / My / Vz / σso. / N / Mx], for each [Load Case / Combination], on a [3D Member / 2D Member / Aligned Member / Grillage / Plane Framework]

For Plate Elements you can choose if you want to see the stress contours for internal forces, deformations, reinforcement As, as well as the load case or the combination as shown below:

Internal Forces [NXX / NYY / NXY / N1 / N2 / Nvm / MXX / MYY / MXY / M1 / M2 / Mvm] Deformations [DX / DY / DZ / RX / RY / RZ / σs] As [X Up / X Down / Z Up / Z Down] for each [Load Case / Combination]

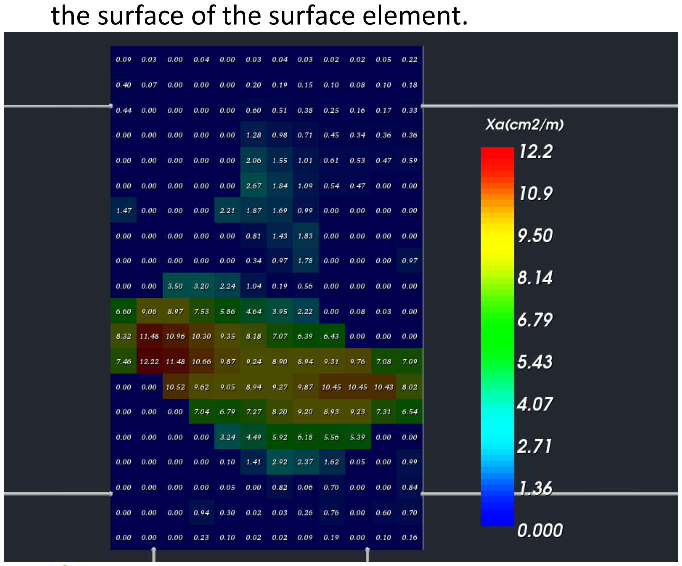


To see the Steel Reinforcement results for the raft along x and z direction, Up and Down, select:



The color bar comes with a color gradation ranging from red to blue (red, green, blue- RGB), declaring in this way the calculated steel reinforcement along each direction for each side.

⚠ Activating VALUES in the lower horizontal bar, you can see the values of the selected size in the surface of the surface element.



⚠ *For more details you can see the User Manual § 8. RESULTS*

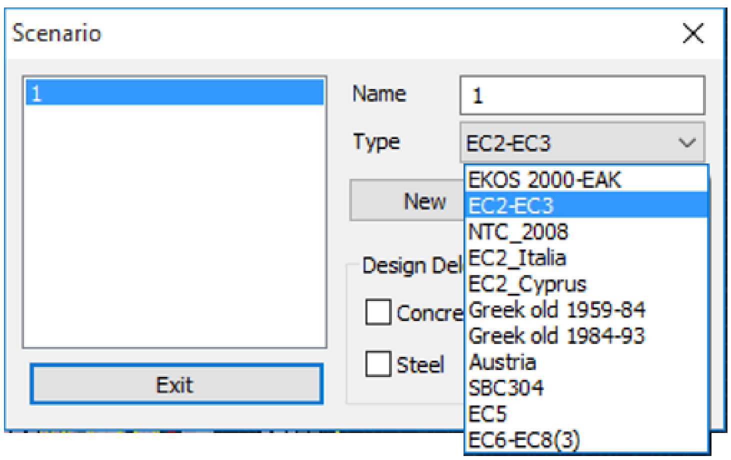
7. DESIGN

Since model analysis has been completed, the design checks of the structural elements are applied according to the design code provisions.

7.1 How to create design scenarios :

Move to "Design" unit and click "New" to create the desired scenario by selecting the considered regulation.

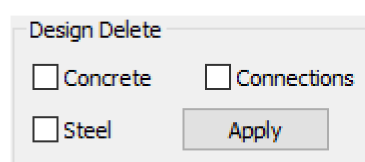
* Predefined scenarios are created according to the Regulation and Attachment option you make at the beginning, within the General Configuration window that opens automatically immediately after the file name is defined.



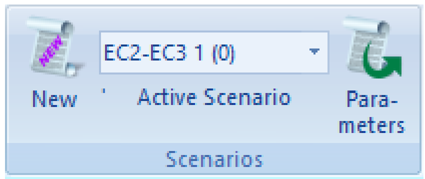
Type a name, select the type and click New, to add the new scenario to the list.

In this example we used a scenario by the Eurocode.

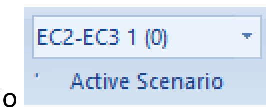
In the field "Design Delete" activate the corresponding checkbox and then press "Apply", to delete the results of previous design checks (concrete elements, steel elements or connections). Repeat using other combinations or parameters or scenarios, etc.



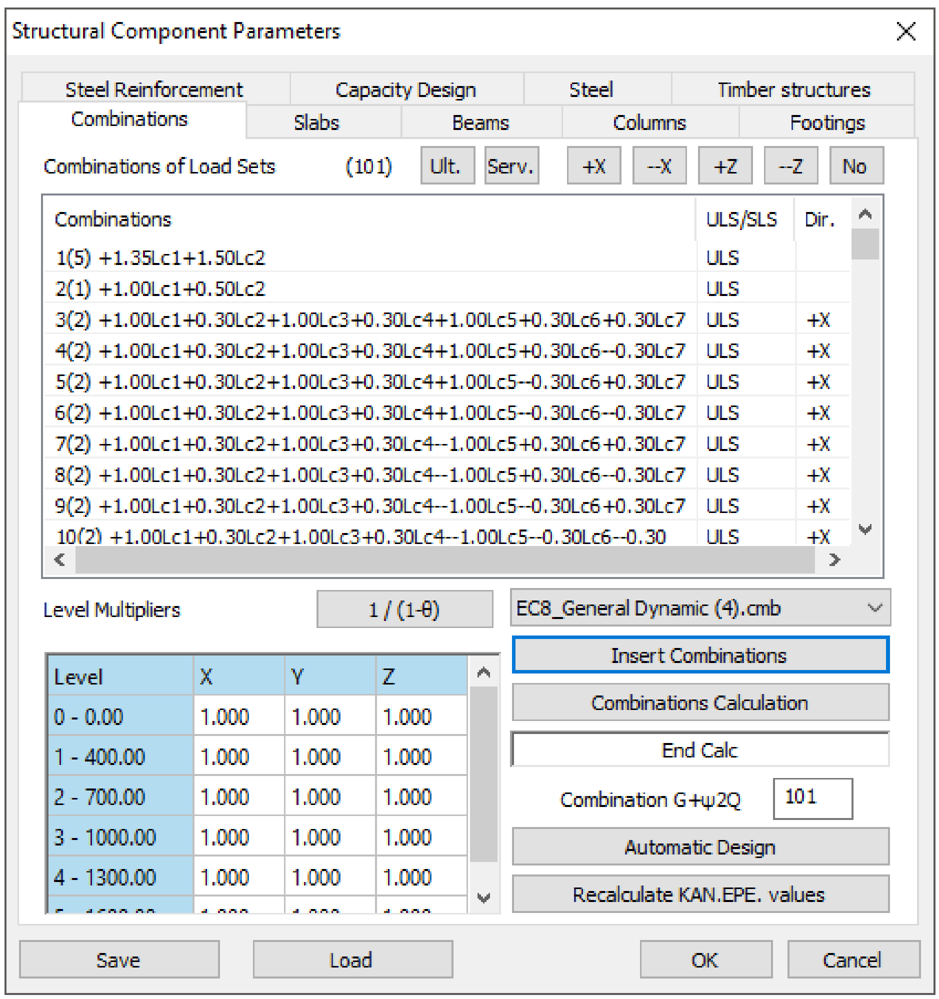
7.2 How to define the parameters of the design for each member type:



The "Scenarios" command group contains the commands for the creation of a new scenario as well as the editing of the parameters of the design checks and reinforcement in every type of structural elements. (**See User Manual Chapter.9 "Design"**)

Select the considered scenario 

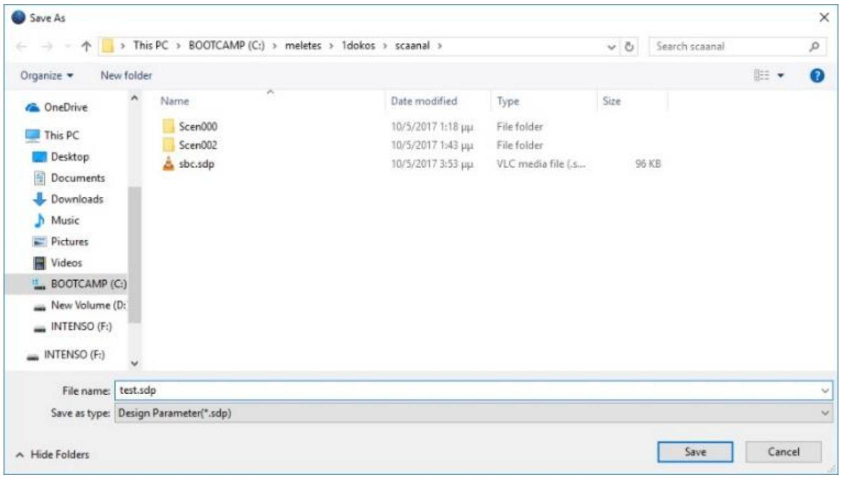
and open the parameters.



Two new commands are related to the storage of the design parameters of the active scenario.

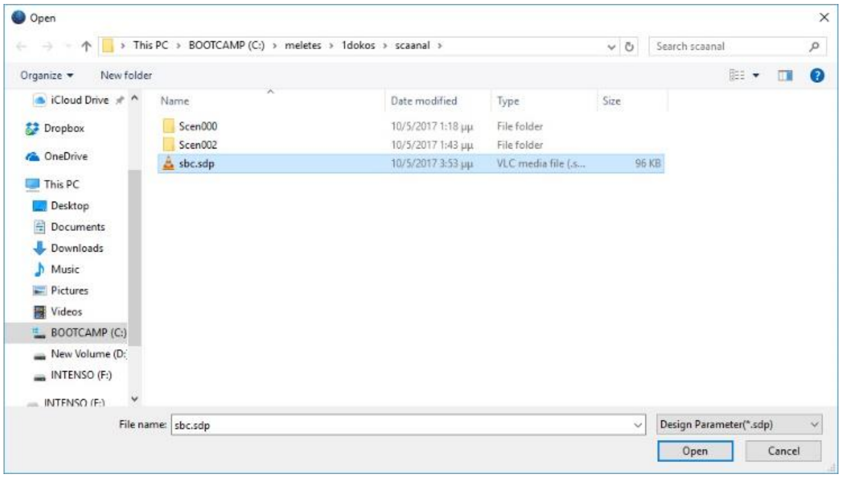
After you define the design parameters you have now the opportunity to save them in a file and use them in your next project.

By clicking the button «**Save**» the storage window opens.



and you type a name (it is good that the name is relevant to the design scenario).

The extension of the files is sdp scenery design parameters.
Respectively, by clicking "Reading" you can load into one of your projects a parameter file you have already saved.



⚠ **ATTENTION**

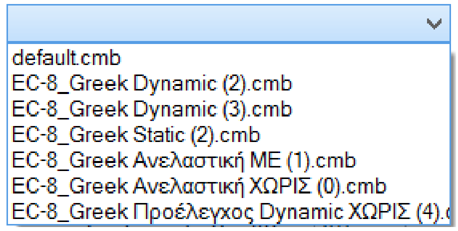
A necessary condition to load a parameter file is that the active design scenario should be the same as the parameters scenario you are loading. Othewise, you will see an error message.

⚠ *A new command that allows the recalculation of all values provided by KANEPE (Greek norm) for all members of the study and is used in cases where the strength of the materials is changed while the reinforcement has been placed by the existing situation*

Combinations

⚠ *Regardless of the material, the calculation of combinations is a condition for designing.

The selection of the existing .cmb combinations file is made:



- from the dropdown list with automatic calculation or

- Through the command Insert Combinations that opens the folder with the registered .cmb files. Select the file and press Combinations Calculation .

Depending on the situation and the conditions being satisfied, you can use either the static or dynamic combinations so as the superstructure is designed (as long as you have the springs free, not fixed). You may also have performed analyzes taking into consideration different regulations (eg EAK and EC8) and by designing according to the corresponding combinations you will be able to see the differences that arise

In "Combinations" tab the combinations list is displayed.
In "Level Multipliers" tab :

Level Multipliers 1 / (1-θ)

Level	X	Y	Z
0 - 0.00	1.000	1.000	1.000
1 - 300.00	1.000	1.000	1.000
2 - 600.00	1.000	1.000	1.000
3 - 900.00	1.000	1.000	1.000
4 - 1200.00	1.000	1.000	1.000

You can increase or decrease the seismic actions in any direction and level, by typing different factors You can modify the default coefficients of the seismic loads per direction and level, by typing values different than the unit.

⚠ Press the button 1 / (1-θ) to take into account the P-Delta effect during the design check. The stress resultants will be increased automatically at the corresponding levels, where $0.1 < \theta < 0.2$.

⚠ **ATTENTION:**

For modification purposes, press the following button Combinations Calculation.

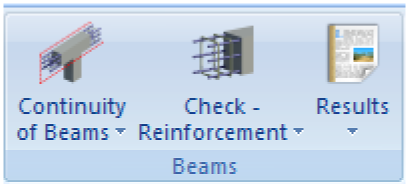
The field Combination G+ψ2Q 99 refers to the scenarios of the Greek Regulation.

⚠ **NOTE:**

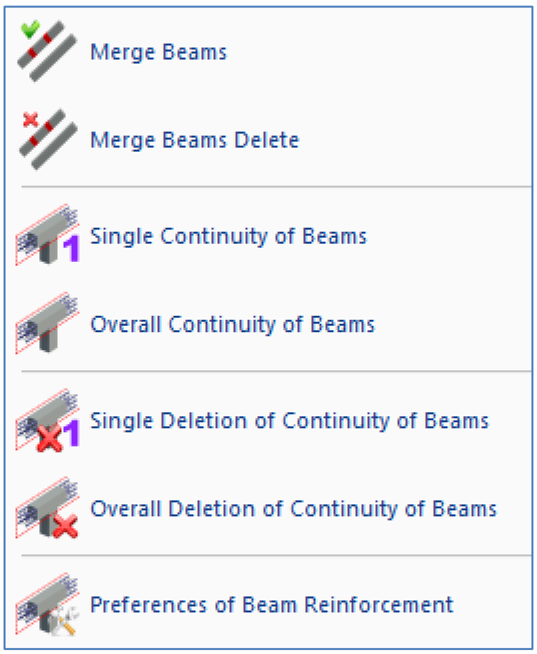
The Automatic Design command offers the possibility for an automatic application of the appropriate design checks and the automatic designing of all structural elements for concrete structures, just by pressing the corresponding button.
Set the parameters in the following tabs :

and then press the button "Automatic Design" or follow step by step the procedure to design the structural elements concerning the fulfillment of the design checks.

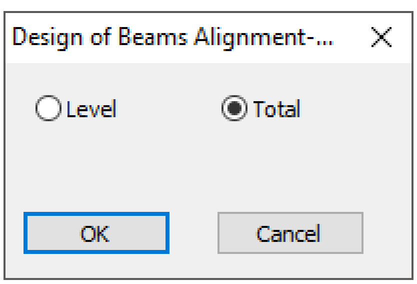
7.3 How to perform Beam Design:



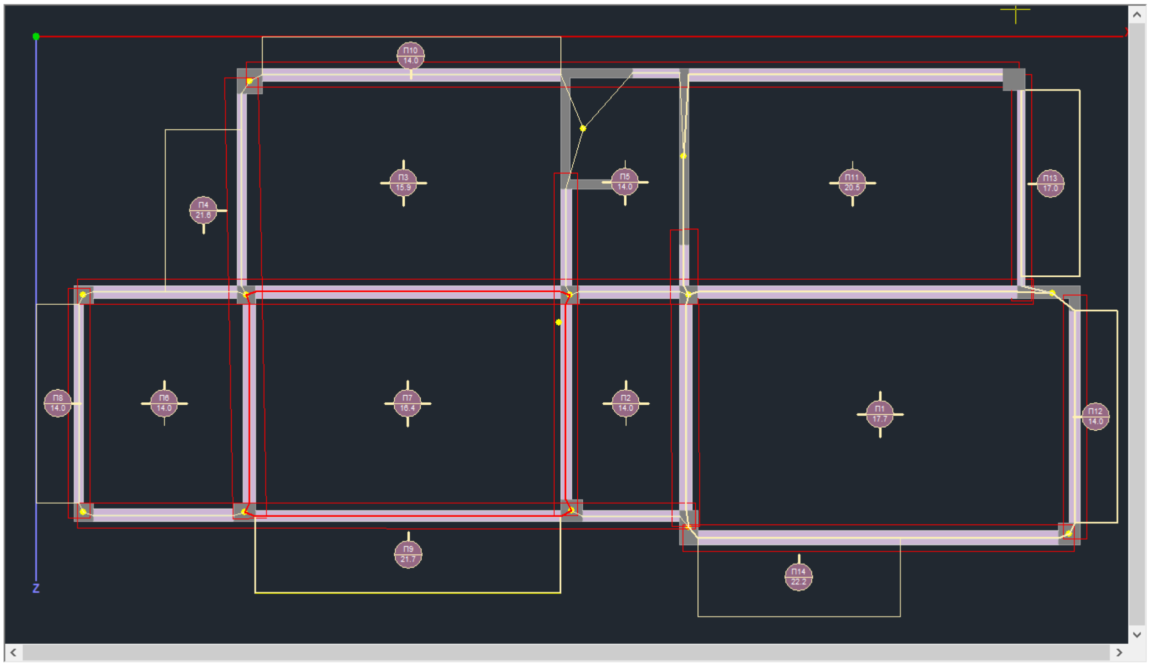
The "Beams" command group, contains the "Continuity of Beams", "Check – Reinforcement" and the "Results" commands.



Select the "Continuity of Beams – Overall Continuity of Beams"

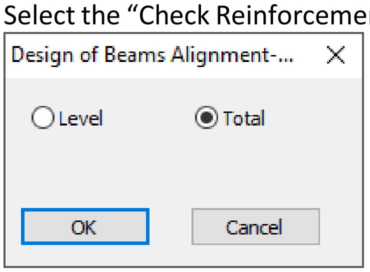


To determine the beams' alignment of the current level or the entire building automatically.
The program creates automatically all the continuities of beams.



Use the “Preferences of Beams Reinforcement” command to, insert one common bar or two different bars on the support of the continuous beams, to take into account both of them, to change the anchorage length and if you wish to modify the support widths.

Select the “Check Reinforcement > Overall” to perform the design of every beam of the structure



The program makes the design checks and displays the results using colors and symbols indicating in this way the type of the failure.

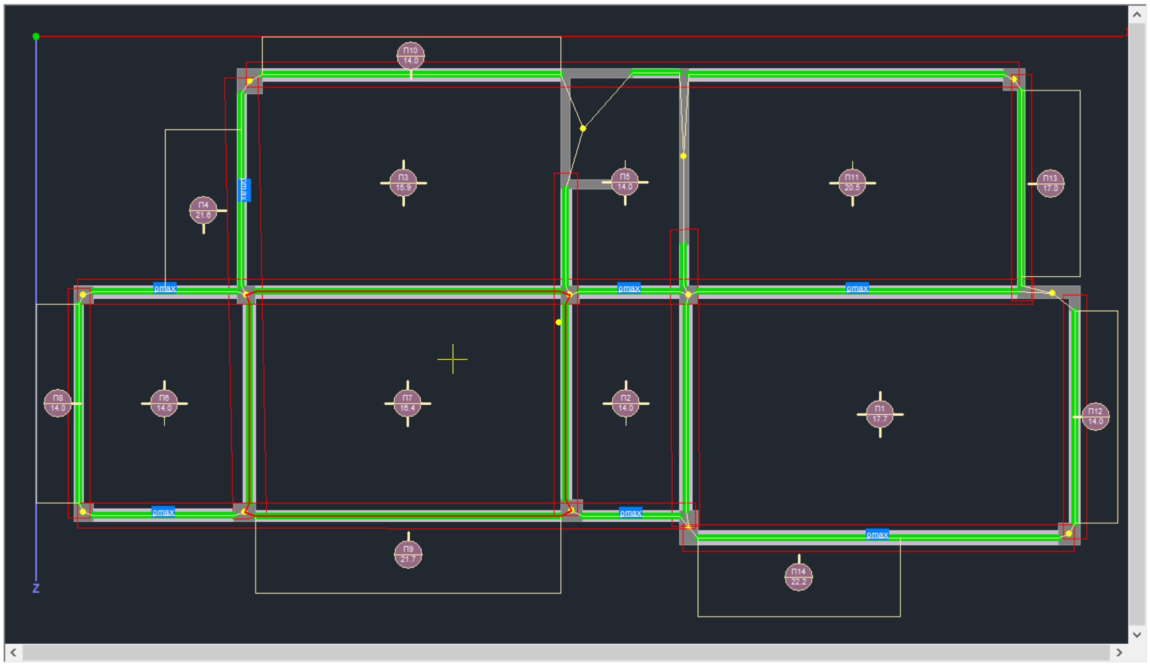
The colored indicators of the beam’s failure:

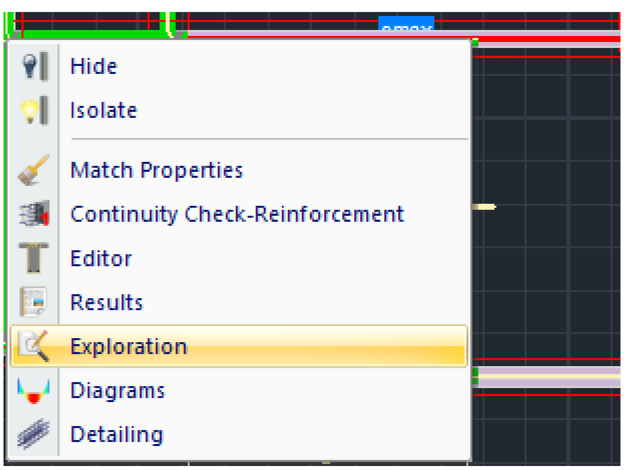
- **Red**. Failure in Bending. It has exceeded the maximum steel reinforcement ratio ρ_{max}.
- **Pink**. Failure in Shear / Torsion.
- **Cyan**. Passed the design checks.

The symbol on the beam indicates:

Failure in Bending	M
Failure in Shear	V
Failure in Torsion	T
Dense positioned Stirrups	Asw
It has exceeded the maximum steel reinforcement ratio	ρ
It has exceeded the maximum anchorage length	ldb
Capacity Design	αcd
It has exceeded the maximum crack width	Wk
Deflection failure	L/d

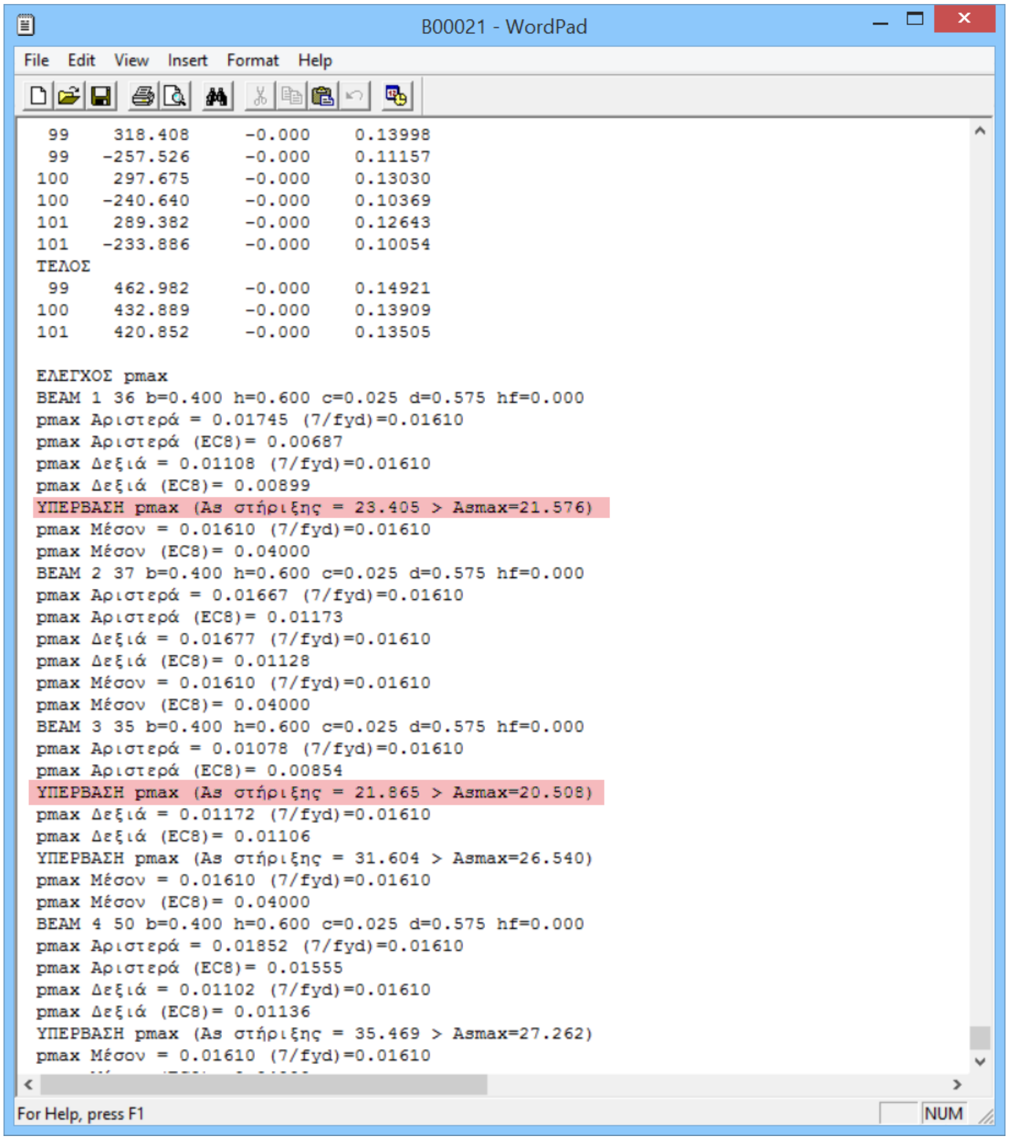
In this example the beam design process indicated some failures related to the exceedance of the maximum steel reinforcement on the supports *“pmax”.*





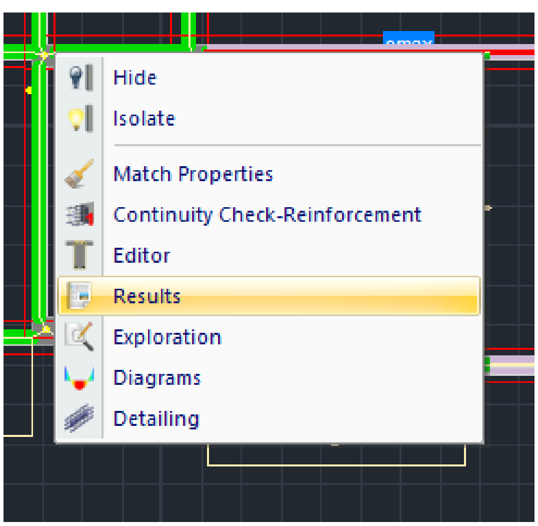
Right click on the beam member that fails the checks to open a list of commands related to the design of the beam continuity.

Click “**Exploration**” to see the results for the selected beam continuity on the file that opens:

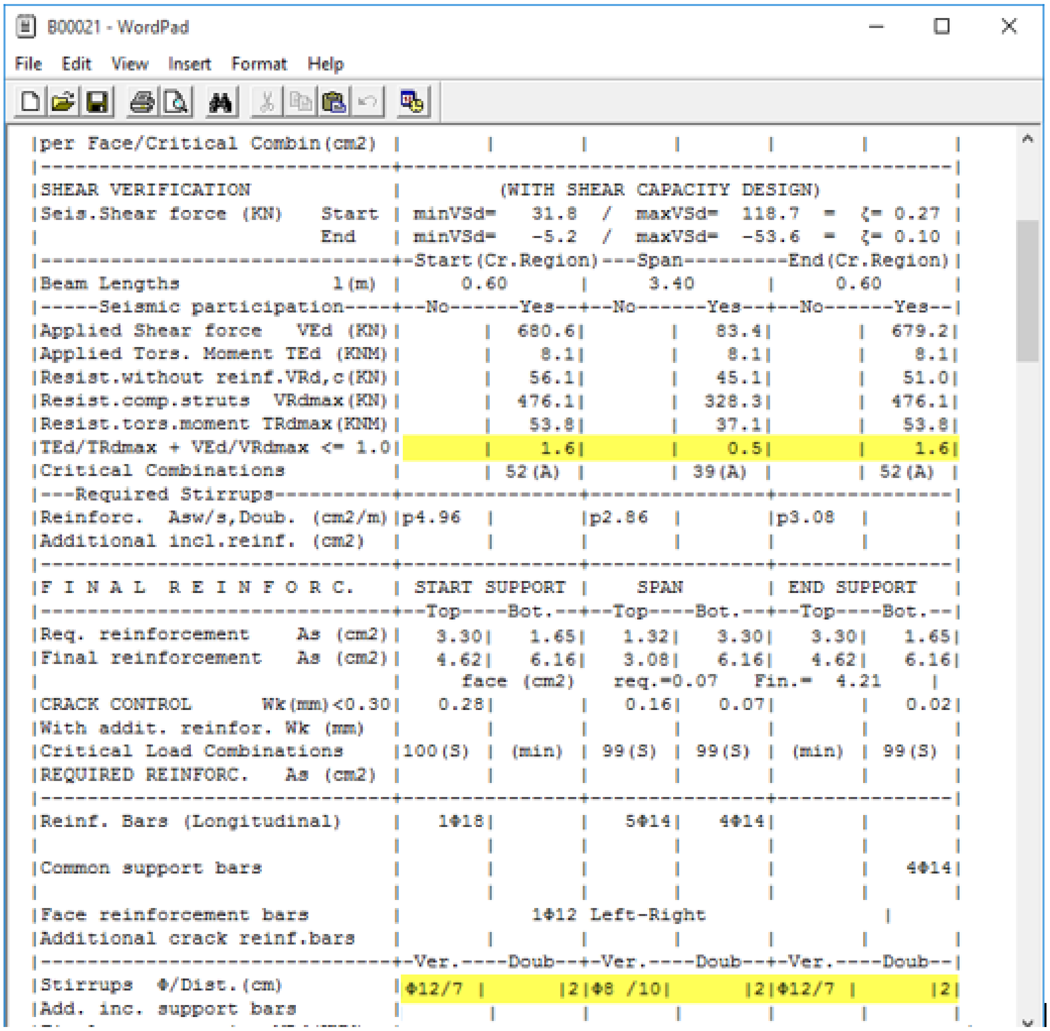



```
 99    318.408     -0.000    0.13998
 99   -257.526     -0.000    0.11157
100    297.675     -0.000    0.13030
100   -240.640     -0.000    0.10369
101    289.382     -0.000    0.12643
101   -233.886     -0.000    0.10054
ΤΕΛΟΣ
 99    462.982     -0.000    0.14921
100    432.889     -0.000    0.13909
101    420.852     -0.000    0.13505

ΕΛΕΓΧΟΣ pmax
BEAM 1 36 b=0.400 h=0.600 c=0.025 d=0.575 hf=0.000
pmax Αριστερά = 0.01745 (7/fyd)=0.01610
pmax Αριστερά (EC8)= 0.00687
pmax Δεξιά = 0.01108 (7/fyd)=0.01610
pmax Δεξιά (EC8)= 0.00899
ΥΠΕΡΒΑΣΗ pmax (As στήριξης = 23.405 > Asmax=21.576)
pmax Μέσον = 0.01610 (7/fyd)=0.01610
pmax Μέσον (EC8)= 0.04000
BEAM 2 37 b=0.400 h=0.600 c=0.025 d=0.575 hf=0.000
pmax Αριστερά = 0.01667 (7/fyd)=0.01610
pmax Αριστερά (EC8)= 0.01173
pmax Δεξιά = 0.01677 (7/fyd)=0.01610
pmax Δεξιά (EC8)= 0.01128
pmax Μέσον = 0.01610 (7/fyd)=0.01610
pmax Μέσον (EC8)= 0.04000
BEAM 3 35 b=0.400 h=0.600 c=0.025 d=0.575 hf=0.000
pmax Αριστερά = 0.01078 (7/fyd)=0.01610
pmax Αριστερά (EC8)= 0.00854
ΥΠΕΡΒΑΣΗ pmax (As στήριξης = 21.865 > Asmax=20.508)
pmax Δεξιά = 0.01172 (7/fyd)=0.01610
pmax Δεξιά (EC8)= 0.01106
ΥΠΕΡΒΑΣΗ pmax (As στήριξης = 31.604 > Asmax=26.540)
pmax Μέσον = 0.01610 (7/fyd)=0.01610
pmax Μέσον (EC8)= 0.04000
BEAM 4 50 b=0.400 h=0.600 c=0.025 d=0.575 hf=0.000
pmax Αριστερά = 0.01852 (7/fyd)=0.01610
pmax Αριστερά (EC8)= 0.01555
pmax Δεξιά = 0.01102 (7/fyd)=0.01610
pmax Δεξιά (EC8)= 0.01136
ΥΠΕΡΒΑΣΗ pmax (As στήριξης = 35.469 > Asmax=27.262)
pmax Μέσον = 0.01610 (7/fyd)=0.01610
```

⚠ ***NOTE**: You can retrieve information related to most of the failure types through the right click menu “**Results**” command*



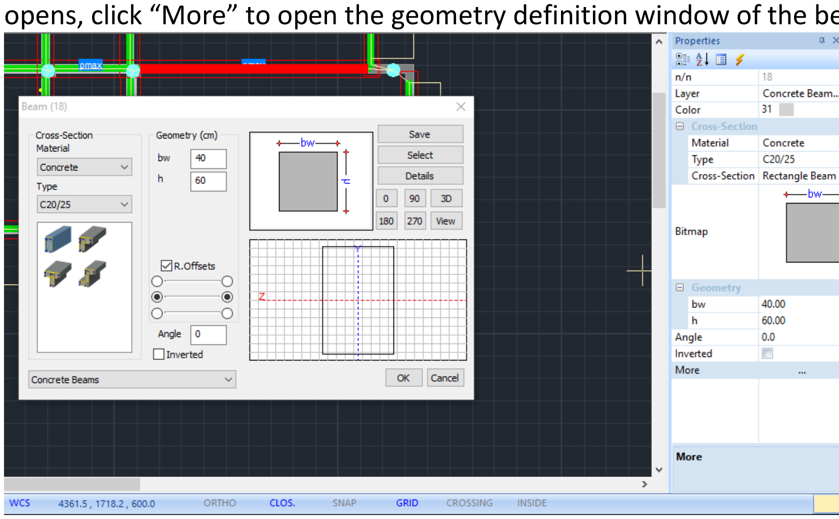
*For example, for a failure described as “**ΔΣ**”: Select “**Results**” to check the failure data at the file summary results that opens:*

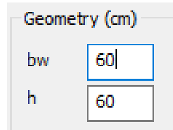


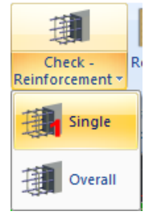
```
B00021 - WordPad
File  Edit  View  Insert  Format  Help
|per Face/Critical Combin(cm2)  |         |         |         |         |         |
|-------------------------------+-----------------------------------------------|
|SHEAR VERIFICATION             |         (WITH SHEAR CAPACITY DESIGN)          |
|Seis.Shear force  (KN)   Start | minVSd=   31.8  /  maxVSd=  118.7  =  ζ= 0.27 |
|                         End   | minVSd=   -5.2  /  maxVSd=  -53.6  =  ζ= 0.10 |
|-------------------------------+-Start(Cr.Region)---Span---------End(Cr.Region)|
|Beam Lengths             l(m)  |     0.60        |     3.40        |     0.60   |
|-----Seismic participation-----+--No------Yes--+--No------Yes--+--No------Yes--|
|Applied Shear force    VEd (KN)|         |  680.6|         |   83.4|         |  679.2|
|Applied Tors. Moment TEd (KNM) |         |    8.1|         |    8.1|         |    8.1|
|Resist.without reinf.VRd,c(KN) |         |   56.1|         |   45.1|         |   51.0|
|Resist.comp.struts  VRdmax(KN) |         |  476.1|         |  328.3|         |  476.1|
|Resist.tors.moment TRdmax(KNM) |         |   53.8|         |   37.1|         |   53.8|
|TEd/TRdmax + VEd/VRdmax <= 1.0 |         |    1.6|         |    0.5|         |    1.6|
|Critical Combinations          |         | 52(A) |         | 39(A) |         | 52(A) |
|---Required Stirrups-----------+-----------------+-----------------+---------------|
|Reinforc.  Asw/s,Doub. (cm2/m) |p4.96    |         |p2.86    |         |p3.08    |         |
|Additional incl.reinf.  (cm2)  |         |         |         |         |         |         |
|-------------------------------+-----------------+-----------------+---------------|
|F I N A L  R E I N F O R C.    | START SUPPORT |     SPAN      | END SUPPORT   |
|-------------------------------+--Top----Bot.--+--Top----Bot.--+--Top----Bot.--|
|Req. reinforcement    As (cm2) |   3.30|   1.65|   1.32|   3.30|   3.30|   1.65|
|Final reinforcement   As (cm2) |   4.62|   6.16|   3.08|   6.16|   4.62|   6.16|
|                               |     face  (cm2)    req.=0.07    Fin.=  4.21   |
|CRACK CONTROL       Wk(mm)<0.30|   0.28|       |   0.16|   0.07|       |   0.02|
|With addit. reinfor. Wk  (mm)  |       |       |       |       |       |       |
|Critical Load Combinations     |100(S) | (min) | 99(S) | 99(S) | (min) | 99(S) |
|REQUIRED REINFORC.    As  (cm2)|       |       |       |       |       |       |
|-------------------------------+---------------+---------------+---------------|
|Reinf. Bars (Longitudinal)     |   1Φ18|       |   5Φ14|   4Φ14|       |       |
|                               |       |       |       |       |       |       |
|Common support bars            |       |       |       |       |       |   4Φ14|
|                               |       |       |       |       |       |       |
|Face reinforcement bars        |          1Φ12 Left-Right                  |
|Additional crack reinf.bars    |       |       |       |       |       |       |
|-------------------------------+-Ver.----Doub--+-Ver.----Doub--+-Ver.----Doub--|
|Stirrups   Φ/Dist.(cm)         |Φ12/7  |      |2|Φ8 /10|      |2|Φ12/7  |      |2|
|Add. inc. support bars         |       |       |       |       |       |       |
```

After locating the elements that fail the checks, you must perform the necessary modifications to pass all the checks for every element

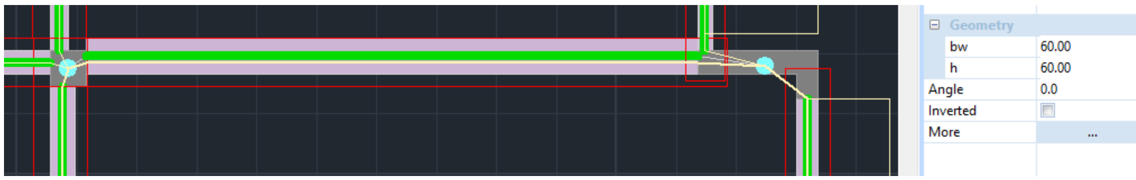
Select the beam with left click on 2D view of the floor plan. On the left, the "Properties" list that opens, click "More" to open the geometry definition window of the beam.

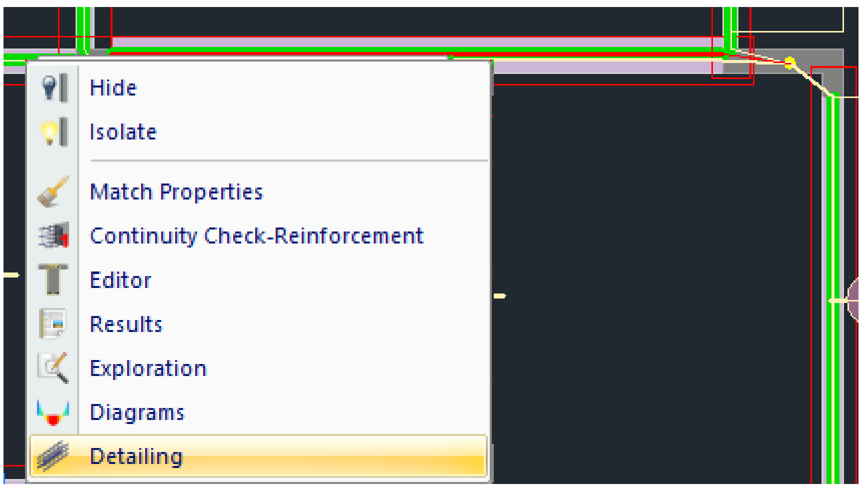


Increase the cross section of the beam 

and use the 

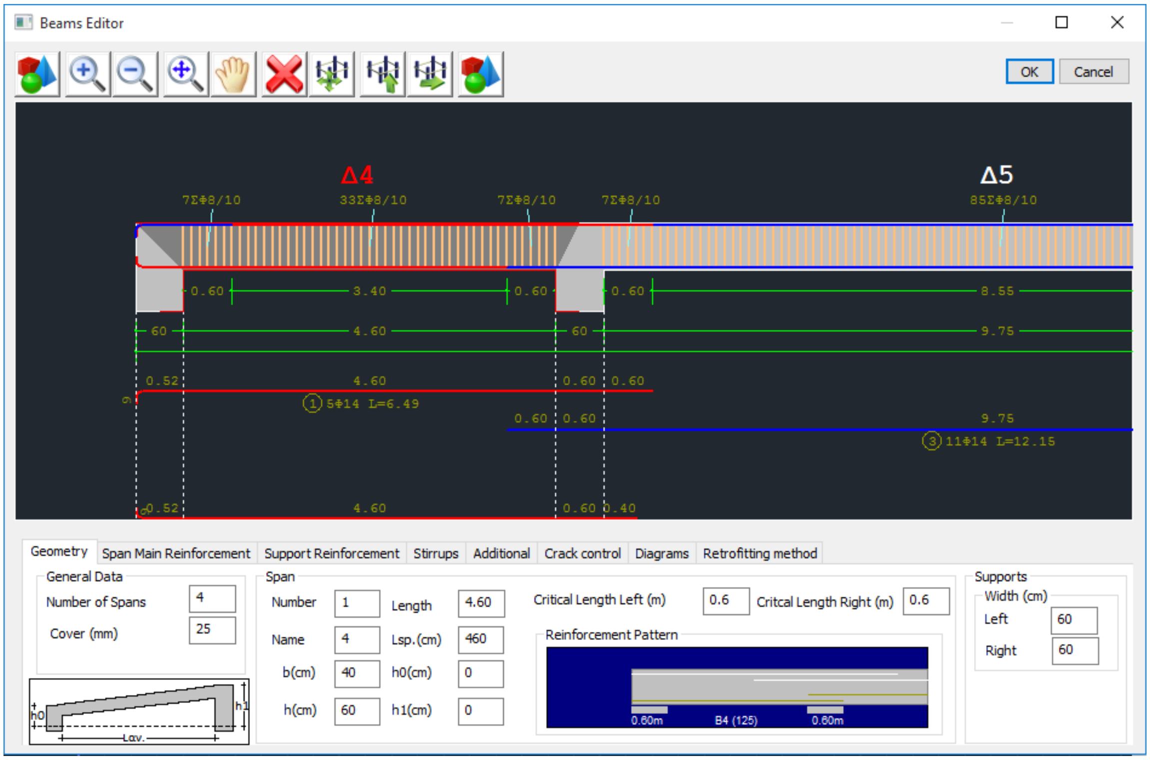
command to design the beam continuity.





Right click on the beam and select "**Reinforcement Detailing**", to open a window with the respective details of the steel reinforcement of the beam continuity, derived from the design process (the direction of the display result meets the direction of the local axis)

⚠ Attention: beams of the same continuity must share the same local axis direction.

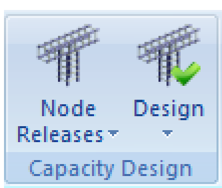


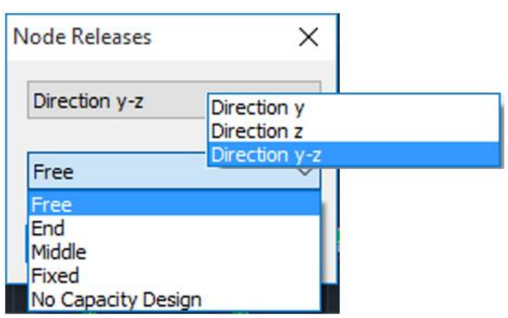
Here, you can modify the main and secondary steel reinforcement.

⚠ *Detailed description on how to perform the desired modifications can be found in the Use Manual. (**Chapter .A "Beam's Detailing"**)*

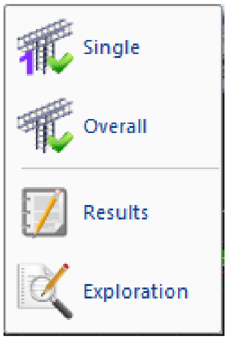
7.4 How to apply Capacity Design

After defining the limits of the acd (capacity design coefficient factor) at the "**Capacity Design**" tab of the **Parameters** section, along each direction (x, z), that will be used during the capacity design, use the command "**Node Releases**" to define the support condition for each node along each direction.





⚠ It is reminded that if you don't define the Node Releases all nodes are considered to be free along both directions (except for the fixed ones)



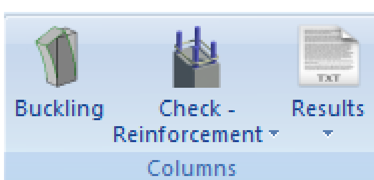
The capacity design can be performed either selectively or overall.

Select the **Results** command to display the TXT file that contains the results of the main design checks of the capacity design. Select the command and left click on the node to open the TXT file and read the results (along each direction).

Node = 33
Col. bottom = 19
Column up = 33

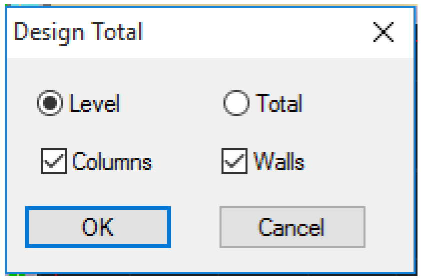
COMB.	SMRby	SMEby	acdy calc	acdy	SMRbz	SMEbz	acdz calc	acdz
3	1016.600	184.132	7.177	3.500	541.400	113.397	6.207	3.500
4	1016.600	174.484	7.574	3.500	541.400	99.594	7.067	3.500
5	1016.600	184.850	7.149	3.500	541.400	113.496	6.201	3.500
6	1016.600	175.202	7.543	3.500	541.400	99.693	7.060	3.500
7	1016.600	181.740	7.272	3.500	541.400	113.067	6.225	3.500
8	1016.600	172.092	7.679	3.500	541.400	99.264	7.090	3.500
9	1016.600	182.458	7.243	3.500	541.400	113.166	6.219	3.500
10	1016.600	172.810	7.648	3.500	541.400	99.363	7.083	3.500
11	1016.600	164.927	8.013	3.500	541.400	13.888	50.678	3.500
12	1016.600	155.280	8.511	3.500	270.700	19.907	17.678	3.500
13	1016.600	164.210	8.048	3.500	414.600	7.586	71.052	3.500
14	1016.600	154.562	8.550	3.500	143.900	20.006	9.351	3.500
15	1016.600	162.535	8.131	3.500	414.600	7.119	75.706	3.500
16	1016.600	152.887	8.644	3.500	143.900	20.237	9.244	3.500
17	1016.600	161.818	8.167	3.500	414.600	13.422	40.158	3.500
18	1016.600	152.170	8.685	3.500	143.900	20.336	9.199	3.500
19	304.000	155.280	2.545	2.545	414.600	19.907	27.075	3.500
20	304.000	164.927	2.396	2.396	143.900	13.888	13.470	3.500
21	304.000	154.562	2.557	2.557	541.400	20.006	35.181	3.500
22	304.000	164.210	2.407	2.407	270.700	7.586	46.391	3.500
23	304.000	152.887	2.585	2.585	541.400	20.237	34.779	3.500
24	304.000	162.535	2.431	2.431	270.700	7.119	49.430	3.500
25	304.000	152.170	2.597	2.597	541.400	20.336	34.610	3.500
26	304.000	161.818	2.442	2.442	270.700	13.422	26.220	3.500
27	304.000	174.484	2.265	2.265	143.900	99.594	1.878	1.878

7.5 How to design columns and walls



The “Columns” field contains the commands related to the Design, Reinforcement Check, columns and walls Results.
(**You can see the User Manual Chapter 9 “Design”)**

Select the command **“Check Reinforcement > Overall”** to perform the design of all the columns and walls of the building (the design will be performed automatically by level for the whole building).



Activate the command and the following dialog box opens:

Select whether to design all columns/walls of the current level or the building in total

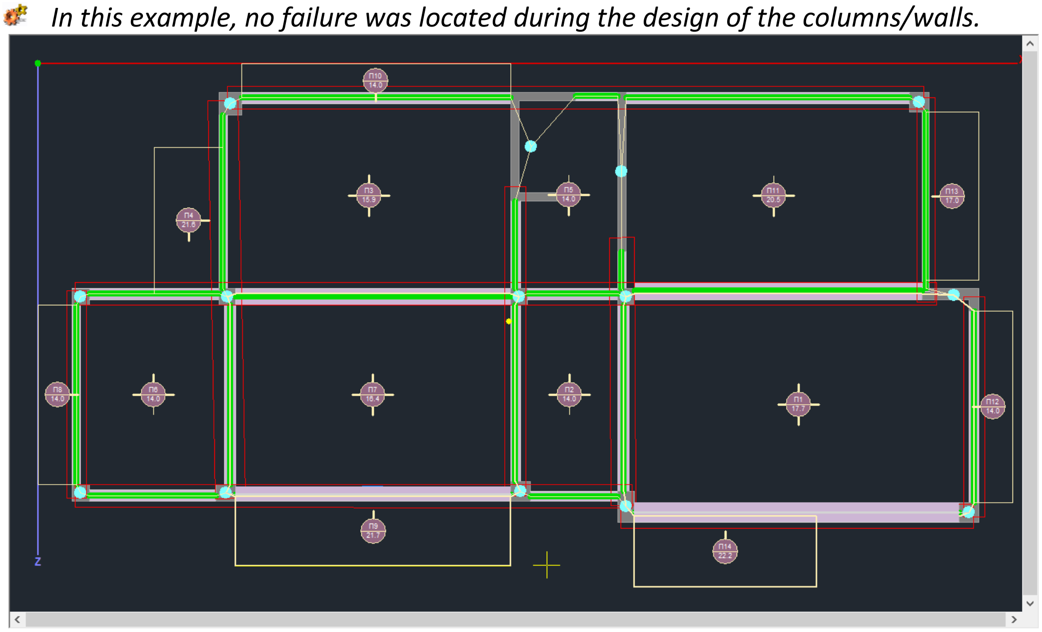
After the design A colored dot is displayed in the center of the element. The color changes according to the type of failure as follows

- **Red:** Failure caused by biaxial bending. The steel reinforcement exceeded the maximum ratio of 4%. Dense stirrups. No results are displayed.
- **Pink:** Failure by Shear / Torsion or exceedance of the ductility level. The results show the reason of failure.
- **Cyan:** All design checks are verified.

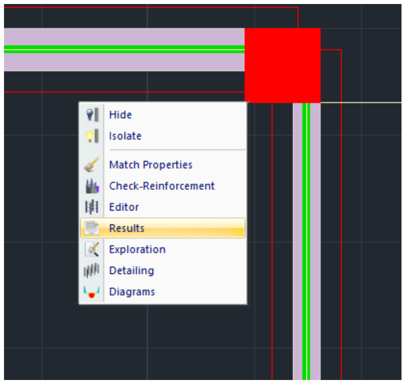
The initially indicated type of failure appears above the element as well:

Failure by biaxial bending	M-N
Failure by Shear	V
Confinement failure	ωwd
Buckling failure	λ
Failure by Torsion	T
Dense Stirrups	Asw
Exceedance of 4% steel reinforcement ratio	ρ
Exceedance of the ductility index	ν

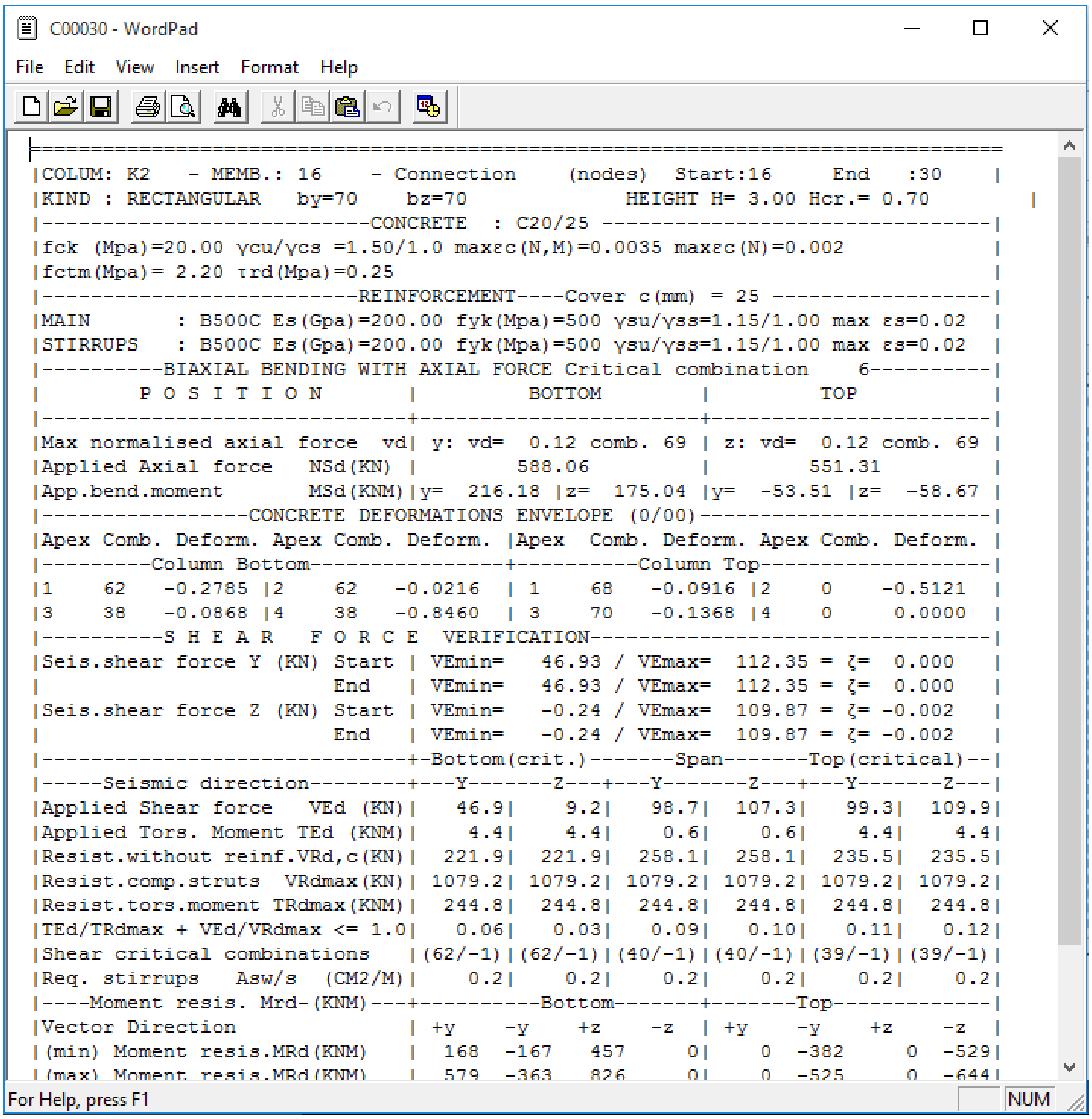
In this example, no failure was located during the design of the columns/walls.



Right click on the section to open a list of commands related to the elaboration and the display of the results derived from the design checks of columns/wall design.

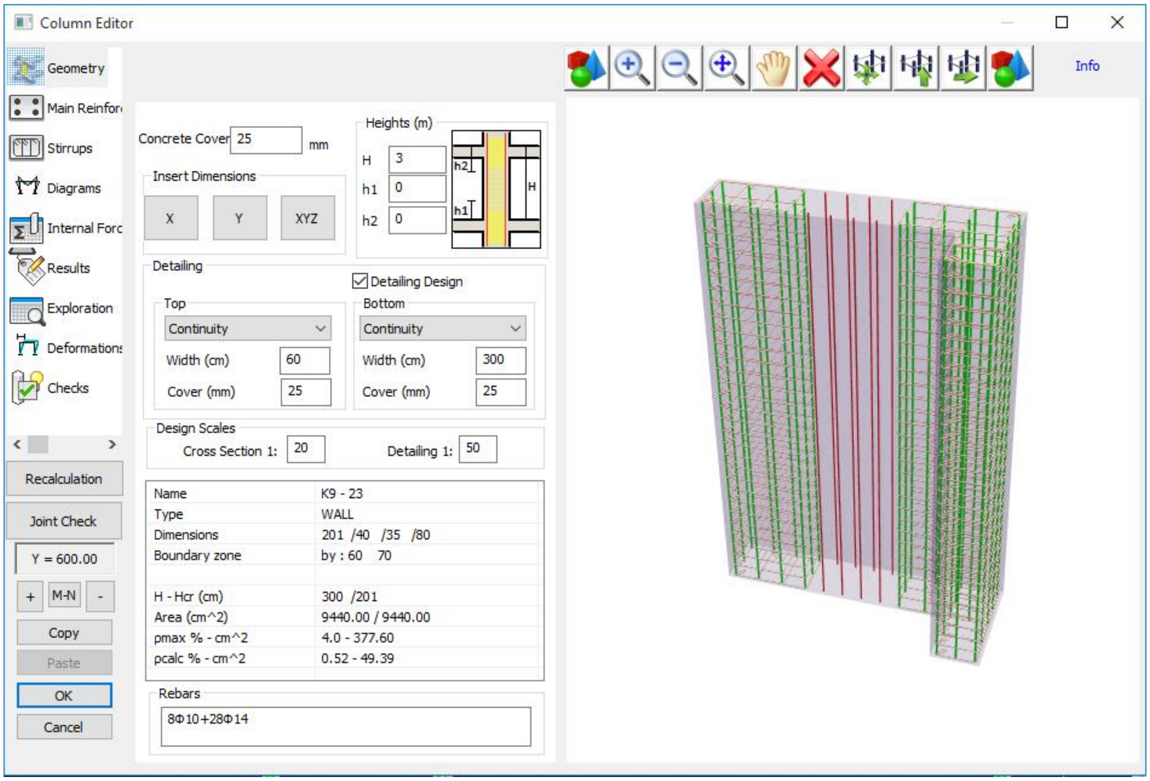


Select the “Results” command to open the .txt file and read the results:




```
================================================================================
|COLUM: K2    - MEMB.: 16    - Connection    (nodes)  Start:16     End   :30    |
|KIND : RECTANGULAR   by=70    bz=70            HEIGHT H= 3.00 Hcr.= 0.70       |
|--------------------------CONCRETE   : C20/25 --------------------------------|
|fck  (Mpa)=20.00 γcu/γcs =1.50/1.0 maxεc(N,M)=0.0035 maxεc(N)=0.002          |
|fctm(Mpa)= 2.20 τrd(Mpa)=0.25                                                 |
|-------------------------REINFORCEMENT----Cover c(mm) = 25 -----------------|
|MAIN       : B500C Es(Gpa)=200.00 fyk(Mpa)=500 γsu/γss=1.15/1.00 max εs=0.02 |
|STIRRUPS   : B500C Es(Gpa)=200.00 fyk(Mpa)=500 γsu/γss=1.15/1.00 max εs=0.02 |
|----------BIAXIAL BENDING WITH AXIAL FORCE Critical combination    6----------|
|        P O S I T I O N         |         BOTTOM          |         TOP        |
|-------------------------------+------------------------+---------------------|
|Max normalised axial force   vd| y: vd=  0.12 comb. 69 | z: vd=  0.12 comb. 69 |
|Applied Axial force   NSd(KN) |        588.06          |        551.31         |
|App.bend.moment       MSd(KNM)|y=  216.18 |z=  175.04 |y=  -53.51 |z=  -58.67 |
|-----------------CONCRETE DEFORMATIONS ENVELOPE (0/00)------------------------|
|Apex Comb. Deform. Apex Comb. Deform. |Apex  Comb. Deform. Apex Comb. Deform. |
|---------Column Bottom----------------+----------Column Top-------------------|
|1    62   -0.2785 |2    62   -0.0216  | 1    68   -0.0916 |2    0    -0.5121  |
|3    38   -0.0868 |4    38   -0.8460  | 3    70   -0.1368 |4    0     0.0000  |
|----------S H E A R   F O R C E  VERIFICATION--------------------------------|
|Seis.shear force Y (KN) Start | VEmin=   46.93 / VEmax=  112.35 = ζ=  0.000   |
|                        End   | VEmin=   46.93 / VEmax=  112.35 = ζ=  0.000   |
|Seis.shear force Z (KN) Start | VEmin=   -0.24 / VEmax=  109.87 = ζ= -0.002   |
|                        End   | VEmin=   -0.24 / VEmax=  109.87 = ζ= -0.002   |
|------------------------------+-Bottom(crit.)-------Span-------Top(critical)--|
|-----Seismic direction--------+---Y-------Z---+---Y-------Z---+---Y-------Z---|
|Applied Shear force   VEd (KN)|   46.9|    9.2|   98.7|  107.3|   99.3|  109.9|
|Applied Tors. Moment TEd (KNM)|    4.4|    4.4|    0.6|    0.6|    4.4|    4.4|
|Resist.without reinf.VRd,c(KN)|  221.9|  221.9|  258.1|  258.1|  235.5|  235.5|
|Resist.comp.struts  VRdmax(KN)| 1079.2| 1079.2| 1079.2| 1079.2| 1079.2| 1079.2|
|Resist.tors.moment TRdmax(KNM)|  244.8|  244.8|  244.8|  244.8|  244.8|  244.8|
|TEd/TRdmax + VEd/VRdmax <= 1.0|   0.06|   0.03|   0.09|   0.10|   0.11|   0.12|
|Shear critical combinations   |(62/-1)|(62/-1)|(40/-1)|(40/-1)|(39/-1)|(39/-1)|
|Req. stirrups   Asw/s  (CM2/M)|    0.2|    0.2|    0.2|    0.2|    0.2|    0.2|
|----Moment resis. Mrd-(KNM)---+----------Bottom-------+-------Top-------------|
|Vector Direction              | +y    -y    +z    -z  | +y    -y    +z    -z  |
|(min) Moment resis.MRd(KNM)   |  168  -167   457     0|    0  -382     0  -529|
|(max) Moment resis.MRd(KNM)   |  579  -363   826     0|    0  -525     0  -644|
```

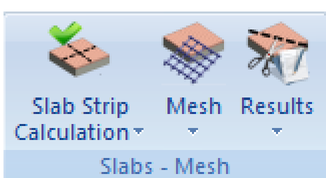
Select “**Detailing**” to open a window for editing the reinforcement of the column - wall in an integrated environment of verification and design:



With this command, you can modify the reinforcement of the column – wall, apply retrofitting methods and calculate the new moment diagrams.

⚠ *Detailed instructions on how to use this command refer to the related user manual (chapter B. Column's Detailing.*

7.6 How to perform Slab Design:

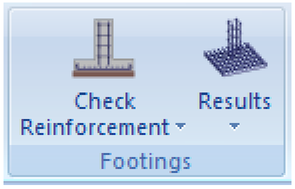


The "Slabs-Mesh" command group includes commands related to the analysis of slabs with the strip method and the corresponding results, and commands to insert, delete, edit and generate a mesh.

Select the command "**Slab – Strip Calculation > Overall**" to calculate all the strips of the current level.

The slab strips are analyzed, the stress resultants are calculated and the designing of the slab is performed. The program calculates the tension (Fe) and compression (Fe') and the steel reinforcement in cm^2. Also the reinforcing bars in span, additional and secondary reinforcement and stirrups, for solid and Zoellner slabs, are calculated.

7.7 How to perform Footing Design



The “Footing” command group contains commands for footing design check, design calculation, editing and the respective results.

.

Select the command “**Check Reinforcement > Overall**“ to check all footings on the current level (foundation).
The program performs the design checks and the corresponding results are displayed by colors and symbols that indicate the type of failure.
The color of the node indicates that the design checks of the footing:

 The footing was satisfied.

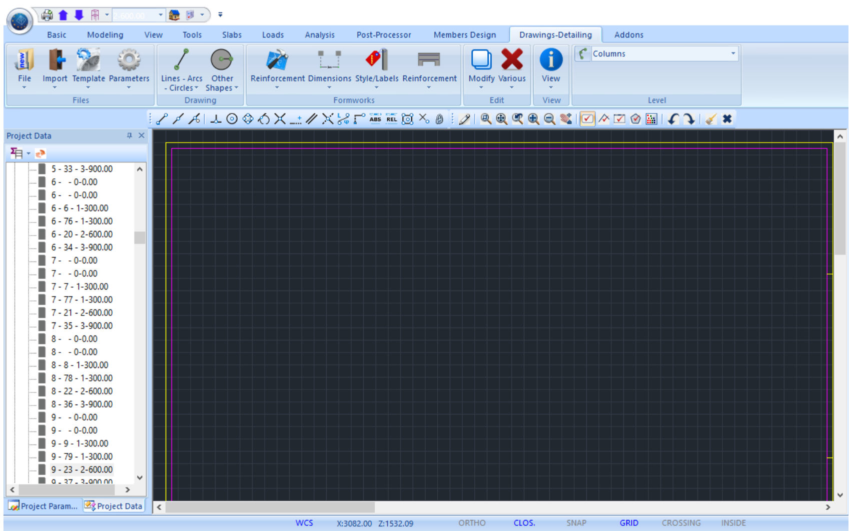
 The footing failed
The type of failure is mentioned with a symbol as well:
(i) “Z” symbol corresponds to the exceedance of the critical load
(ii) “e” symbol corresponds to the exceedance of the load eccentricity
(iii) “σ” symbol corresponds to the exceedance of the normal stresses.

⚠ Necessary precondition for the footing designing, is the columns designing in level 1.

8. DRAWINGS

Since the design and reinforcement of the structural elements of the concrete structures or the design of steel connections of the steel structures have been completed, you can open, modify and finally produce all the drawings in the "Drawing-Detailings” Ribbon.

The “Drawing-Detailings” Ribbon incorporates a drawing application in the interface.

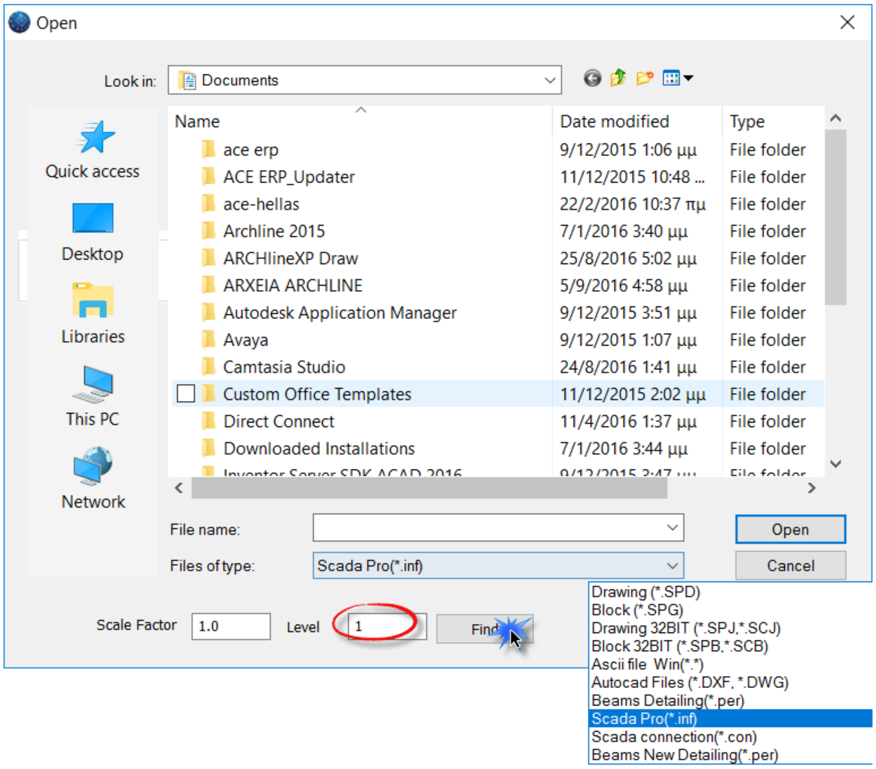


8.1 How to import drawings and beam’s detailing in drawing environment:

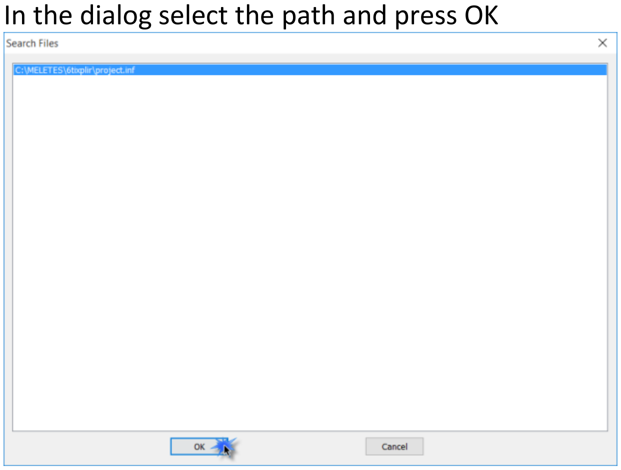
Selecting “Import” command opens the following dialog box for choosing the project’s folder. Then select:

- the type of project from **Files of Type**
- the number of the floor
- the Scale factor

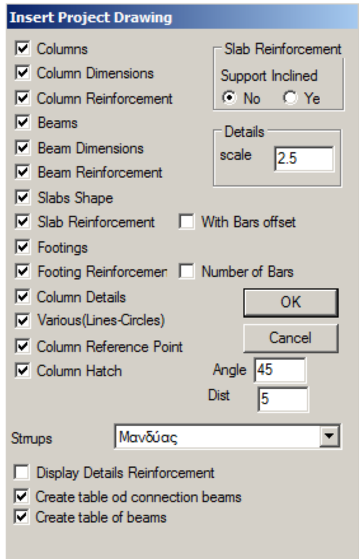
and press **Find.**



In the dialog select the path and press OK



In the dialog box:



❖ Select the elements that will be imported in the design by activating the corresponding checkboxes.

❖ In "**Slab Reinforcement**" select "Yes" if you want the inclined reinforcing bars to be designed as well as the additional reinforcement in slab's supports. Otherwise check "No".

❖ In "**Scale**" type the zoom factor for the columns' detailing.

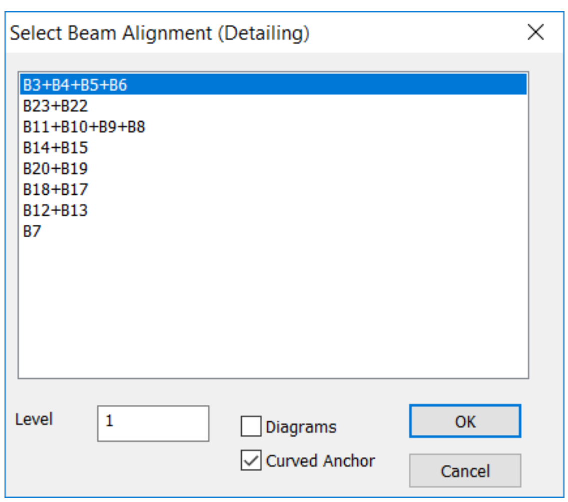
❖ Example: In case that you design a drawing in scale 1:50 and the columns'detailing is in scale 1:20, you will have to type the factor 50/20=2.5

From File Beams' Detailing drawing (*.per) :

Import the beams' detailing in the drawing for the beams 'alignment you will choose from the available ones.

This choice is suitable for the beams' detailing which are created with the existing editor of the beams, while the choice "Beams New Detailing (*.per)". refers to the beams' detailings which are created with the new editor " Design Details"

By choosing the Beams' Detailing (old and new) the path in Find guides you to a new window to select the alignments one by one

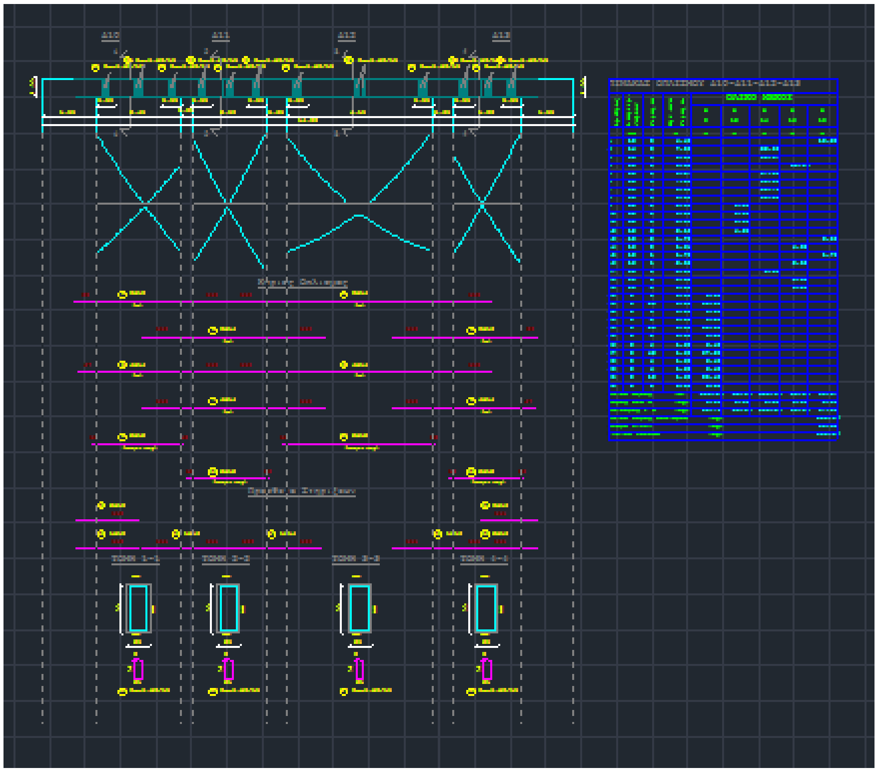


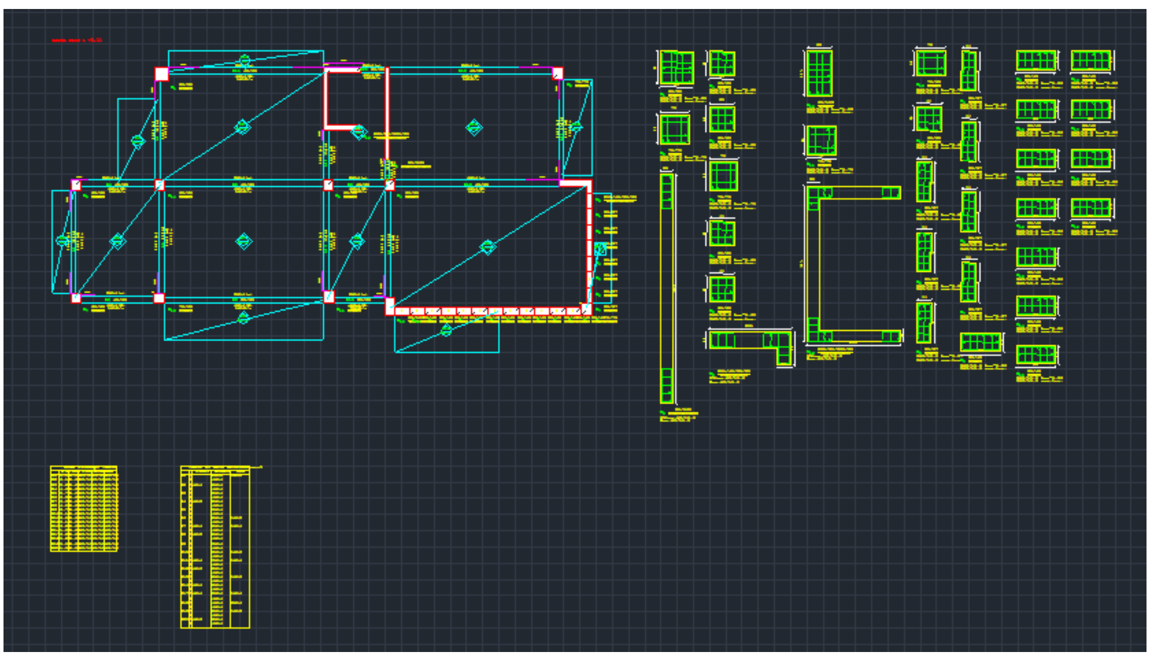
Select the "Level" by typing the level's number.

Active:

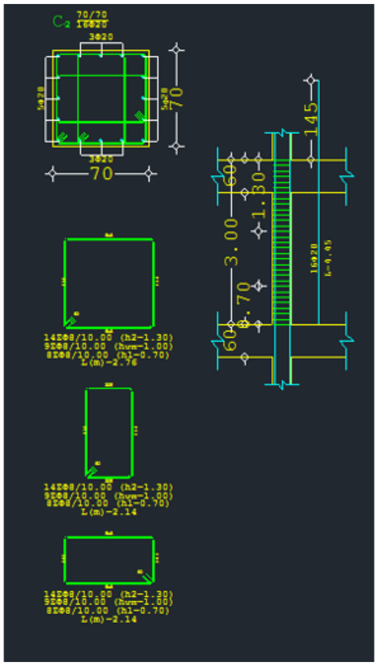
- "Diagrams": Beams' Detailing the corresponding moment diagrams will accompany drawing.
- "Curved Anchorage": The anchorages will be curved at the end.

Select a beam alignment from the list and press "OK". Then left click on the screen to define the position in the design. You xan repeat this procedure for the rest of the levels and details.





8.2 How to import analytical columns details with the ability to perform modifications directly inside the editor:

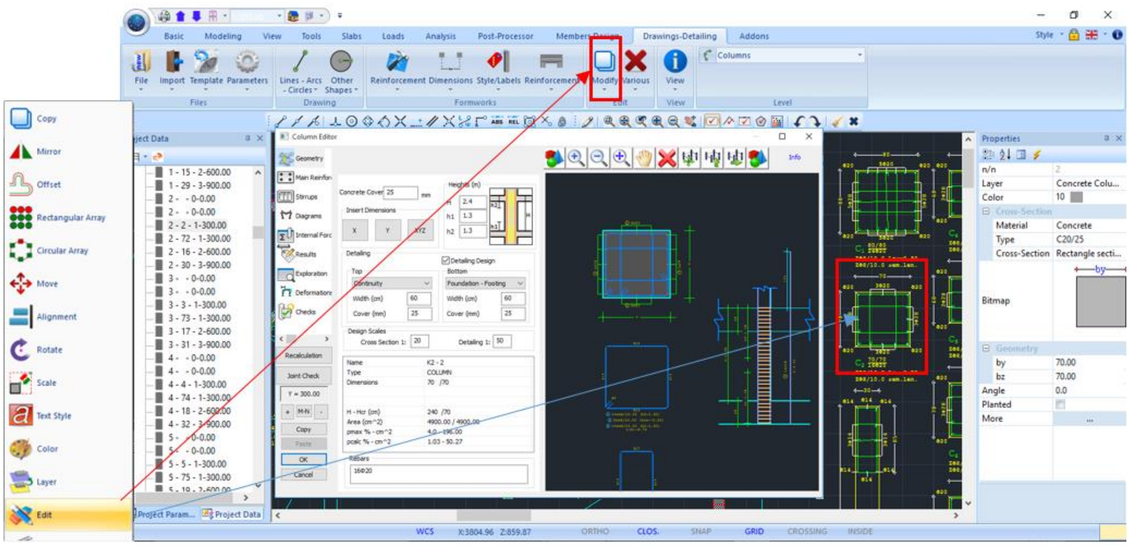


Precondition for the import of analytical columns and walls detailing inside the drawing environment is:

1. To have already selected the "Detailing" for the respective columns and walls, and
2. To click "OK" to the respective windows.

Then, and only the, importing the "project. Inf" file, will also import the analytical details of columns and walls.

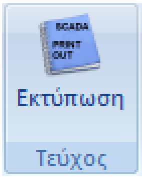
Using the "**Modify > Edit**" command you can modify detailing drawings directly inside the editor.



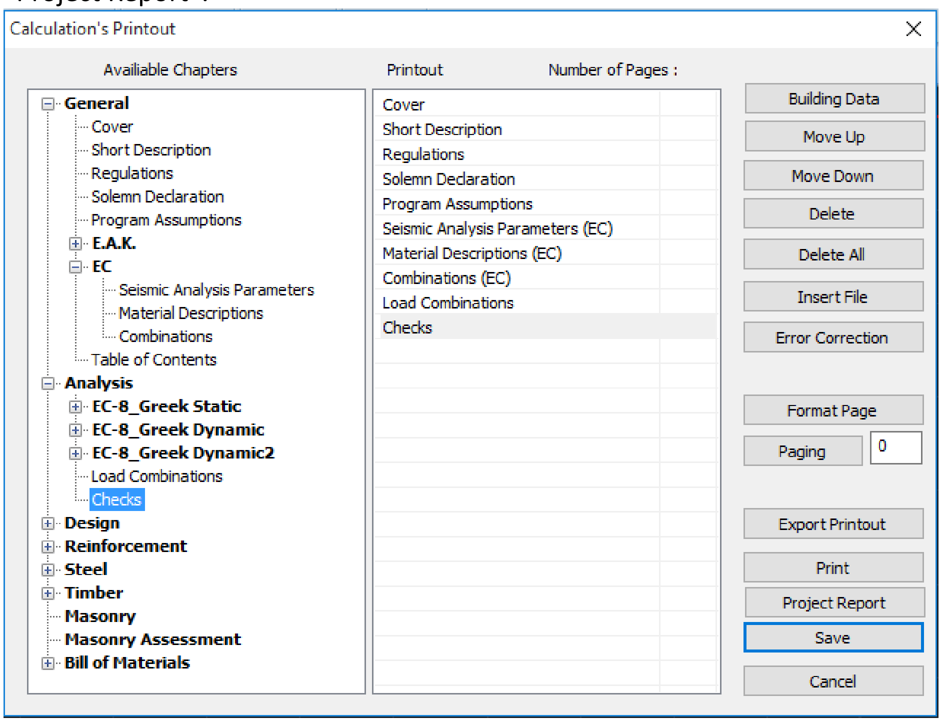
Select "**Edit**" and click on an object. The corresponding dialog box, in which you can change the corresponding parameters, opens. Click OK to save the changes that automatically update the drawing and the report as well.

9. PRINT

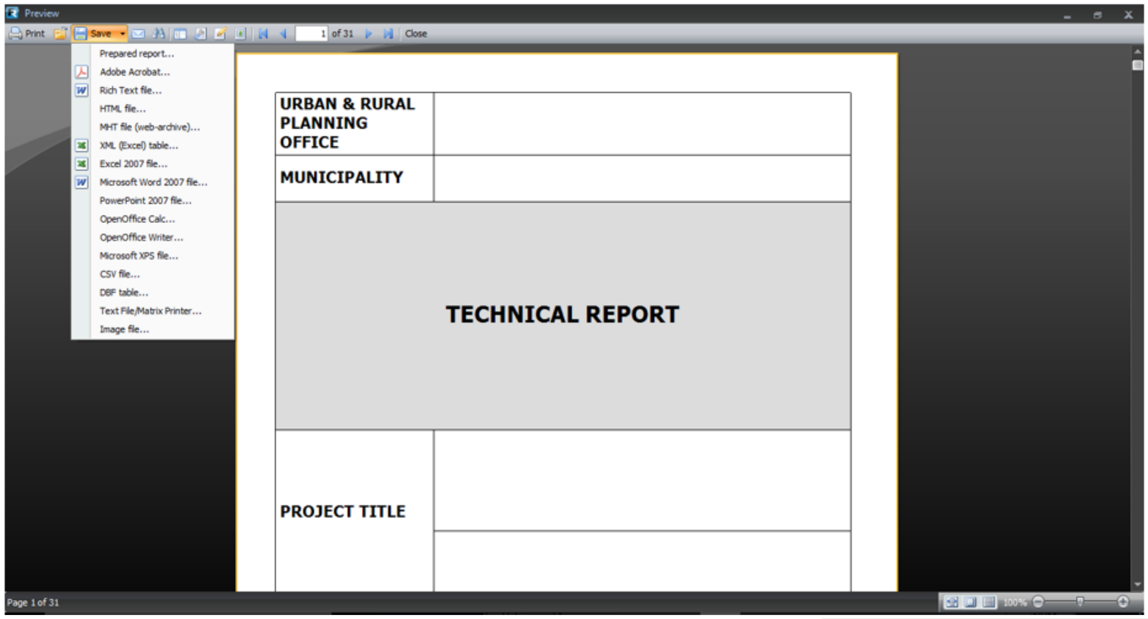
9.1 How to create the Project Report:



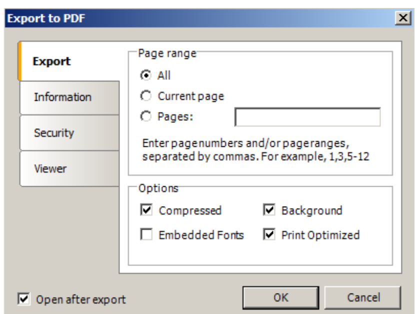
To create the project report, select the “Addons” unit and click Print.
In the dialog box “Calculation Printout”, on the left, the list with the Available Chapters is displayed. Double click on the selected chapter to show it on the right list.
Fill in the Printout list by double clicking on the available chapters and then press “Project Report”.

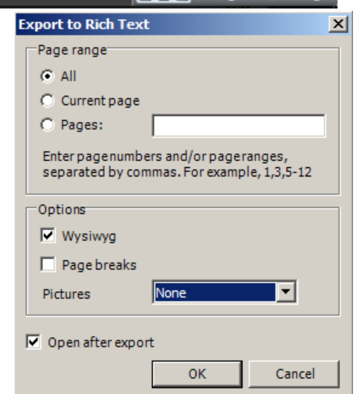


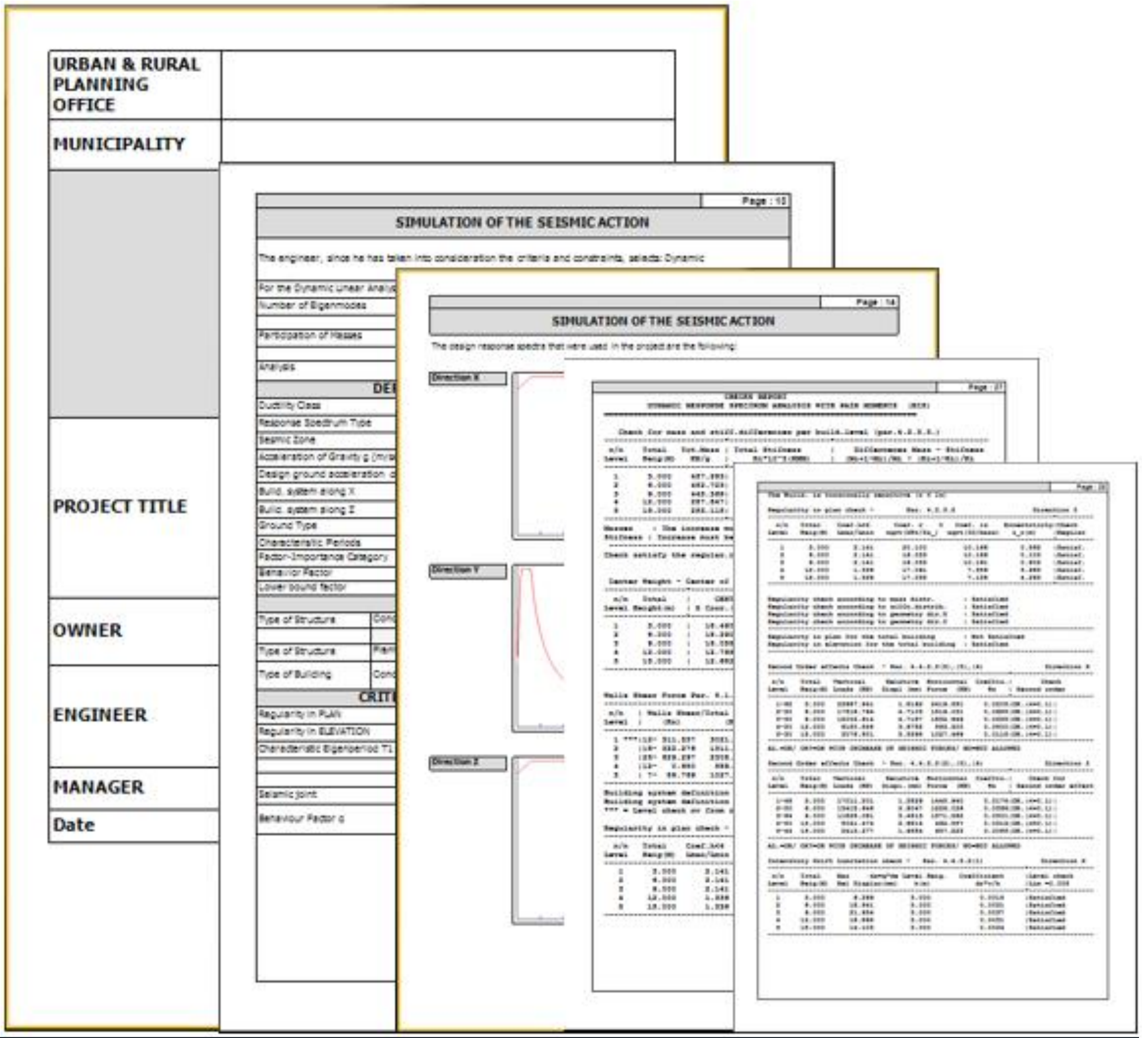
Click “Project Report” to open the print preview of your report.



You can save the file in PDF, or DOC, XLS, XML format and edit it.






URBAN & RURAL PLANNING OFFICE
MUNICIPALITY
PROJECT TITLE
OWNER
ENGINEER
MANAGER
Date
SIMULATION OF THE SEISMIC ACTION
SIMULATION OF THE SEISMIC ACTION