SCADA Pro™
Structural Analysis & Design

Example 1

New Building Design from Reinforced Concrete

ACE HELLAS
INTEGRATED SOLUTIONS

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

CONTENTS

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

FOREWORD

The product of SCADA development is the NEW upgraded SCADA Pro. It is a new program that includes all the applications of the "old" one and incorporates additional technological innovations and new features.
SCADA Pro offers a single integrated environment for the analysis and design of new structures, as well as the control, evaluation and enhancement of existing ones.
It combines linear and surface finite elements, incorporates all applicable and non-applicable Greek regulations (N.E.A.K., N.K.O.S., E.K.O.S. 2000, E.A.K. 2000, E.A.K. 2003, Old Seismic, Method of Allowable Stresses, KAN.EPE, KADET) and the corresponding Eurocodes.
It offers the designer the possibility to design structures of different materials, concrete, metal, wood and masonry, individually or mixed.
With the use of new cutting-edge technologies and based on the requirements of construction project designers, a program was created with a number of smart tools with which we can create 3D constructions, process them in the field and build the final structure in simple steps and complete even the most complex studies.
SCADA Pro is a program that is constantly upgraded, evolving and adapting. The technical department of ACE-Hellas in permanent cooperation with Metsovio Polytechnic University is engaged in its continuous development and its updating based on new data, applications and needs. A "living organism" that matures!

INTRODUCTION

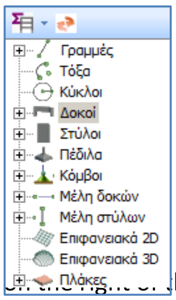
This manual was created to guide the designer in his first steps in the new SCADA Pro environment. It is divided into chapters and based on a simple example guide.
This is not an actual study, but an educational example aimed at understanding the process and mandates of the program rather than consistency with regulations.
Each chapter contains information useful for understanding both the commands of the program and the procedure to be followed in order to perform the input, check and dimensioning of a reinforced concrete structure.

THE NEW ENVIRONMENT

In the new interface SCADA Pro uses the technology of RIBBONS for even easier access to the commands and tools of the program. The main idea of the Ribbons design is to centralize and group similar commands in the program, so that you can avoid navigating through multiple levels menus, toolbars and tables, and make it easier to find the command you want to use.

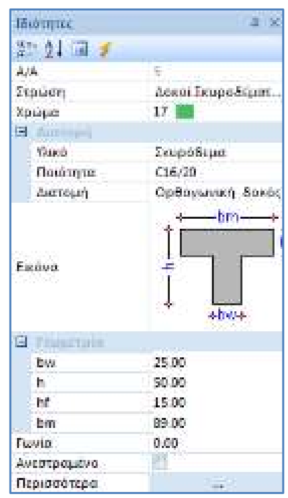
EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'


The user has the option, for the most frequently used commands, to create his own group of commands for easy access to them. This toolbox is maintained after closing the program and you can add and remove commands as well as move it via the "quick access toolbar customization" that opens by right-clicking on the command.



The new SCADA Pro environment displays on the left side of the screen all the entities of the construction categorized in a tree format either per level or for the whole building as a whole. This categorization allows easy identification of any element and by selecting it, it is displayed in a different color in the entity. At the same time, the level to which it belongs is isolated , while its properties displayed on the right side of the screen with the possibility of modifying them directly. This function can be performed bidirectionally, i.e. the selection can be made graphically on the vector and the element is automatically displayed in the tree with its properties of the right of the screen. It is also possible to apply specific commands to each element of the selected tree. The menu of commands is displayed with the right mouse button and this menu changes depending on the section of the program that is active.

The "Properties" list displayed on the right is automatically populated by selecting an element of the operator. It informs the user of its attributes, as well as allowing changes to them.

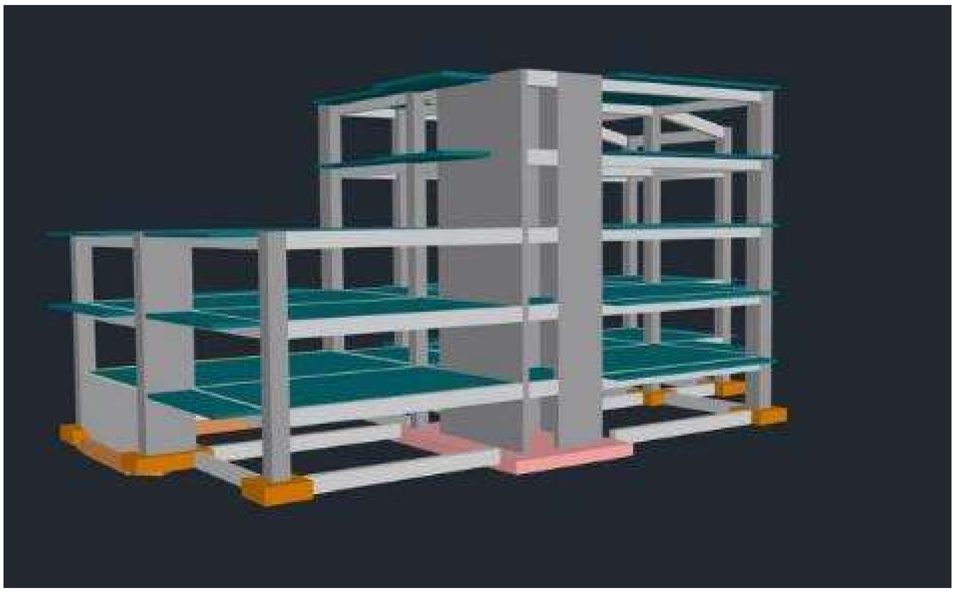


EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

GENERAL DESCRIPTION

1.1 Geometry

The building under study consists of a basement and four above-ground floors. A portion of the basement is surrounded by walls and the fourth floor includes a sloping section. The foundation is mixed and consists of a section with footings, footings and connecting beams and a section with a cavity foundation. 2 different floor plans will be used to create the static carrier.



1.2 Materials

For the construction of all members of the structure, concrete of quality C20/25 and for the reinforcement B500C quality steel.

1.3 Regulations

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

1.4 Loading - analysis assumptions

Dynamic Spectral Method with homosynchronous torsional pairs.
The loadings according to the above analysis method in SCADA Pro are as follows:
(1) G (permanent)
(2) Q (mobile)
(3) EX (epicyclic loads, earthquake forces at XI, from dynamic analysis).
(4) EZ (epicyclic loads, ZII earthquake forces, from dynamic analysis).

(5) Erx ± (epicontic torsional moment loads resulting from the epicontic forces of the earthquake XI displaced by the random eccentricity ±2eτzi).
(6) Erz±(epicyclic torsional moment loads resulting from the epicyclic forces of the earthquake ZLI displaced by the random eccentricity ±2eτxi.
(7) EY (vertical seismic component -earthquake by y- from dynamic analysis).

1.5 Comments

All the commands used in this example, (and all the other commands in the program) are explained in detail in the **User Manual** that accompanies the program.

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

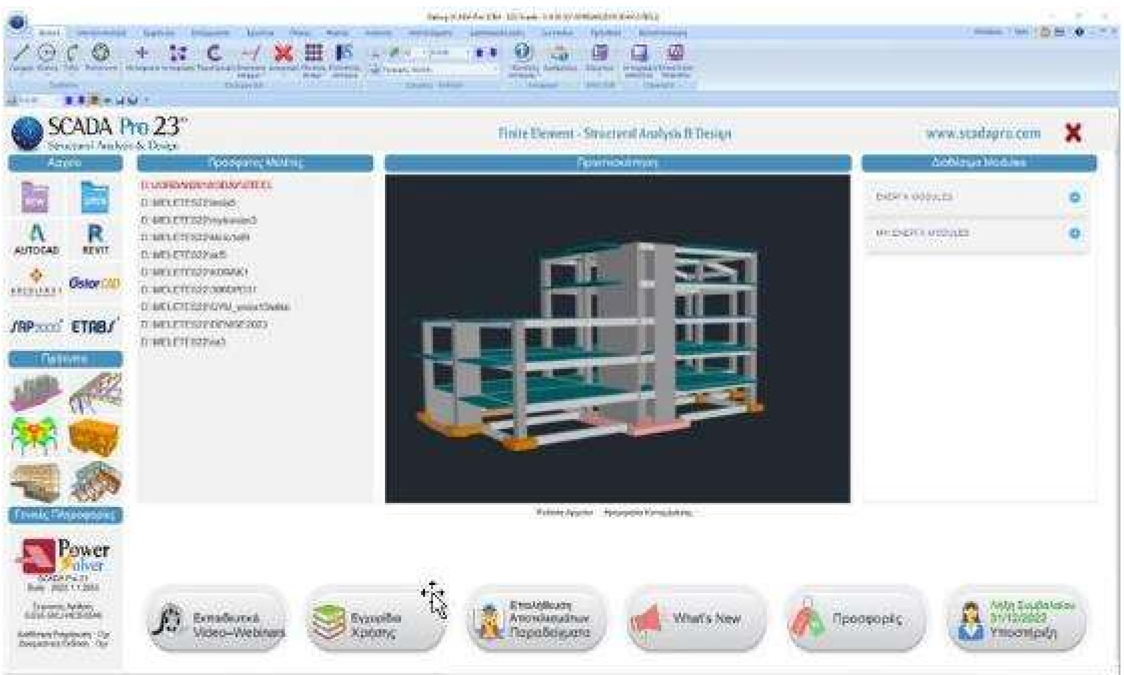
DATA IMPORT - MODELLING

2.1 How to start a new study:

SCADA Pro offers a variety of ways to start a new study. Some criteria for choosing a starting point are: construction materials, the records available to the designer in collaboration with the architect, the shape of the floor plan, the choice of using linear and/or finite elements, etc.

> ⚠ This example will detail how to use dwg files to import data and model a concrete carrier.

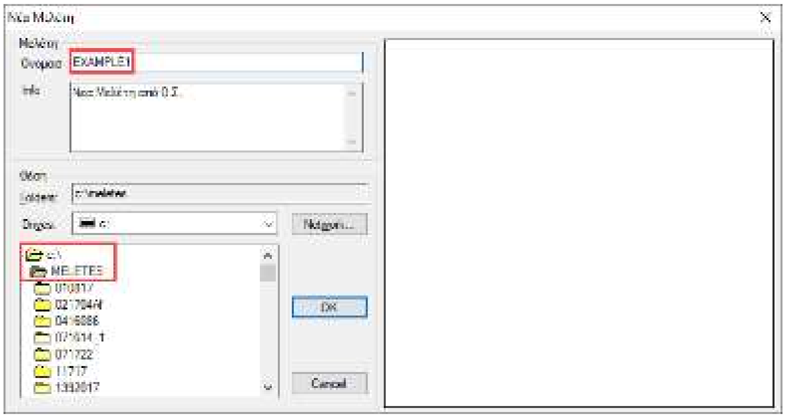
Upon opening the program, the startup window appears on the screen, which includes a set of commands to start the program:



Pressing the left mouse button on the respective icons will result in one of the following startup modes:

> ⚠ Regardless of the way you choose to start a new study, the same window always opens where you specify a Name and path for the file entry, a <u>procedure necessary for the operation of the program commands</u>.

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'



IMPORTANT NOTE:

The file name must consist of **a maximum of 8 Latin characters and/or numbers, without spaces and without the use of special characters (/, -, _)** (e.g. ARXEIO1). The program automatically creates a folder where it enters all your study data. The "Location" of the folder, i.e. the place where the study folder will be stored, should be on the local C drive, exactly where the "Scada19" program folder is located, but outside of it.

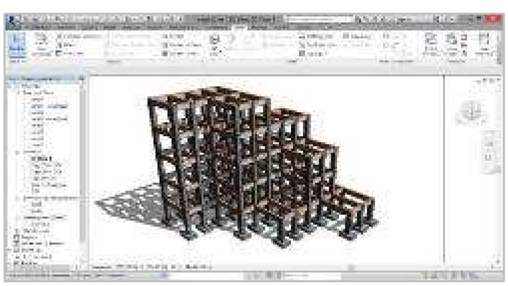
It is recommended to create a folder in C (e.g. MELETES), where all SCADA studies will be located (e.g. **C:\MELETES\ARXEIO1**)

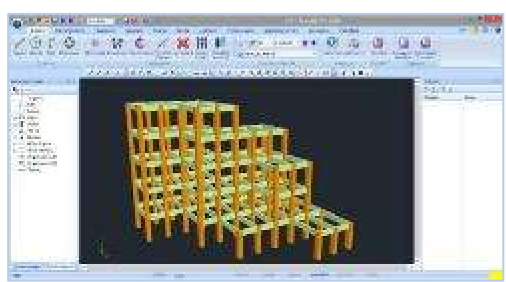
If you wish, write some general information about the study in the "Info" field.

ADDITIONAL INFORMATION:

*"**new**": usually used when there is auxiliary file electronic form. The startup is done in a blank interface. O the designer starts by defining the stations and inserting the cross-sections, using the modelling commands and with the help of the pulls of the canvas.*

*"**REVIT**": read ifc files from Autodesk's Revit program. Using appropriate libraries, it automatically identifies all structural elements (columns, beams, slabs, etc.) with their respective properties so that the structure is ready for analysis.*

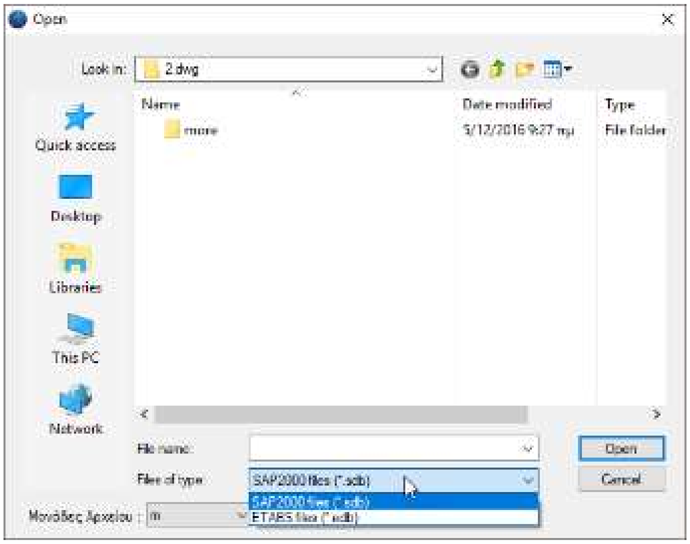




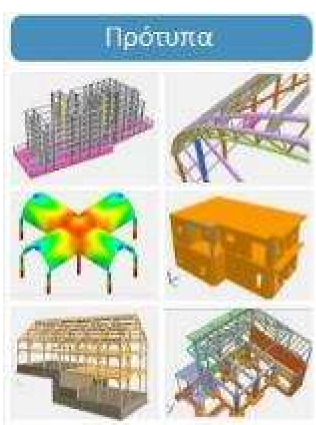
*"**ArchlineXP**": reading xml files from the architectural program ArchlineXP.*

*"**ETABS, SAP2000**": reading .edb & .edb files .sdb from the static programs ETABS & SAP2000 .*

The new bi-directional communication of SAP2000 and ETABS with SCADA Pro, allows the import and export of any project to SCADA Pro and SAP2000/ETABS, respectively.



*"**Standard Constructions**":SCADA Pro has a rich library of standard constructions for all materials. The standard constructions tool can be accessed in 2 ways: either by left-clicking on one of the icons on the home screen, or by using the command MODEL>MODEL>MODEL>TYPICAL CONSTRUCTIONS. A detailed description can be found in the corresponding chapter of the user manual (Chapter 2. Modelling)*



concrete metal

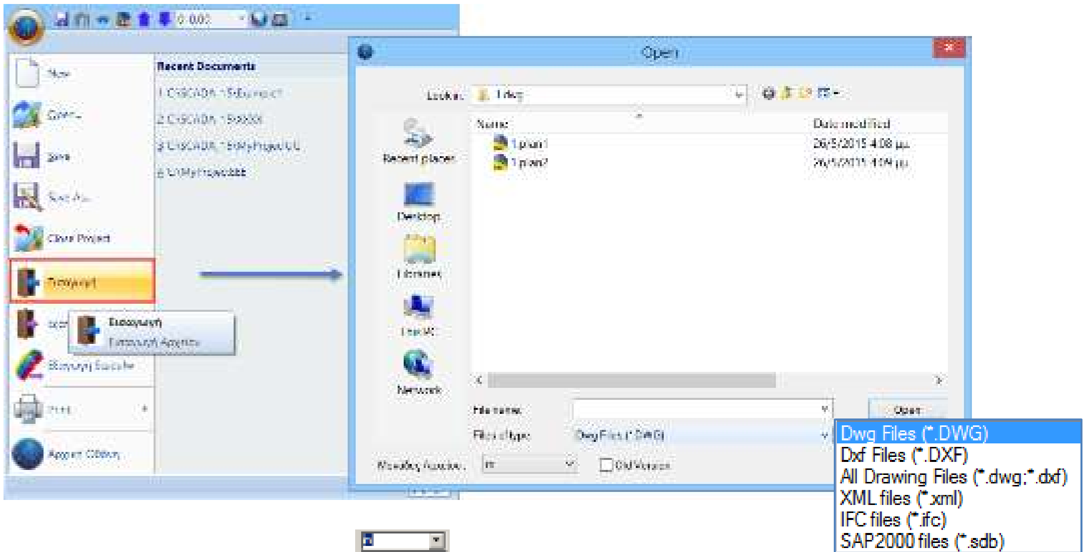
surface connections(IDEA StatiCa)

wooden masonry

Usually, before the structural design of a concrete building, an architectural study is foreseen, often accompanied by dwg or dxf files. These files can be read and used by SCADA Pro in a variety of ways.

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

Import dwg or dxf file as an auxiliary file for the import of the cross-sections of the static elements either manually, semi-automatically or fully automatically.



Remember to select from the list  *the correct unit of measurement, i.e. the one used when creating the .dwg, .dxf file.*

In addition, besides cad files, you can import Revit, SAP2000 etc. files into the SCADA Pro interface. The cooperation of SCADA Pro with Revit is even more complete, since it is not only limited to importing design auxiliary files, but also the entire vector.
The cooperation of the new SCADA Pro with SAP2000 offers the possibility of importing any type of structure into SCADA Pro for the dimensioning of reinforced concrete, metal, load-bearing masonry and wooden structures based on the respective Eurocodes and the Greek National Appendices.

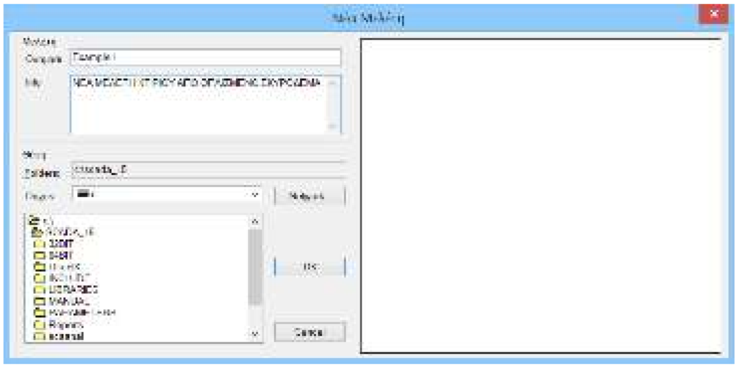
*"**dwg-dxf**": another way is by importing an auxiliary dwg or dxf file, but in the new SCADA Pro it is not just a background that provides tractions on the drawing lines, not even a semi-automatic way*
data input with manual selection. It is a completely automated tool that allows the reproduction of a floor plan on the selected floors and automatic creation of the vector.
The command is used for this example and is described in detail below.

2.2 Automatic Section Recognition from dwg file:

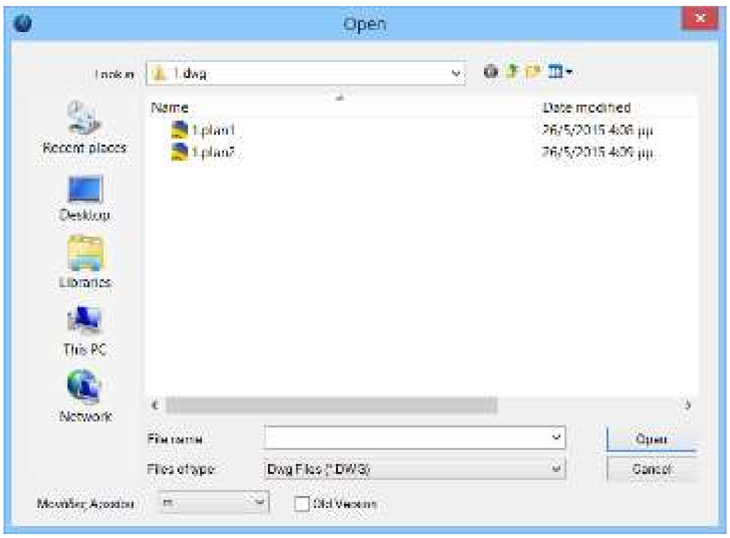
- Select the relevant icon and in the dialog box 

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

Set the Name and location of the file. If desired, write some information about the study in the "Info" field and OK.



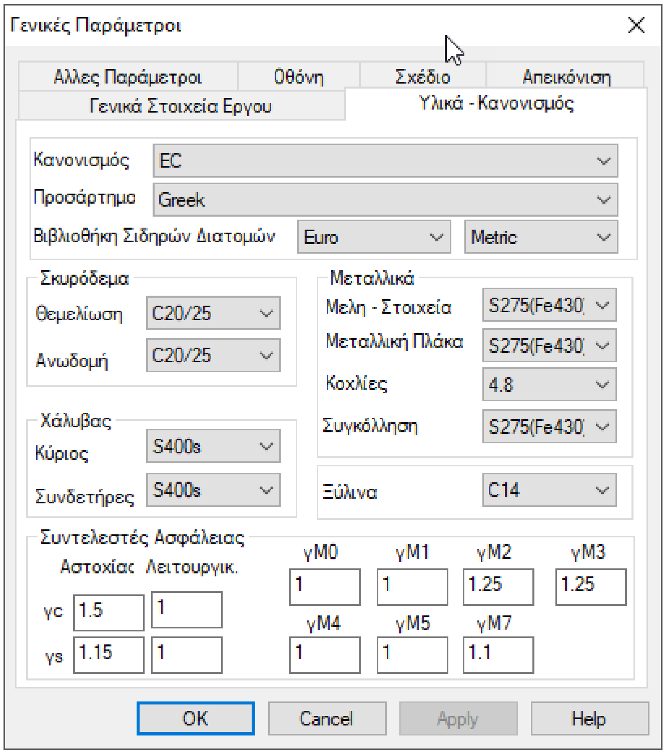
In the next window that opens, select the auxiliary file and Open.



⚠️ In cases of organisations without a standard floor, or with several standard floors, or with completely different floor plans in height, there is a need to import more auxiliary files. SCADA Pro enables the designer to import as many dwg/dxf files as desired. These are saved in design file and can be used to create the static model, combining the fully automatic mode with the semi-automatic and manual modes.

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

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

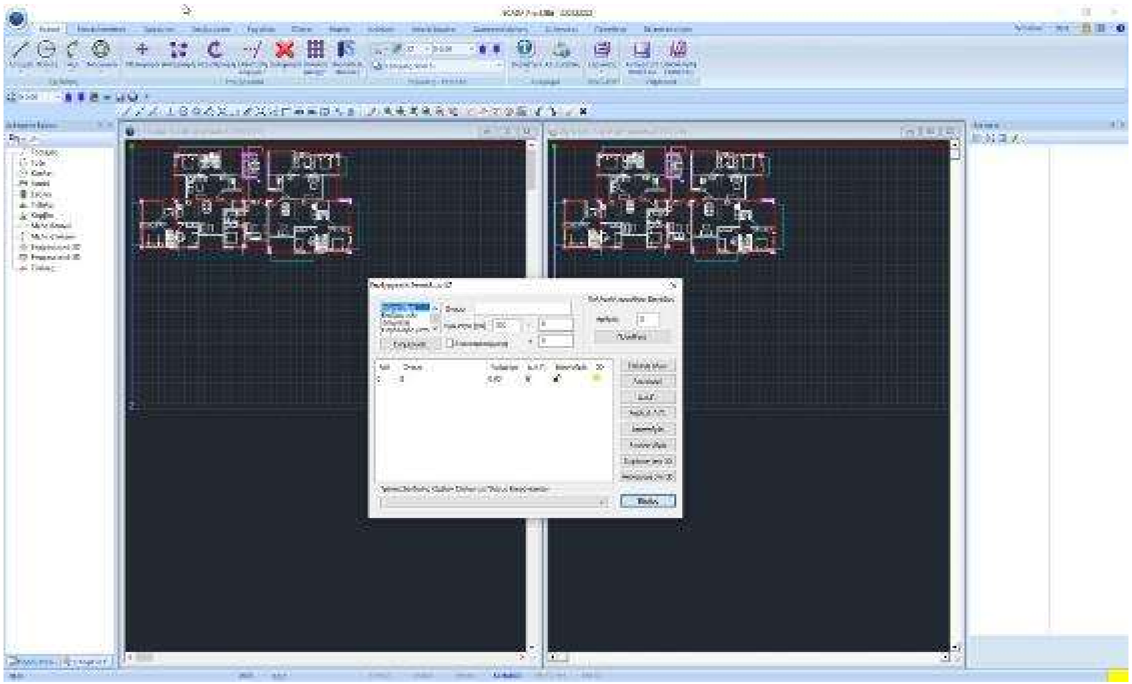


ATTENTION: The materials must be in accordance with the selected regulation, and when entering data, all cross-sections must have the correct grades (C for newer regulations, B for older ones)

* Predefined scripts are created according to the Rules and Attachment option you make at the beginning, within the General Parameters window that opens automatically immediately after you define the file name.

- OK and automatically the design opens in the SCADA Pro environment, with all its design elements, in two separate windows, which will later offer me a 2D and a 3D visualization.

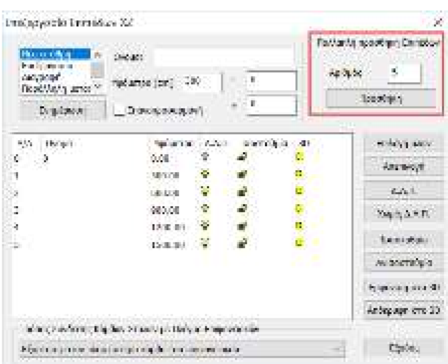
EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

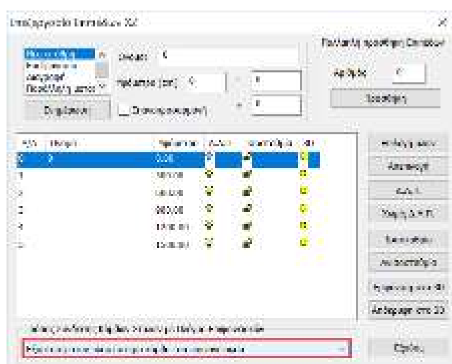


- At the same time the "**Edit HZ Levels**" window opens, to set all the levels of the vector. By default only the foundation level (level 0) is defined and you define the other levels of the whole design.

To create a new level, select "**New Level**" and enter the name and altitude. The **-** and + fields are filled in if there are unevenness or slopes in some levels. By selecting "**Edit**" and a level from the list you can change the name and altitude.

There is now also the possibility to create stations automatically, in the "Multiple Level Addition" section. Set the Number of levels to be created and press "**Add**":





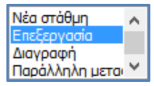
EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

IMPORTANT NOTE:

⚠ Make sure to set "**Dependency on the nearest surface node**" for level 0, so that the nodes of the column members will automatically depend on the nodes of the slab we will create in the foundation.

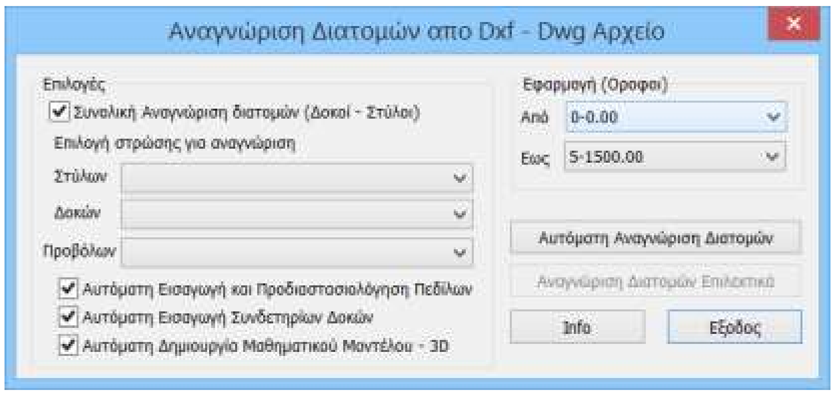
The list shows the levels with a height difference of 3m (300cm), editable via

the "Edit" option



(see the corresponding chapter of the Manual)

- Close the window to automatically display the next window of "**Cross-section Identification from Dxf - Dwg File**".



It is an automation system that recognizes beams, columns of any cross-section (T, P, C), plates and cantilevers, pedestals and connecting beams, while at the same time it automatically creates the mathematical model of the structure.

The list with the arrow next to "**Select layer to identify**" Columns, Beams and Bollards, includes all Layers of the .dwg auxiliary file.

OBSERVATION:

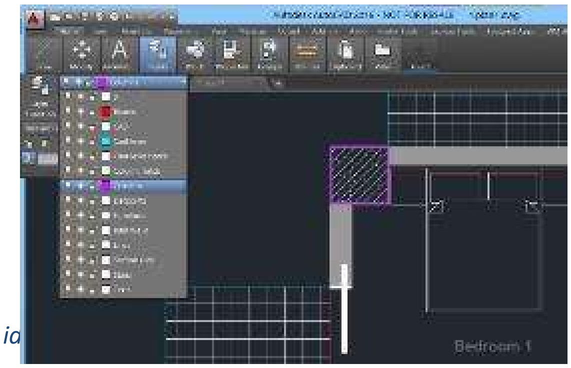
⚠ *The correct functioning of the cross-section recognition automation is ensured by some simple conditions that must be provided for when designing the auxiliary file.*

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

CONDITIONS:

1. *Each floor plan to be used as an auxiliary file should be in a separate file that does not include any other drawings other than the floor plan with all its design entities.*

2. *The lines (lines and/or polylines) defining both the columns and beams and projections belong a single separate layer of their own.*

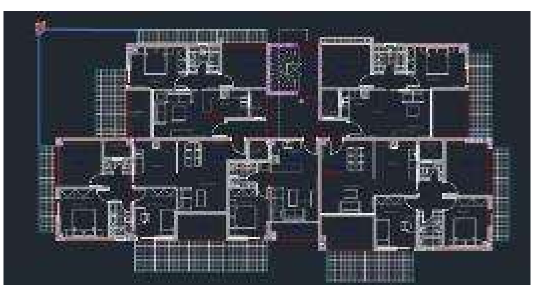


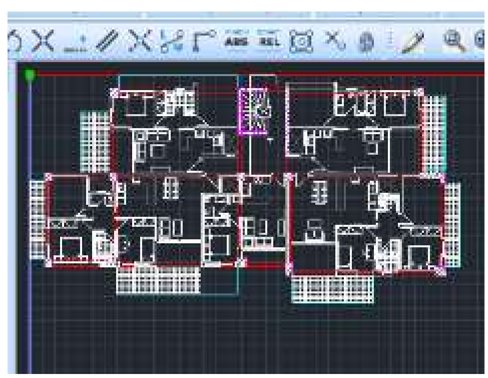
3. *The utility Archive is imported from SCADA Pro environment in the active level XZ by ia*

   projection of all points of the plan . 

This means that any floor plan that is imported should be clear of random lines or other marks in the drawing's surroundings to avoid any displacements. To find the insertion point inside the drawing you can define the outlined rectangle of your drawing and you will know that its upper left corner will its insertion point inside the SCADA Pro environment.

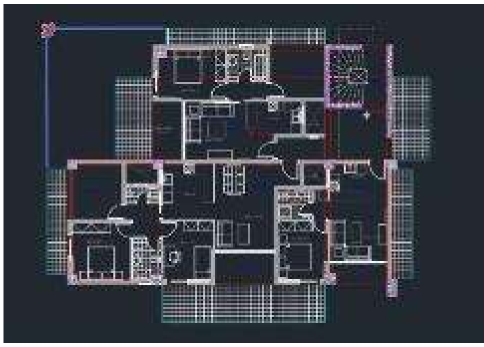
When inserting more height-independent auxiliary drawings, pay attention to the insertion point in order to achieve the correct height continuity of the floors.



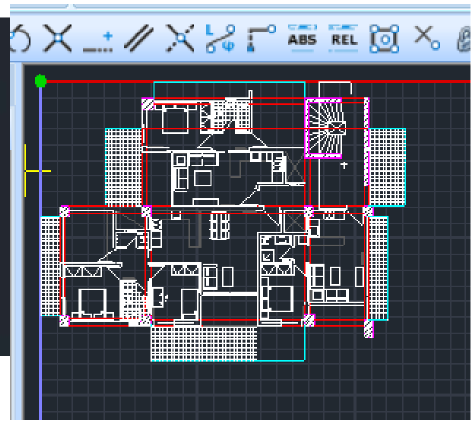


Plan view 1 (dwg) *Plan view 1 (Scada)*

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

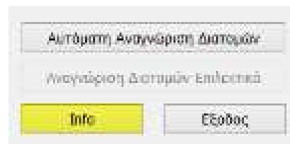


Plan view 2 (dwg)



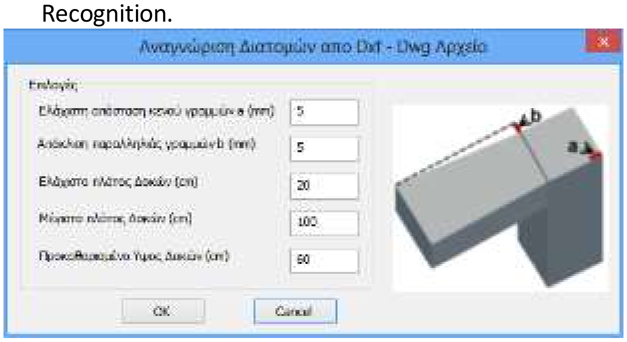
Plan view 2 (Scada)

- The "**Info**" button offers the possibility of some options relating to design flaws and the



limit

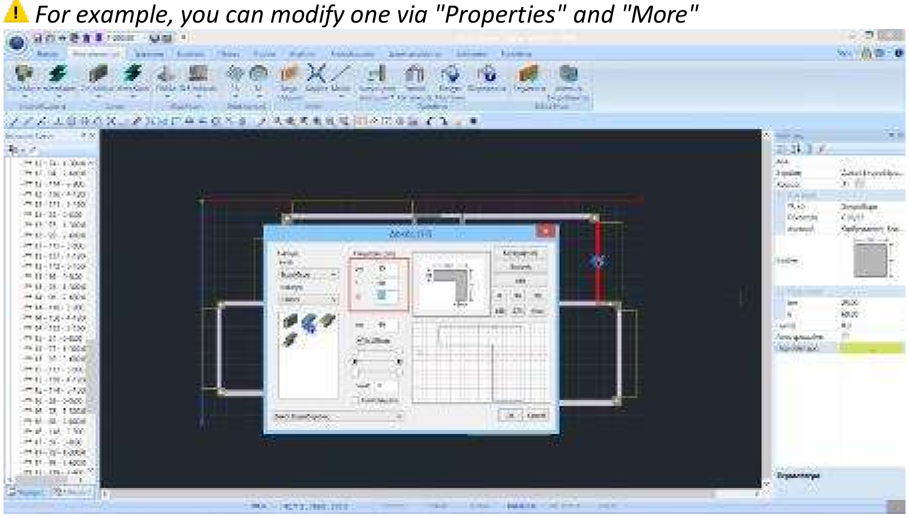
s so that they are not taken into account in the automatic Recognition.



For the identification of the beams, the limiting distances of two lines in the layer of the beams are defined, as well as the predefined height, i.e. the hanging of the beams that the program recognizes from the floor plan and always enters with a rectangular cross-section.

The modification of the cross-sections of the beams after their insertion can be done in several ways.

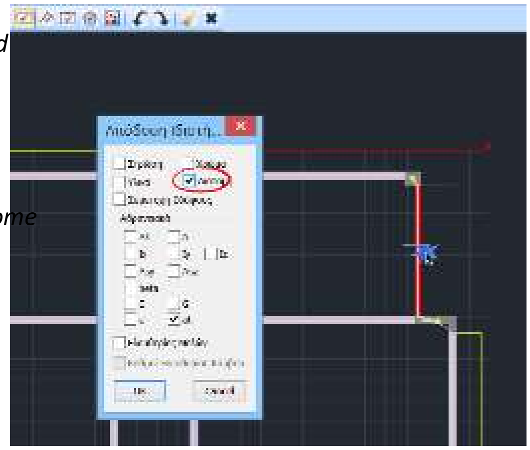
⚠ *For example, you can modify one via "Properties" and "More"*

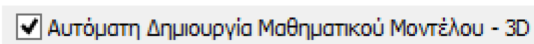


and then using the "Assign Properties" command

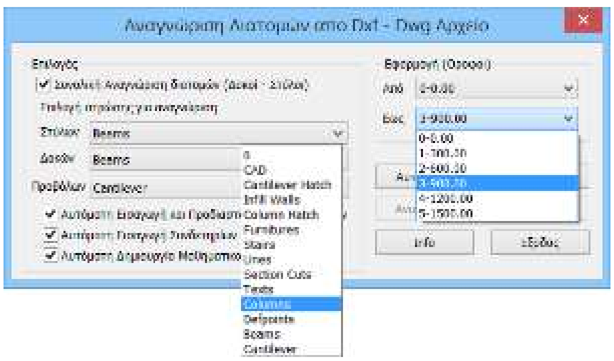
modify the beams that have the same cross-section by selecting them with some from their known ways


.





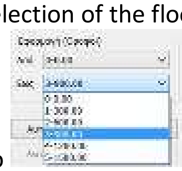
By activating the automatic creation of the mathematical model, the program not only recognizes and enters the physical cross-sections (physical model), but also calculates the inertial elements and creates the mathematical model directly.

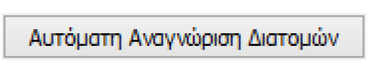


The basic requirement for the automatic recognition of plates and projections is that both the columns and beams have been selected for creation, and that the automatic creation of Mathe is activated. Model, so that the members that will surround the slabs exist.

"**Apply (From-to)**" allows the selection of the floors to reproduce the model.

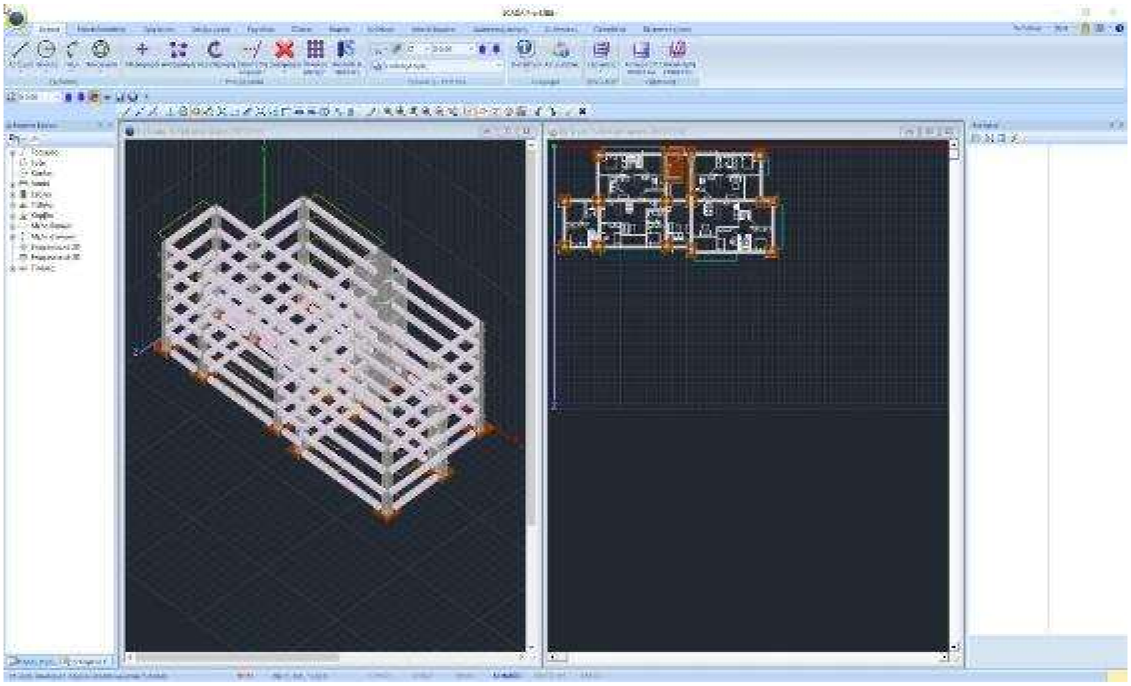
The example defines: From 0 To



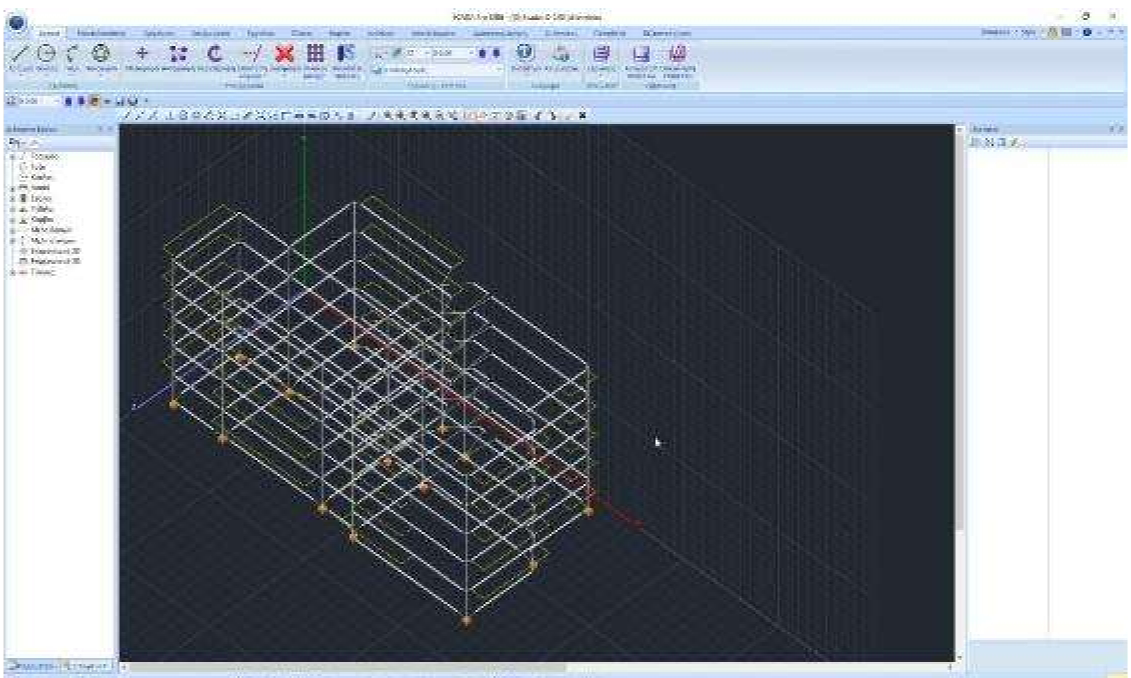


Select "**Automatic Cross Section Recognition**" to display the 3D and 2D representation of the model on the screen.

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'



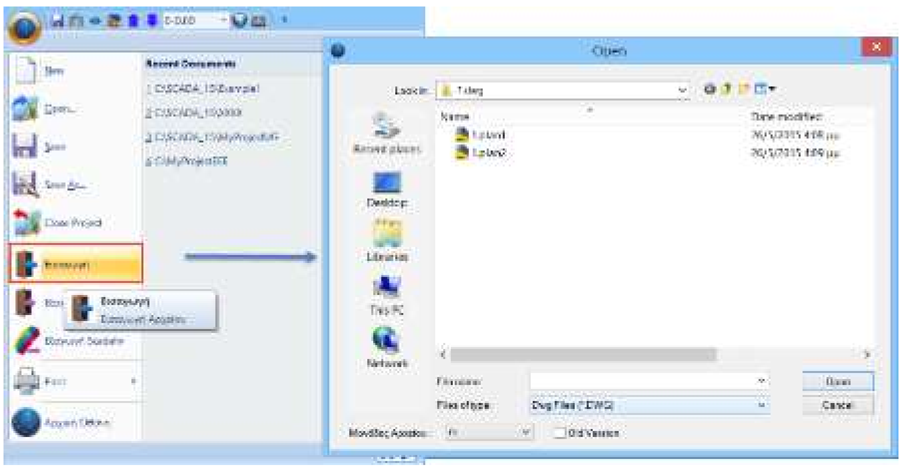
You can keep the 2 windows, add more or close by continuing in one :



2.3 Insert a new floor plan (new dwg file) into the existing model to create the additional floors:

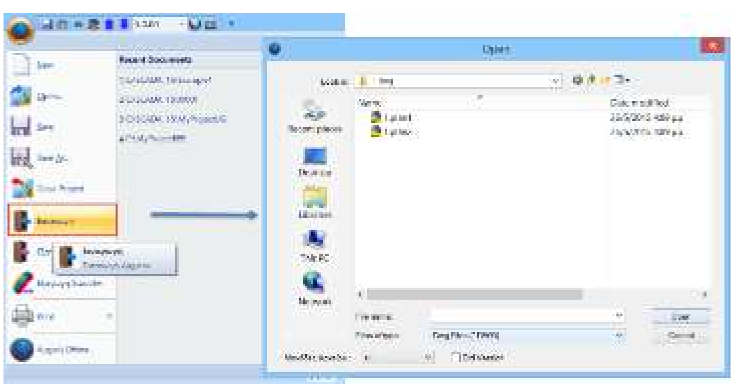
Through command AUTOCAD that appears in the initial window, an auxiliary file can be imported with the possibility of automatic modelling.

For each subsequent auxiliary file in the same study, use the "**Insert**" command and with the corresponding blank level HZ of SCADA Pro active, insert the drawing.



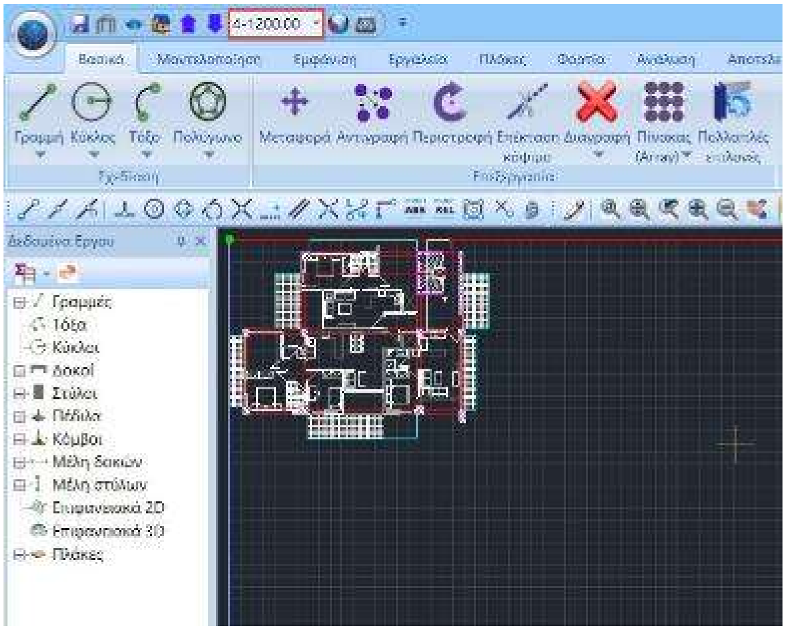
Having chosen to reproduce the first floor plan (*plan1.dwg*) for floors 0 to 3, levels 4 and 5 of the model do not include any elements.
For the identification of the elements of the second floor plan (*plan2.dwg*) on levels 4 & 5 <u>follow the automated procedure that includes:</u>

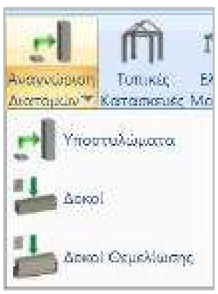
- The "**Import**" of plan2.dwg into the active empty level HZ of SCADA Pro (level 4) Display the empty level 4 on the desktop and select the command Import and the 2h floor plan:



Automatically the floor plan is displayed in the SCADA Pro interface

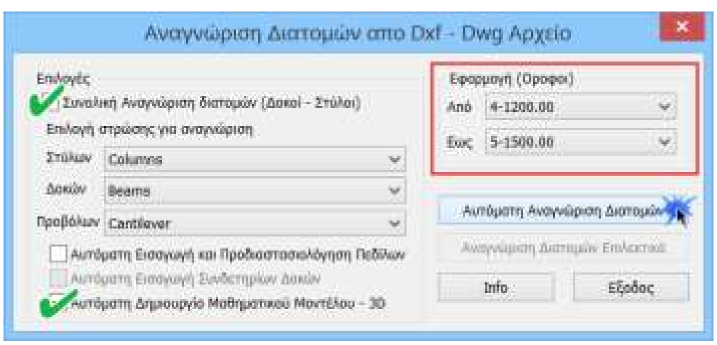
EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'





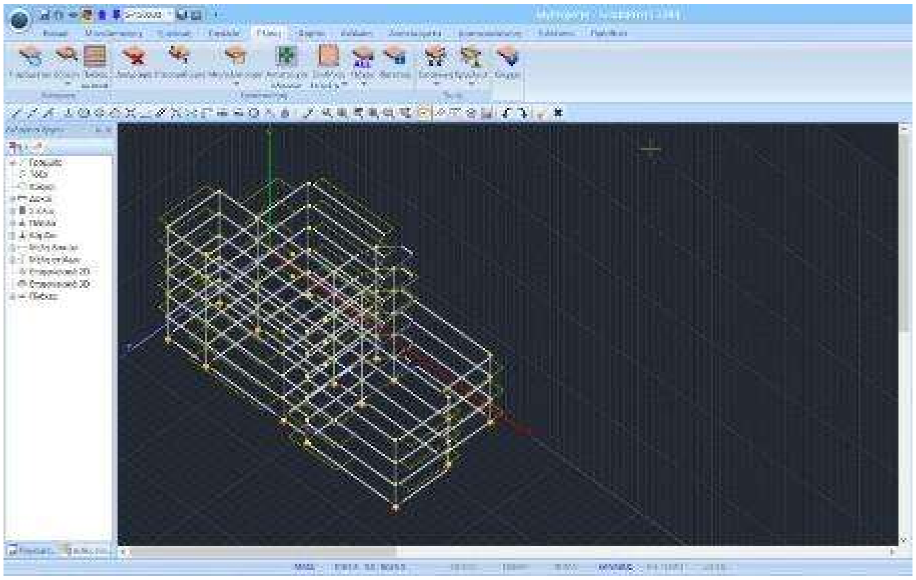
- On automated modelling via of "**Modelling**" module and the "**Cross-sectional Recognition**" command.

Select Columns or Beams and in the dialog box activate:



- The "Overall Cross-section Identification (Beams - Columns)" which in turn activates all the fields for the selection of the corresponding layers for identification of both Beams, Columns and Cantilevers.
- The "Automatic Creation of Mathematical Model - 3D"
- The "Application" at levels 4&5
- The "Automatic Section Recognition"

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'



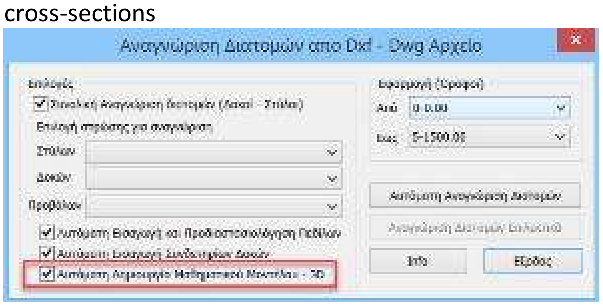
2.4 Mathematical and Physical Model:

When we refer to the modelling of a building block we mean the creation of the PHYSICAL and the MATHEMATICAL of the model.

- The PHYSICAL model is the cross-section, i.e. the geometry and material of a structural element.
- The MATHEMATICAL model is their mathematical properties, i.e. its inertial, its freedoms.

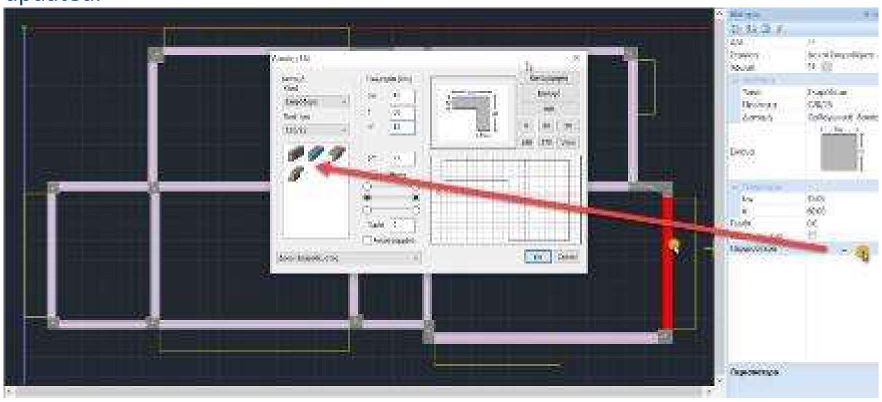
When we make changes to the Physical Model of an existing element, these of course affect its Mathematical Model. A change in the dimensions of a cross-section automatically updates its inertial and therefore its Mathematical Model.
However, changes that decisively alter the type of cross-section cannot be made when the Mathematical Model of the cross-section is already in place. These must first be carried out in the Physical Model and then the Mathematical Model must be calculated.

In our example we chose to create the Mathematical Model during the automatic recognition of the cross-sections



Therefore all the building blocks we created include both their Physical and Mathematical models.

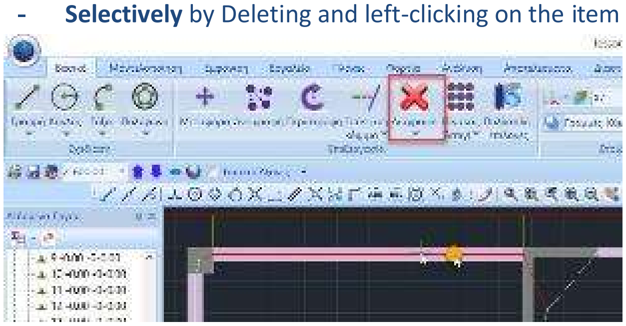
To change the dimensions of a cross-section or its type, it is enough to select it and modify its properties through the More field in Properties, and both its Physical and Mathematical models will be automatically updated.



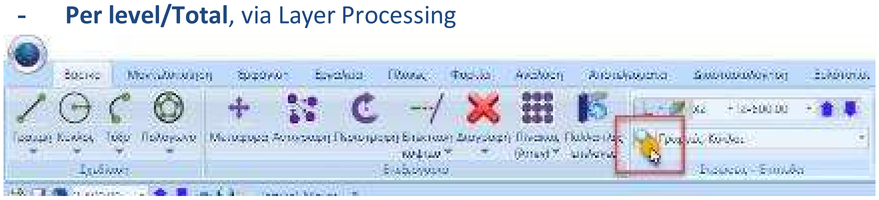
But when we are going to make changes in the category of the cross-section e.g. Convert Beam to Columns , in its connectivity e.g. Beam on Beam, Beam Partitioning etc. then the existence of the Mathematical Model makes this change impossible. These changes will have to be made at the Physical Model level and then we will calculate the Mathematical Model. If the Mathematical Model already exists, then it will first have to be deleted, the Physical Model will remain, the changes will be made and then recalculated.

Deletion of the Mathematical Model can be done selectively, per level or in total.

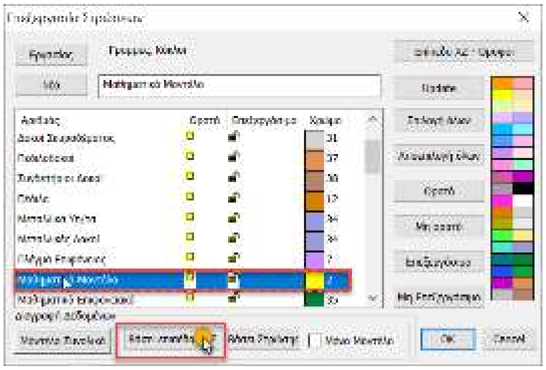
- **Selectively** by Deleting and left-clicking on the item member.



- **Per level/Total**, via Layer Processing



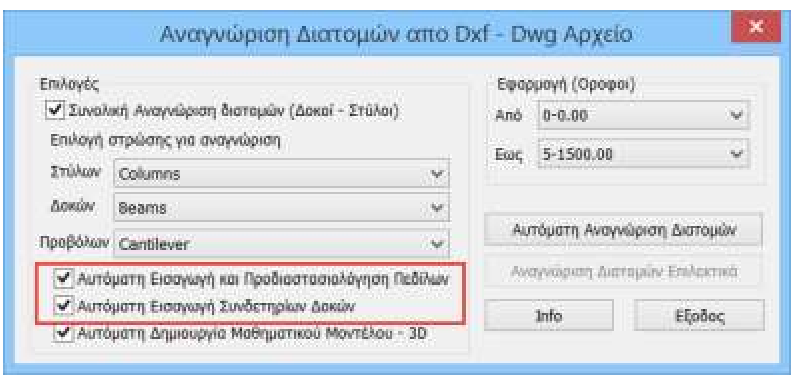
The "Delete data" field allows deleting the mathematical model of the study or part of it.
By level, by selecting the Mathematical Model layer and clicking on the command Based on Level XZ



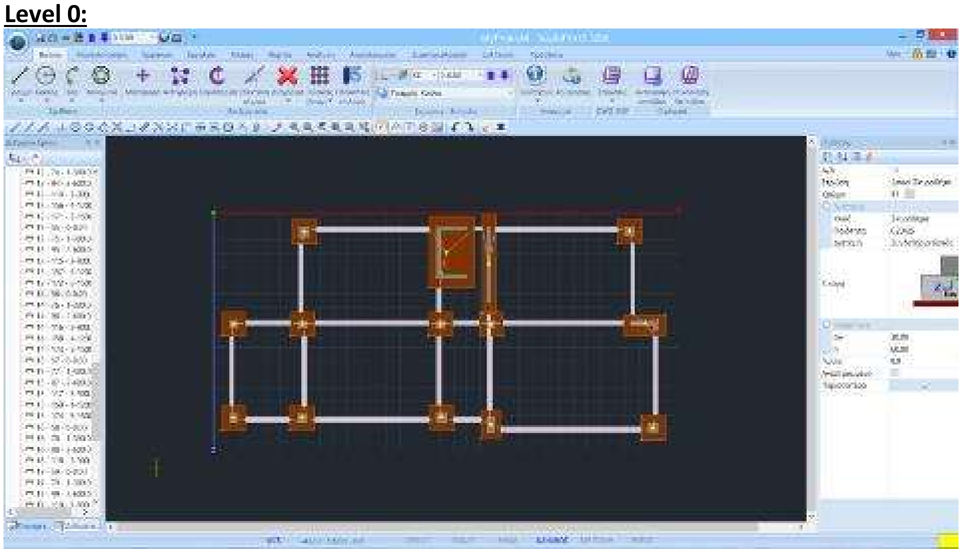
Overall, by selecting the Model Overall command only.

2.5 Automatic insertion of Pedestals and Connecting Beams at the foundation level:

During the **Automatic Section Recognition from dwg-dxf File** the user has the possibility to select the simultaneous **Automatic Import and Pre-dimensioning of Fields** and the **Automatic Import of Connecting Beams**.

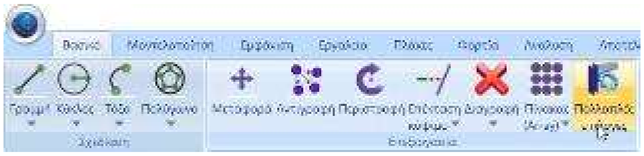


Level 0:



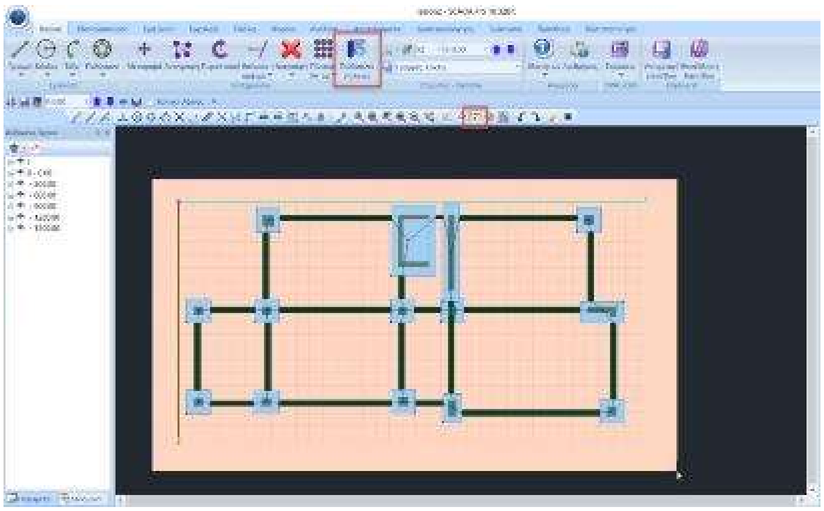
EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

The dimensions of the pedestals are obtained from a specification taking into account only the weights and are entered as packed (with Ks=0). The user is invited to select all the skids and through the Multiple Options to set the value of Ks according to the terrain.

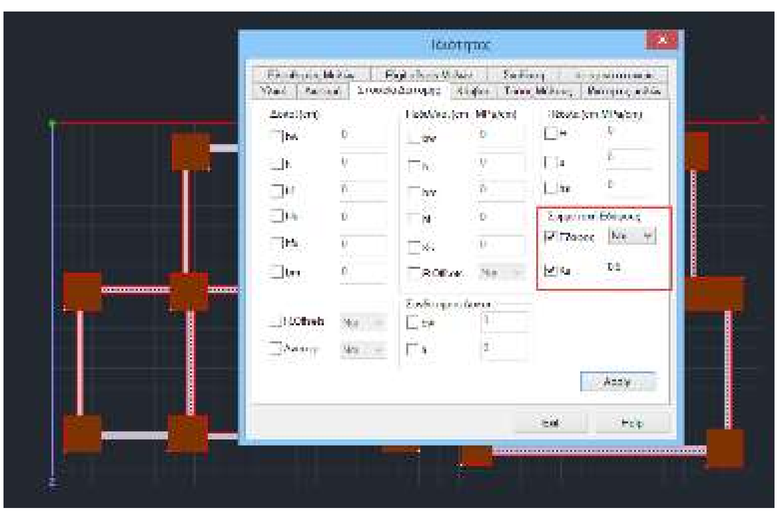


You select the command and with Select with Window

Select all level 0 and right click to open the dialog box.

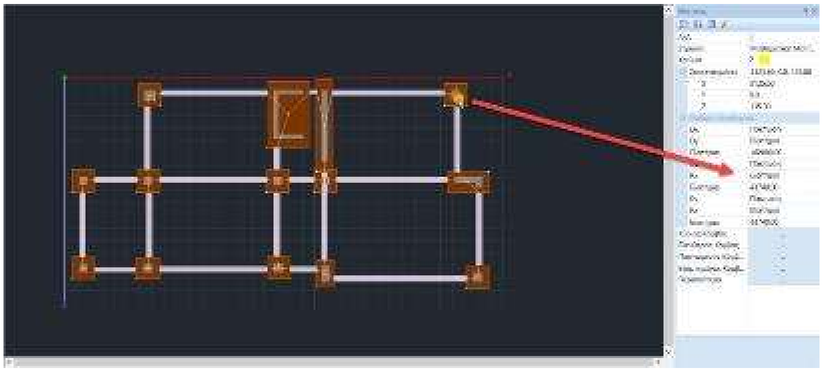


You select the Cross-section Elements and the Soil Involvement field. Set the participation and value and select Apply and Exit.



Left-click to select a node of a pedestal and check in the properties that the change has been made.

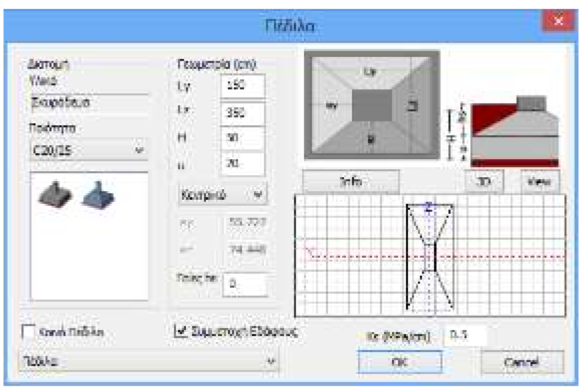
EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'



⚠ *For the sake of completeness, the example also shows the manual way of inserting pedestals and connecting beams.*

2.5.1 Sandals

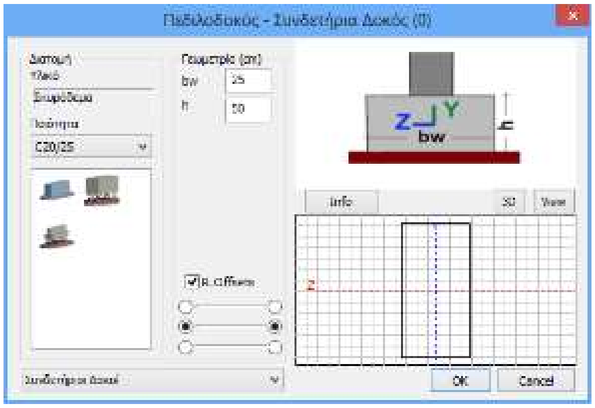
From the "**Modelling**">"**Foundation**" section select "Petal"> "Cone":



In the dialog box set the characteristics of the material and the geometry of the pedestal. Click on "**OK**" and place the skirt on the desktop by left clicking on one of the sides of the superstructure column at level 0.
Repeat the process to insert the remaining sandals.

2.5.2 Connecting beams

Select "Pedilodokos"> "Connectors":



In the dialog box set the characteristics of the material and geometry of the beam and the insertion tolerance(*) .
Click "**OK**" and place the beam at level 0 by left clicking on the start and end points. Repeat the process to place all the connecting beams.

(*)*When inserting a member on the , you can change the insertion pass at the beginning and end with the TAB key and the insertion angle of the members with SHIFT.*

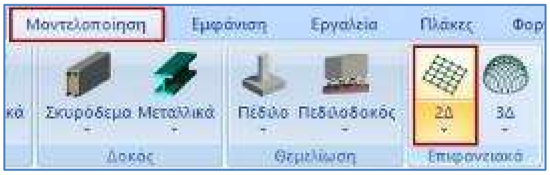
2.6 How to insert a paving:

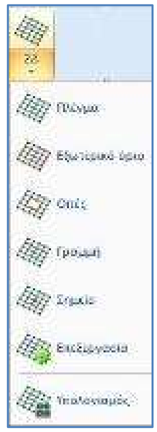
⚠ *Also for completeness, in the example a part of the foundation will be replaced with a paving in order to analyse all the already existing foundations.*

To model a pavement use the 2D surface elements (if you have purchased the 3D elements then use them).

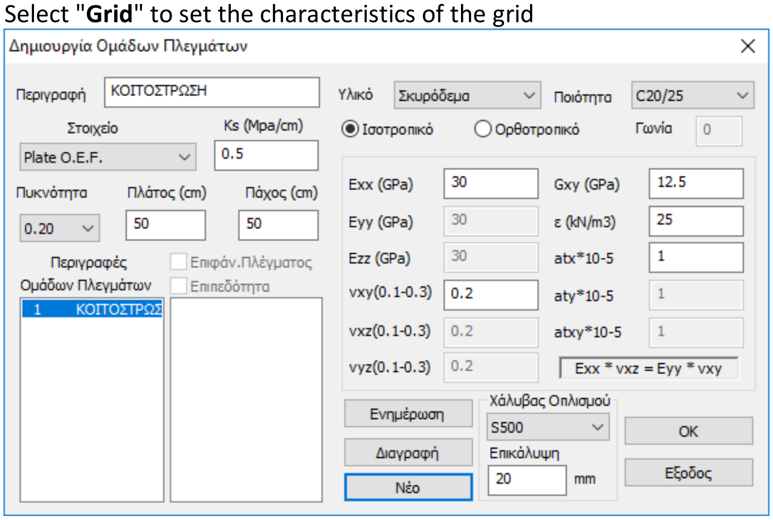
OBSERVATION:
First of all delete the Mathematical model from the foundation level and the footings that existed before the automatic import.

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'





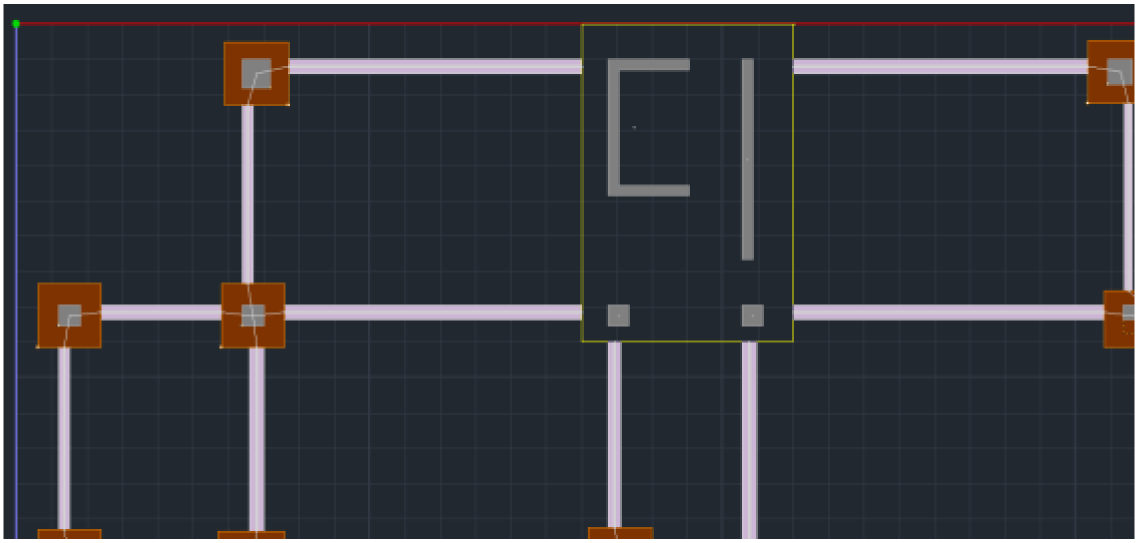
Select "**Grid**" to set the characteristics of the grid



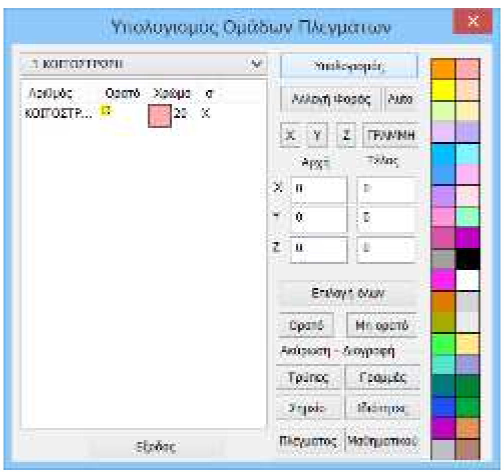
In the dialog box, give a description, the material and quality, the type of element (Plate O.E.F.), the value of the Ks constant, its density and dimensions (the Width refers to the grid and the Thickness to the thickness of the pavement), as well as the quality of the reinforcing steel and the coating. Click on "**New**" and then "**OK**".

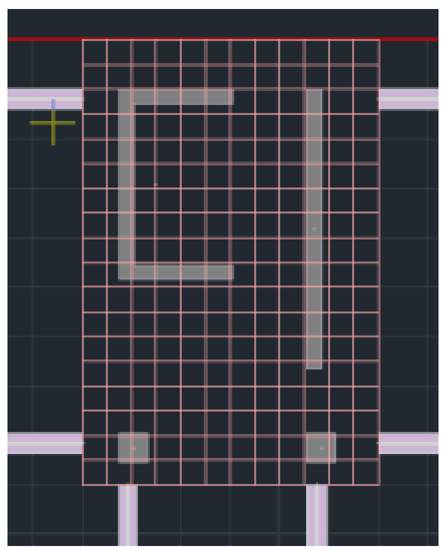
Then select "**External Limit**".

Set the perimeter of the pavement by left-clicking on the corners of the perimeter. Finish by right-clicking to define a closed perimeter.



Finally, select "Calculate".
In the dialog box, select the grid so that it turns blue, then click on "Calculate". The grid is automatically created. Click on "Exit" and the grid will be created.





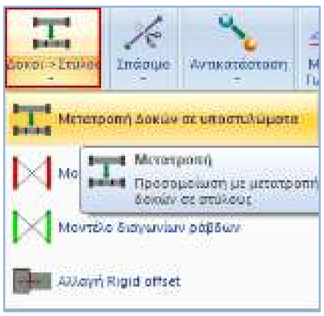
⚠ *Leave the grid for now and continue with the insertion of the remaining elements of the foundation. After you have finished importing the physical model elements, you will create the corresponding mathematical model.*

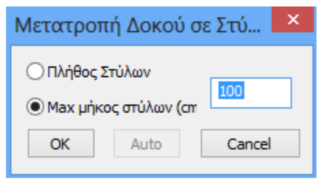
2.7 How to simulate the basement walls:

There are several methods for simulating the walls of the basement. In this example, the method "Beams in columns" was used.

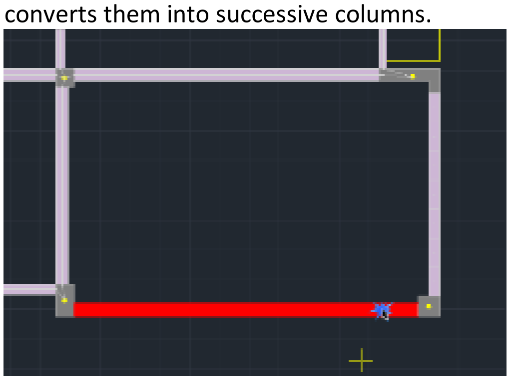
A basic prerequisite for substantial modifications to the physical model is the deletion of the mathematical model.

For the 1° level (basement roof), from the "Toolssection>> "**Model**" select the command "**Convert beams to columns**". In the dialog box select one of two options and enter the appropriate number.

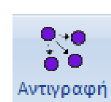




First you draw the mathematical model of the beams that will be converted into basement walls. Then you select the command and left click on the level 1 beams to be modified and automatically the program converts them into successive columns.


Level 1:

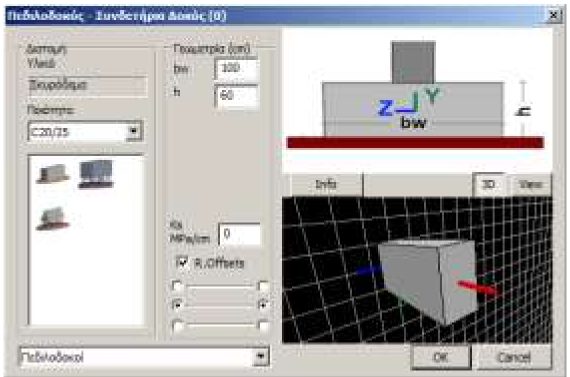
You can repeat the same procedure on the foundation level or copy these new columns of level 1th to level 0 using "Copy" command.

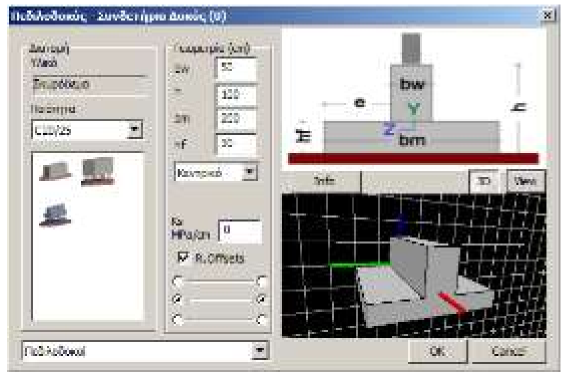


First delete the corresponding connecting beams from level 0. Then, call the command and s… objects you want to copy. The selection can be done either individually, by window, by polygon, etc. You then press the right mouse button to indicate the end of the selection and point to a feature point (line end, pole top, beam end, etc.). Go to level 0 and set the corresponding point for copying the objects.
The modelling is completed by creating the mathematical model and connecting the nodes of the columns with rigid rods.

2.8 How to insert footings under basement walls:

From the "**Modeling**">"**Foundation**" section select "Pedestal> "Rectangular or Tau"

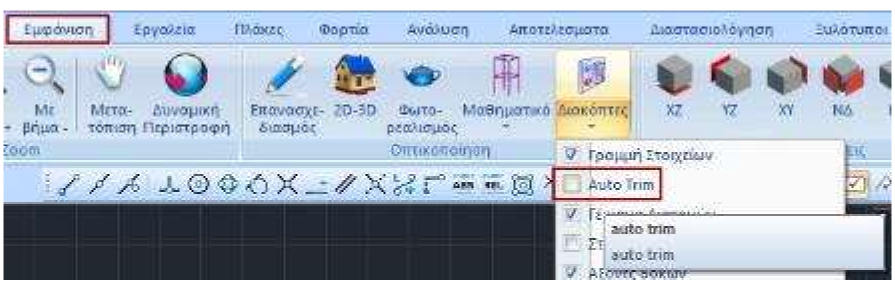




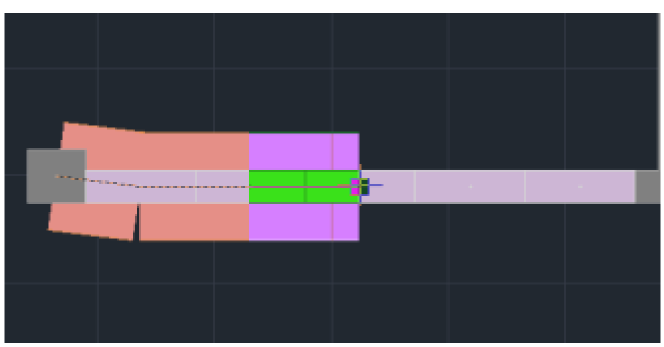
In the dialog box set the characteristics of the material and geometry of the footbridge.

To insert footings under the basement walls, first of all, you need to turn off:

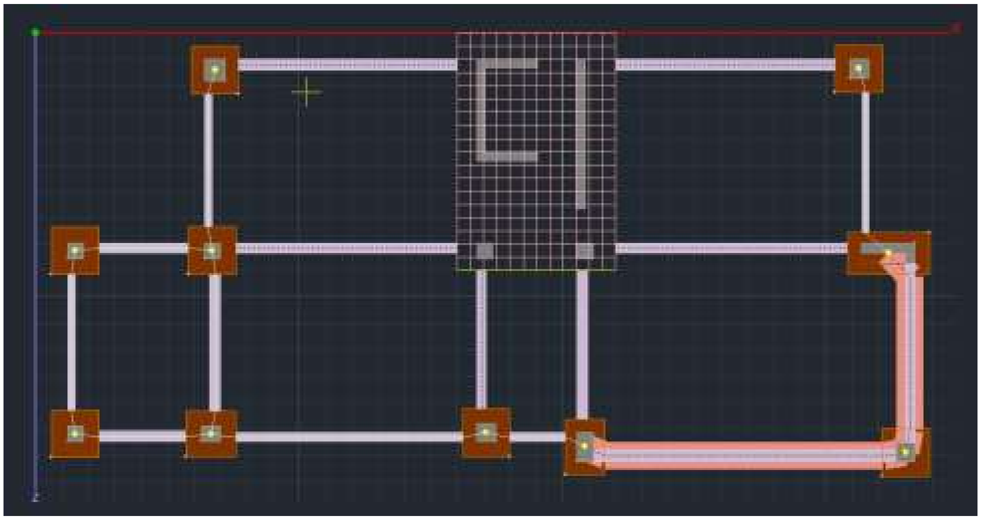
- "R.Offsets" (inside the dialog box) and
- "Autotrim  (within the Appearance>Diaplays section)



Then insert the pedestals under the basement walls, with the help of the pulls, from centre to centre.



Complete the input of all foundation data, as described above, until you have created the floor plan of the example:

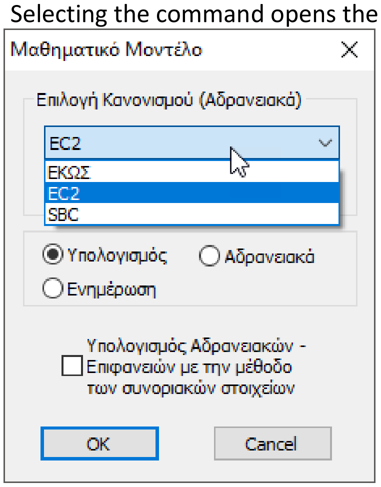


2.9 Creating a mathematical model:

After you have completed the modifications of the physical model of the study (copies, deletions) and the import of the additional data, you proceed to the creation of the Mathematical Model of the study.

With the command "Calculate" , the program calculates and produces the mathematical model of the study (nodes and bars). That is, an automatic simulation of the physical model (structural elements: columns, beams, etc.) is performed with linear members connected by nodes.

Selecting the command opens the dialog box:

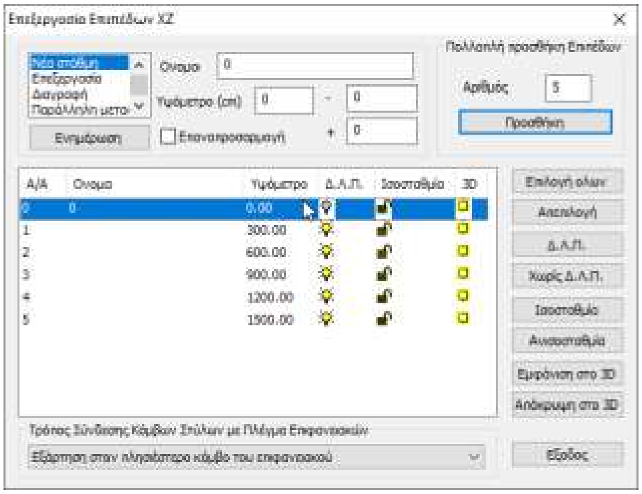


If after creating the mathematical model you decide to change the regulation, to update the elasticity measure, select regulation and "Convert regulation".

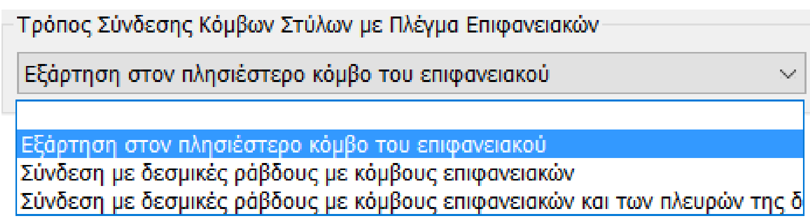
SCADA Pro allows the collaboration of linear and surface elements in the same interface. The connection of linear elements to a corresponding surface element node is done automatically with the following command.

Connection of Column Nodes with Surface Grid

SCADA Pro allows the collaboration of linear and surface elements in the same interface. The need for binding between them is therefore born.

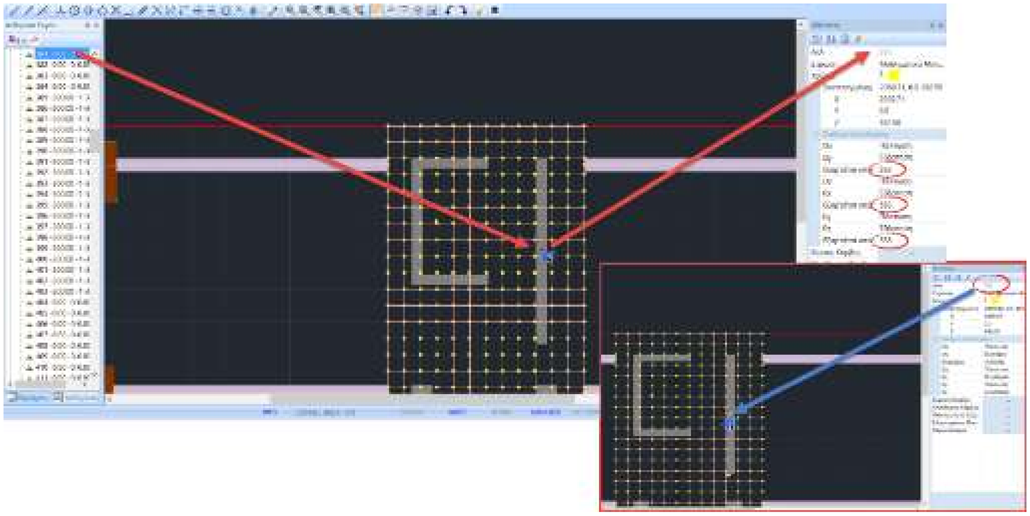


At the bottom of the window there is the choice of the way of connecting the nodes of the columns to the surface grid, for the selected level, by selecting one of the three ways, connecting the nodes either by simple dependency or by connecting through tie rods.



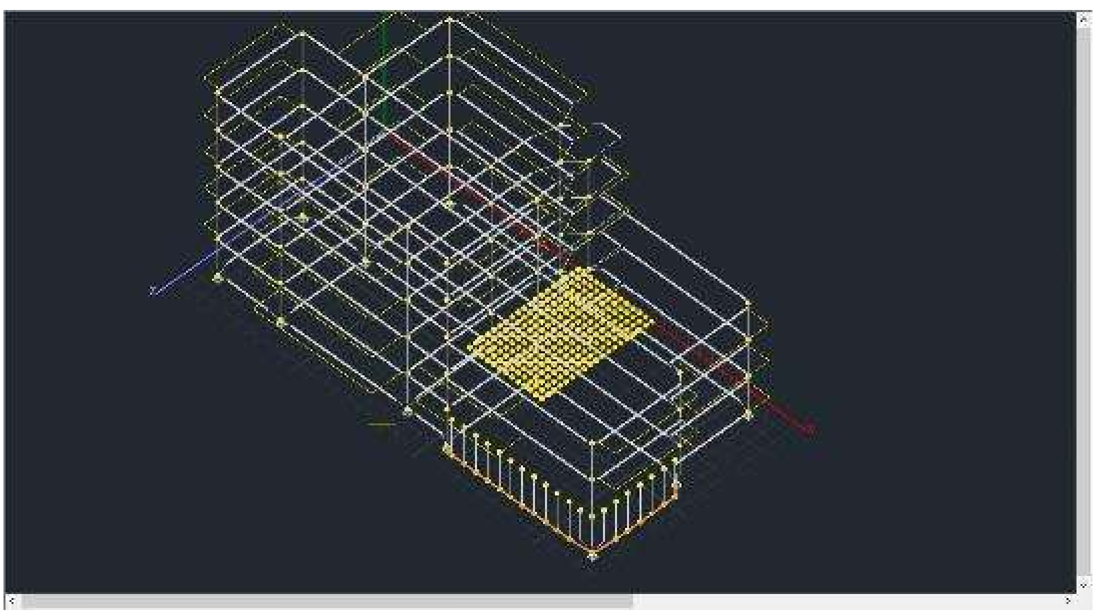
⚠ *Select the node of the column inside the pavement (node 381) and observe its automatic dependence on the nearest surface node (node 568).*

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'



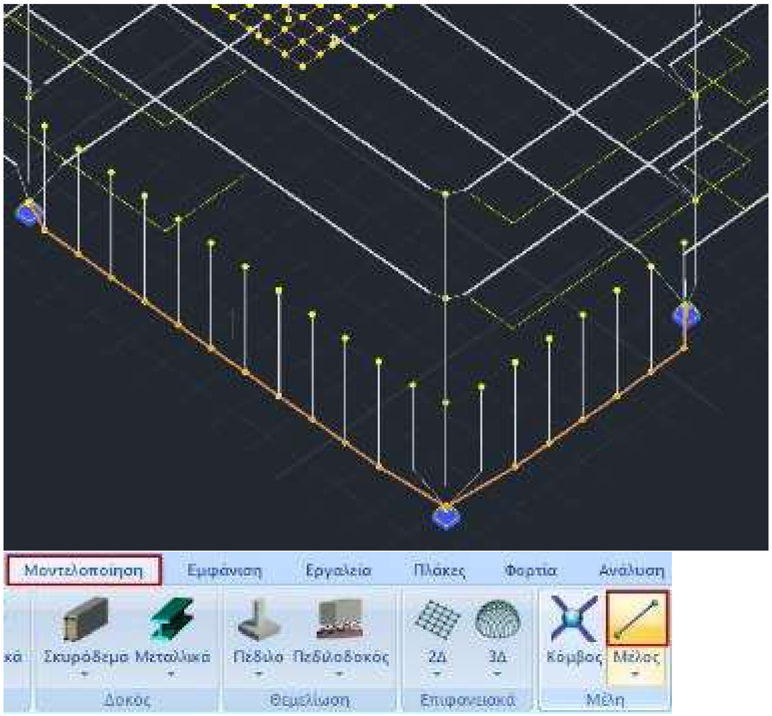
2.10 Three-dimensional imaging:

After the creation of the mathematical model, it is possible to visualize it in a photorealistic way, while it is possible to make modifications concerning the mathematical members and the nodes, within the 3D visualization environment . 
For example, can create a slope or an anisostropy, and also insert mathematical members to connect the unconnected nodes of the basement walls.

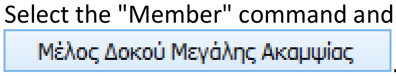


2.11 Connection of basement wall nodes - High stiffness beam member:

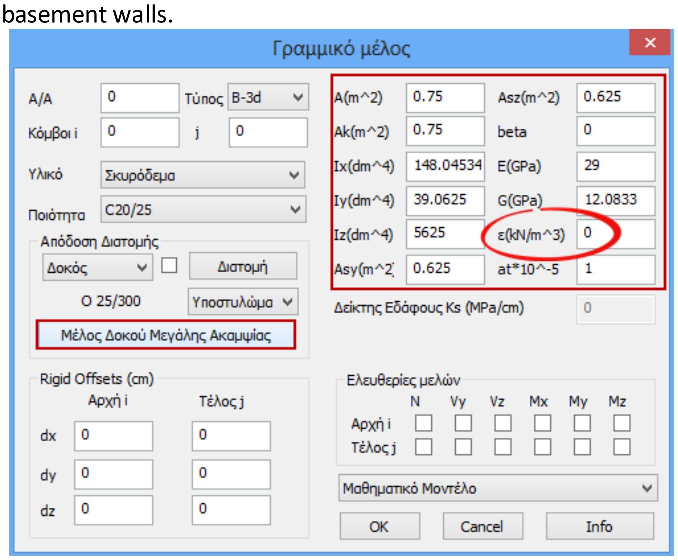
The simulation of the basement walls through the "Beams in columns" command is completed with the connection of the column nodes at level 1 (at level 0 the connection has already been made by inserting the footings).



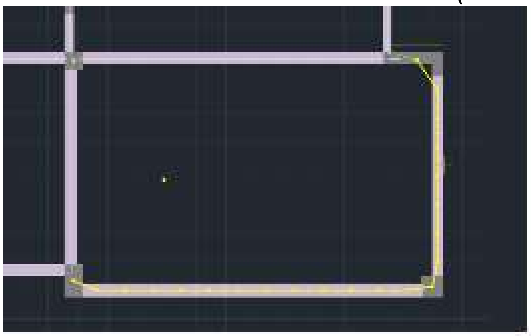
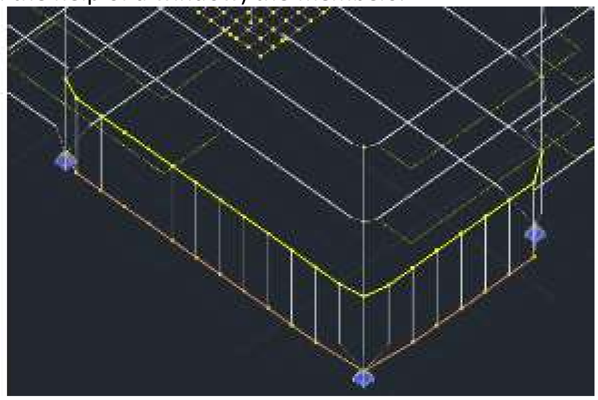
Select the "Member" command and in the dialog box the button



. The field is automatically populated with the inertial elements of a high stiffness and zero specific weight beam that simulates the high stiffness element required to connect the columns for the basement walls.

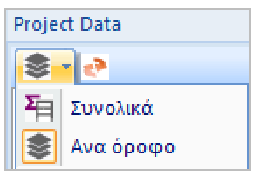


Select "OK" and enter from node to node (or with the help of a window) the members:

2.12 How to create a slope:

To create a gradient in a simple way, take advantage of the 3D view of the model by selecting the command .

Also, on the left 

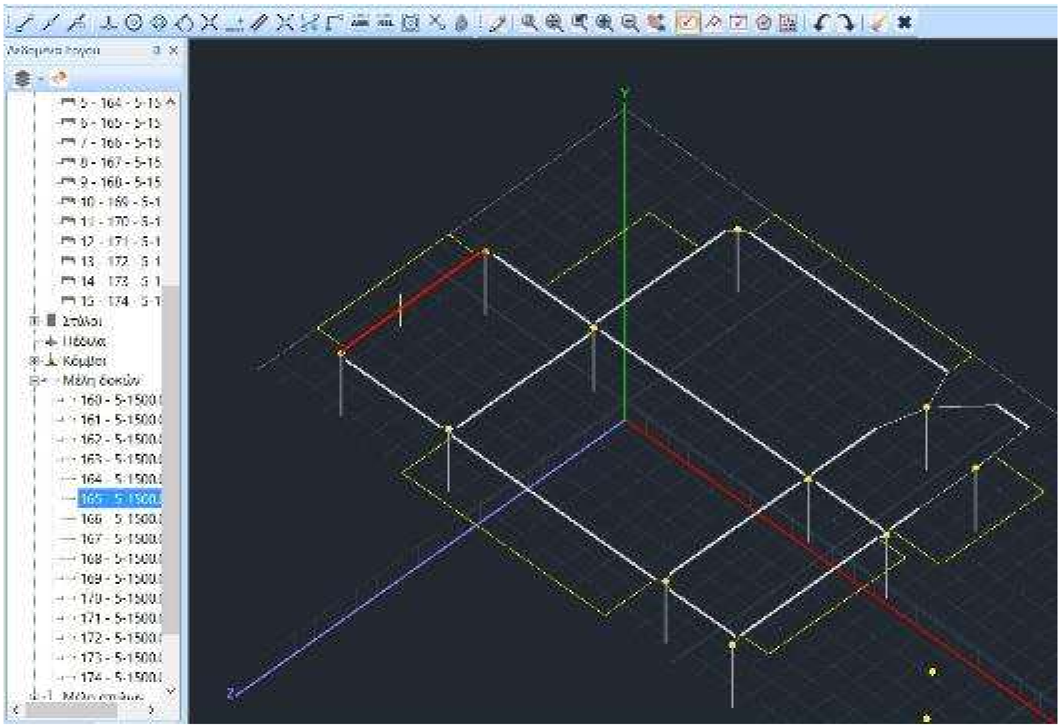
select the display "By Floor"

And open the Level 3 group and the Beam Members subgroup. Select the

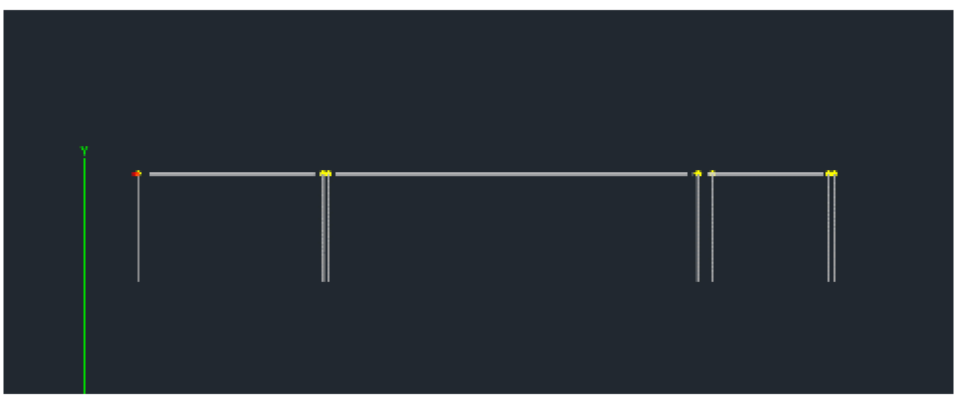
member whose slope you want to modify:

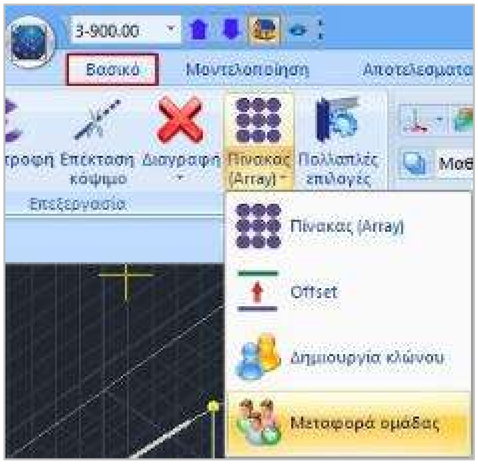
The member is coloured red while level 5 is isolated to facilitate detection and graphical modification.

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'



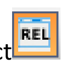
From the "Appearance" section, display the "XY" view:

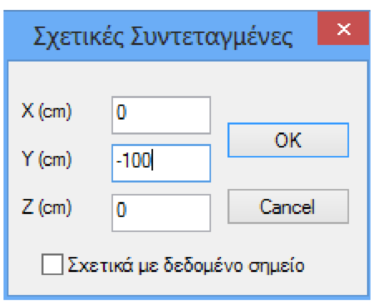




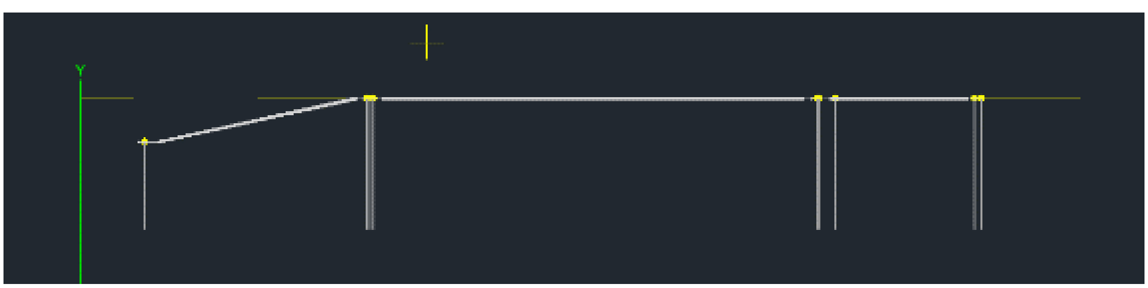
From the "Basic" section select the "Group Transfer" command, activate the "With window" option and enclose the node and right click.

In this way you also select all nodes behind the selected node in the XY layer.

Left click on the node and select  .

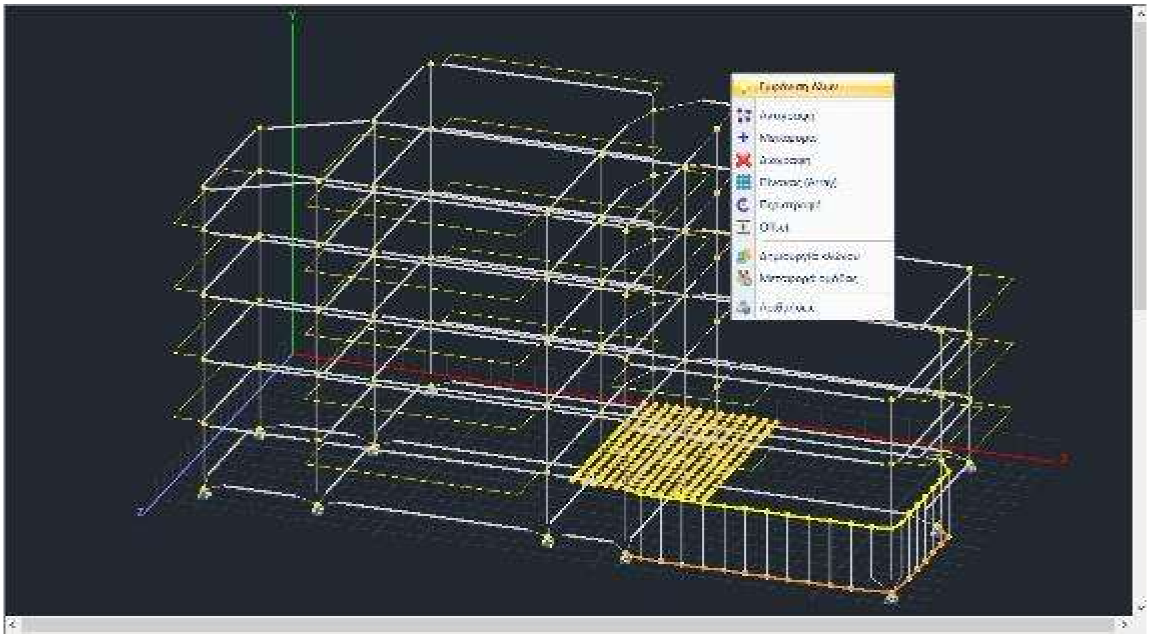


In the "**Relative Coordinates**" window, write in cm the relative offset and click "**OK**". Automatically the nodes are lowered and the slope is created.

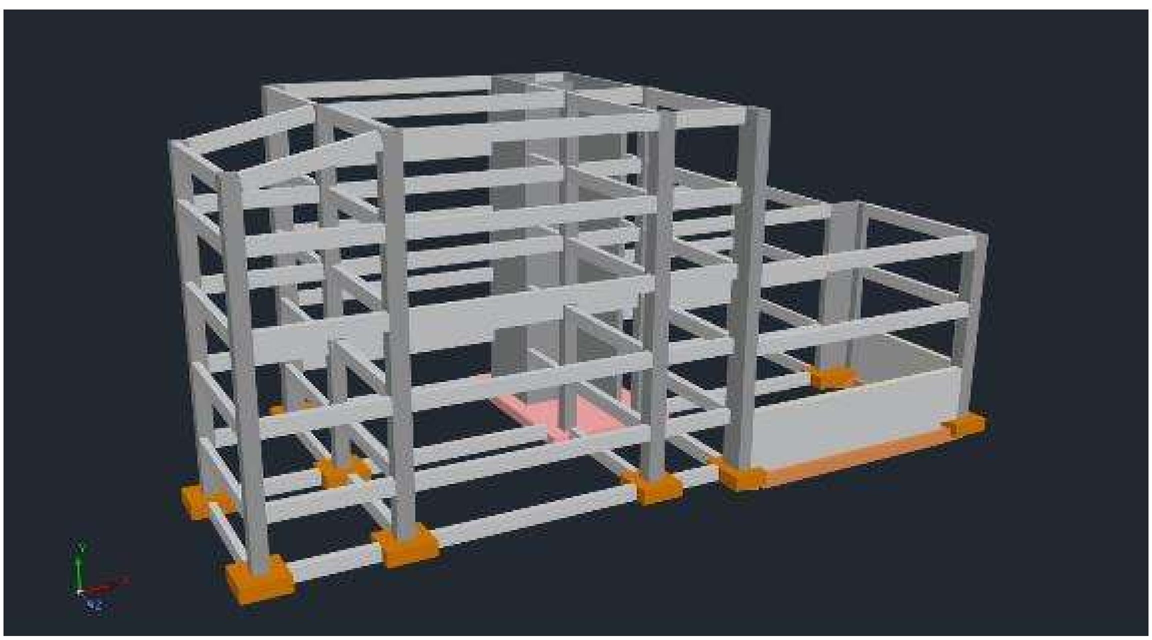


Right click on the desktop and "Show All" to display the entire vector again.

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

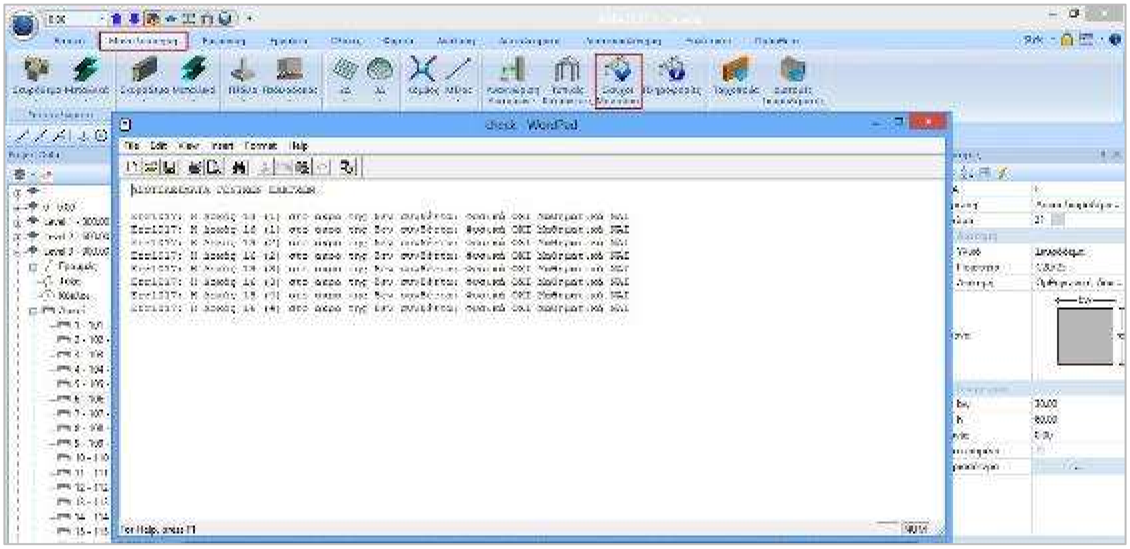


Open the photorealism to see the final form of your vector:



Select "Checks" to identify possible errors:

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'



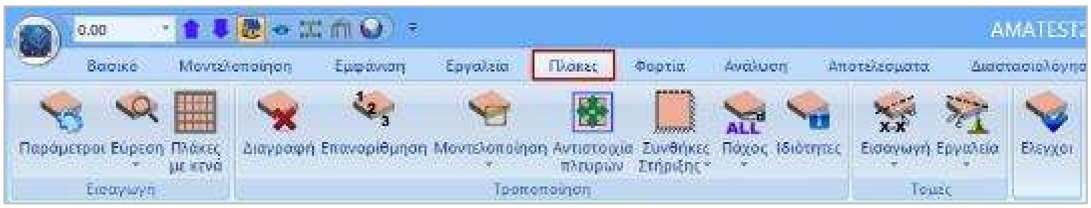
Possible errors and warnings appear in the GENERAL AUDIT RESULTS.

⚠ *Error1017 in the example is the warning about the existence of a beam on beam and is not an error. If errors occur in the model, correct them using program tools before proceeding to the next stages of the study.*

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

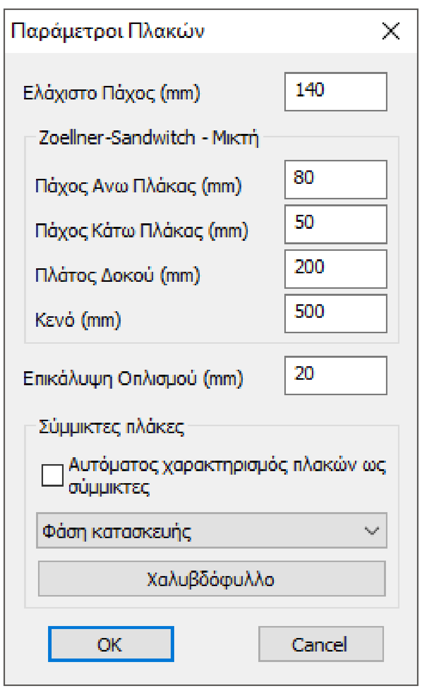
PLANS

The data input option via automation AUTOCAD also provides for the automatic input of the plates. However, if you delete the mathematical model in order to make the required modifications, then the corresponding plates will also be deleted.
To re-insert deleted or new plates you can use the commands in the "Plates" section.



3.1 How to insert solid slabs:

To insert the plates, bring the model to plan view and select, with the arrows, one layer at a time. From the "Insert" field, select "Parameters", and enter in mm the values of the minimum thickness and the reinforcement overlap.



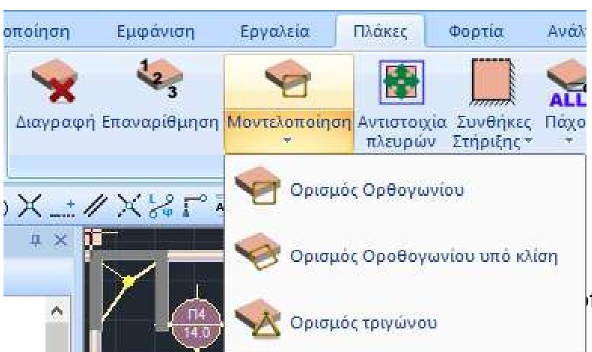
With the command "**Find**">"**Total**" the program automatically detects all closed contours present on the level and automatically enters all the slabs of the floor.

Each slab is displayed with a symbol  indicating the number of the slab, its thickness resulting from the greater of the minimum thickness you have specified in the parameters and the thickness resulting from the looseness check, as well as the support conditions,

The symbols for the support conditions in the plate symbol are :

- Thin line with vertical lines which means continuity of plates, i.e. foothold.
- Thick line meaning no continuity i.e. articulation.
- Dotted line meaning free end.

In the case of a randomly shaped plate, notice that in the plate symbol a question mark "?" appears in place of the "P" on the plate. To solve it you need to model it, i.e. define another plate, rectangular, or rectangular with a slope, or triangular, equivalent to the first one.



Select the "**Modeling**" command and one of the three definitions.

Left-click inside the random shape plate to select it and set the equivalent plate:

f the diagonal, hold down the mouse button, and

- If it is Rectangular sloped: select one side of the slab for parallelism. Click on the first vertex of the diagonal, hold down the mouse button, and click again on the second vertex of the diagonal.
- If it is Triangular: Left-click on the three corners of the triangle.

Finally, you must define the correspondences of the sides of the equivalent plate with those of the real plate. This procedure is used to assign the members of the physical model of the plates to the sides of the mathematician to be solved.

Select "**Match model sides**" and the plate. The rectangle or triangle that appears is the mathematical model of the equivalent plate.

You select the side of the mathematical model of the plate (a coloured dot appears on it), and then the physical members you want to assign to this side of the mathematical model of the plate (a dot of the same colour appears). Complete the mapping to one side by right-clicking, and continue the process for the remaining sides of the mathematical model. Finally, you map each vertex of the equivalent rectangle (which is denoted by a triangle) to points in the physical model so that the lengths of the sides of the mathematical model are also reduced to the physical ones, so that the loads of the equivalent plate will be distributed to the actual lengths of the physical members. The matching is done by first selecting the vertex of the mathematical model and then pointing the mouse to its new position. The process is repeated for the remaining 3 vertices of the mathematical model without cancelling with the right mouse button.

command displays the results of the general plate checks. If there are any errors, correct the plates before proceeding. Repeat the command after the slab loads have been assigned to ensure that they were assigned correctly.

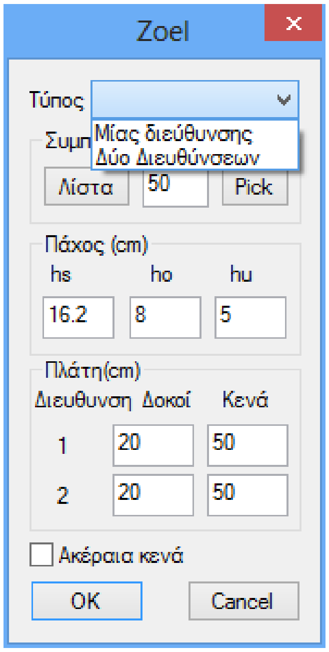
3.2 How to create a slab with gaps:

From the "**Find**" field select "**Plates with Gaps**" and click on the plate.

In the dialog box, select the type and write the width of the compact zone. Click on "*Pick*". To place a compact band, first select one side (the beam to which it will be parallel) and then the slab. The line that appears is the inner boundary of the compact zone. Repeat for all compact zones.

To place a solid belt of a different width, right-click and the dialog box reopens. Modify the width and continue as before until the last belt is placed. Right click to finish.
The window opens again. Fill in the remaining fields concerning the plate (*5) and the widths of the beams and gaps, and click "**OK**". (*6)

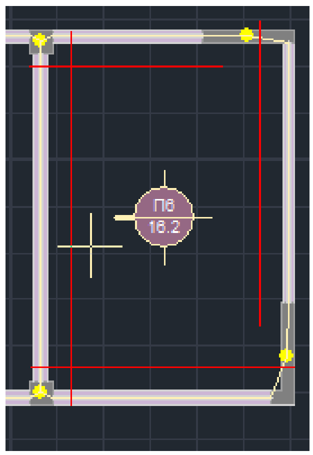
(*5) hs: total thickness of the plate
ho: thickness of the upper solid part of the plate

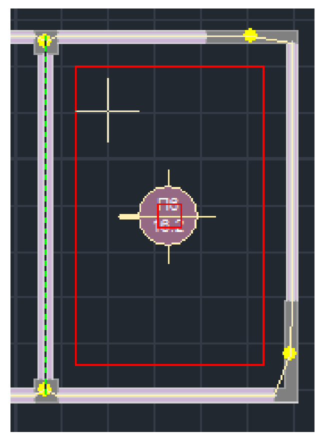


hu: thickness of the lower solid part of the plate
(hu≠0 for Sandwich plates)

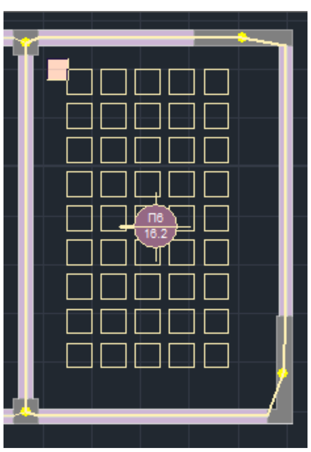
(*6)Select the checkbox next to "**Integer Gaps**" to draw integer gaps.

The mathematical model of the plate is displayed on the screen and the program asks you to set the 1th main address. Click on the side of the mathematical model that will define the 1h direction of the beam.





Two parallelograms are automatically formed, a smaller one, which is the gap with the dimensions you specified, and a larger one, which is the inner boundaries of the solid zones. Select one vertex of the gap (small par/me) and then one vertex of the large one, from which to start placing the gaps.



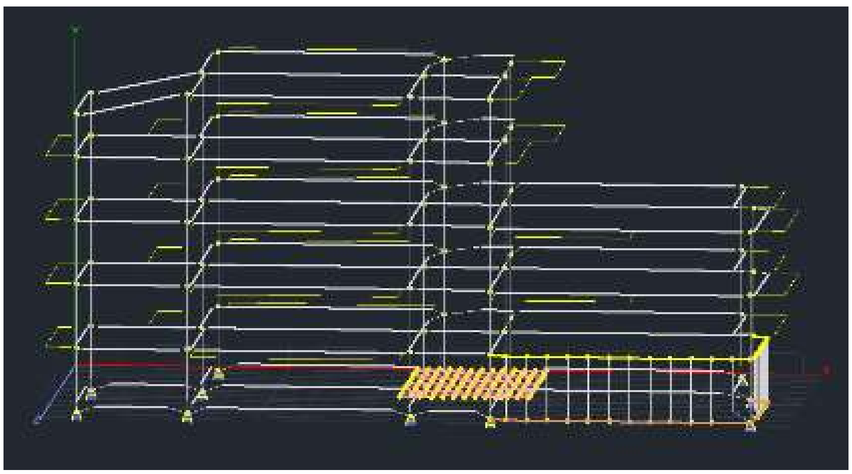
The result of this process is shown on the left.

3.3 Insertion of plate sections:

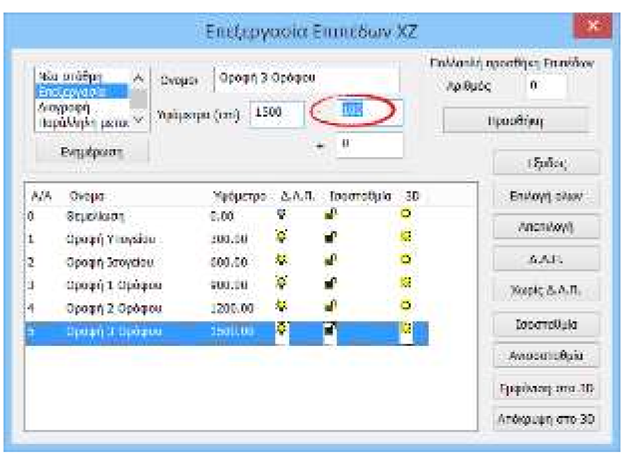
A prerequisite for the solution and sizing of the plates is the introduction of the sections. From the "Plates" field select "**Insert sections**" by **X** and/or **Z** and enter the sections by left-clicking. The direction of the section defines the direction of the main reinforcement.

3.4 In case of a sloping slab:

As in the example under consideration, in cases where we encounter a slope plate, some procedures are required for its correct simulation:

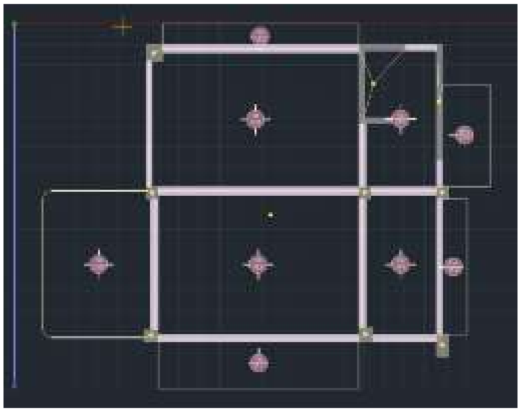
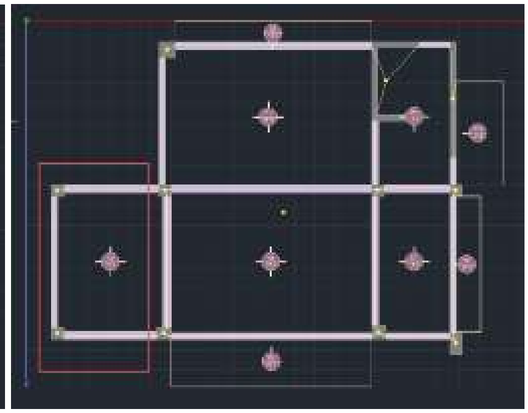


In order to be able to enter the sections of a slab, it is necessary that its static elements (beams-pillars) belong to the same level. So in the case of a sloping slab you must define the unbalance of the floor to which it belongs:

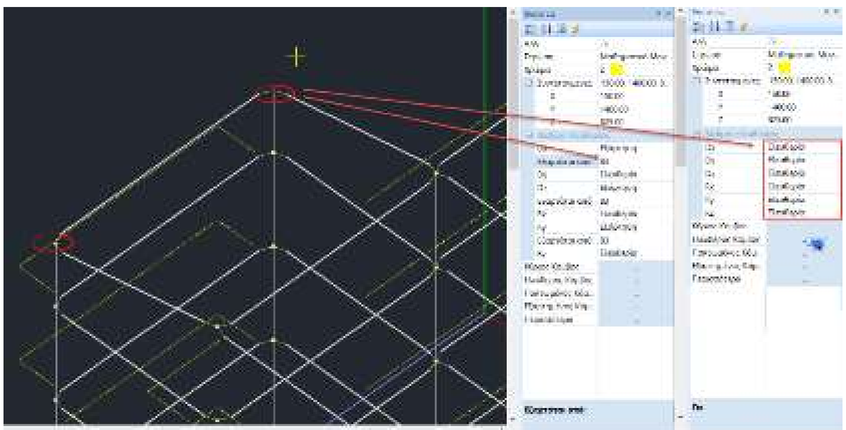


indicating the imbalance (e.g. - 100)> Update> Exit.

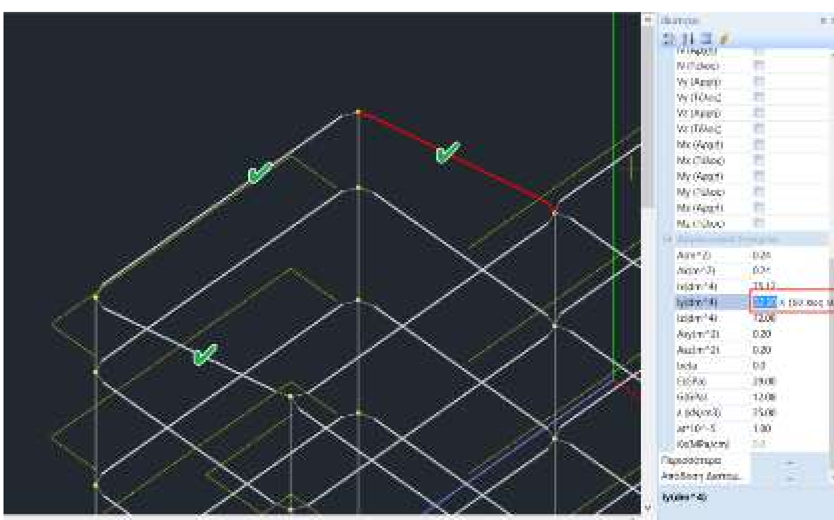
EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

The nodes of anisostasy should be excluded from the septal function.
Select one of the two nodes and release all their dependencies from the baffle node.



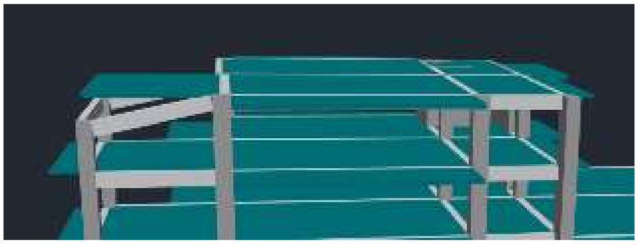
In addition, to simulate a sloping slab or roof slab diaphragm mode , you can multiply the Iy of its perimeter beams by a factor (50 to 100).



EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

⚠ **OBSERVATION:**

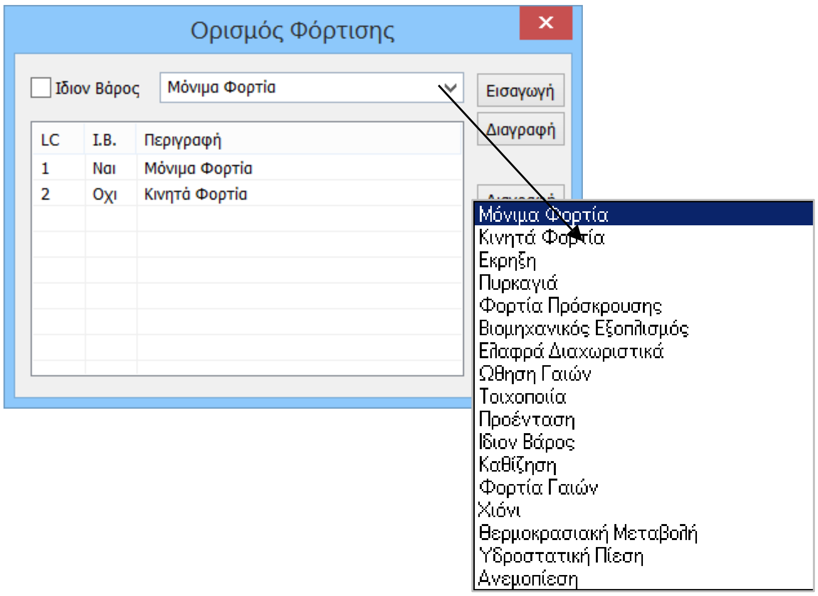
When opening the photorealism, notice that the slanted plate is not simulated as slanted by the program. In any case the slabs will be considered as horizontal. The call only applies to the beams.



SHELTERS

4.1 How to set uploads:

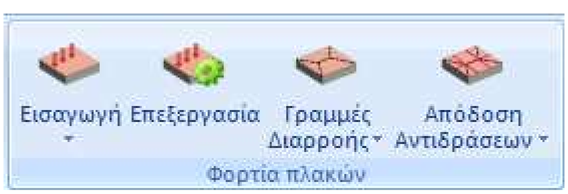
To import the loads you must first set the loadings. Open the "***Loads***" section and select the "***Loads***" command.



The dialog box includes by default the two main loadings, Permanent and Mobile. The user has the option to delete ("*Delete*") or insert other loadings by selecting from the list and clicking on "*Insert*", as well as the corresponding loadings included in them.

- Select the checkbox next to *"'Same Weight*" if the specific charge includes the same weight and the I.B. column says "*Yes*". Otherwise, it says *"No*". (Same weight to be included only in a load and as a rule in permanent loads)
- "*LC*" is Load Case".
- "*OK*" to enter and exit.

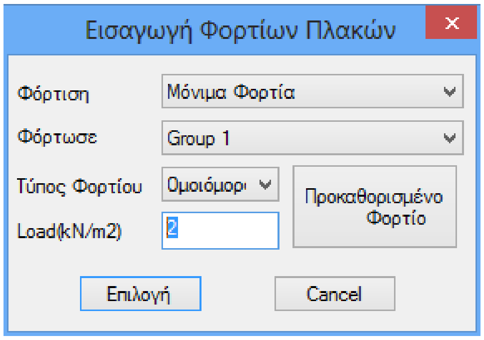
4.2 How to insert loads into the plates:

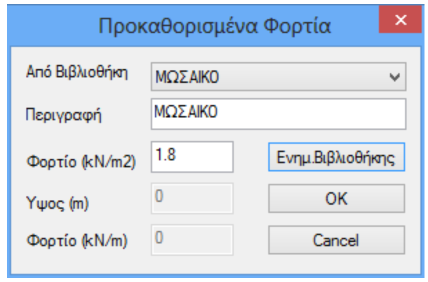


From the "**Plate loads**" field, select "**Insert**". You have the option of entering loads, either in total for all slabs of the active floor, or selectively for one slab at a time.

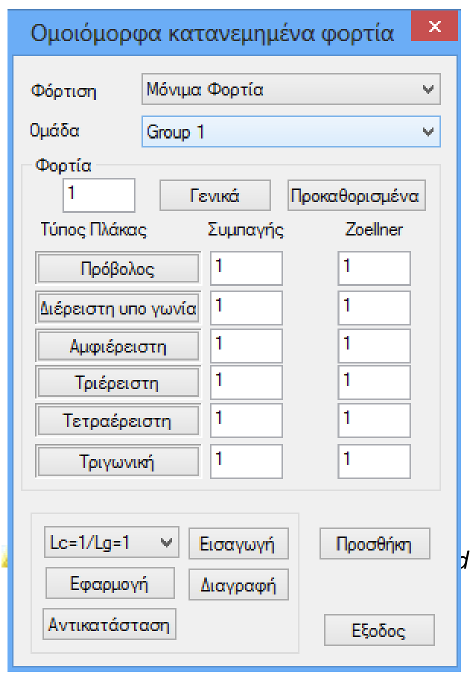
EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

In the dialog box select, load, load type and enter a value in (KN/m^2). "*Select*" and click on the plate.





The "Preset load" button includes a library of coating materials, which automatically updates the value of the imposed load. The user can update the library with new materials by setting the corresponding load values.



In the dialog box select, load, enter a value in (KN/m^2) and click on "*General*" to enter this value in the plates of all types. The "*Insert*" command creates the load but does not apply it definitively. The final application is done by clicking on the "Apply" button.

Select another charge and repeat the process.

To apply the loads you have just set to the slabs, select "*Apply*". The loads are automatically distributed evenly over the surface of the floor plates.

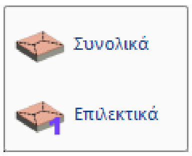
The first time you enter a load (e.g. permanent) after the "*Insert*" command you select the "*Apply*" command. Then , if you want to insert a mobile load, you define it and select "*Add*".

d 2KN/m2 mobile load was applied to all plates.

4.3 How to distribute the loads of the plates:

After inserting the loads into the plates, select:

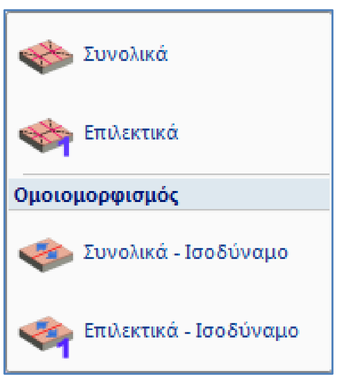
"Leakage lines": to calculate the loading surfaces resulting from geometric division of the floor plan surface, which are then used to calculate the design actions of the beams (surfaces whose loads will be applied to the beams),



The calculation is done automatically by the program according to the support conditions, either *Overall* per level, by simply selecting the command, or *Selectively*, by selecting the slabs one by one.

and

"Reaction Rendering": to render the loads of the plates as reactions to the members defining the plates. More specifically, loads from the plates are attributed to beams and nodes, based on the geometric division made previously (yield lines).

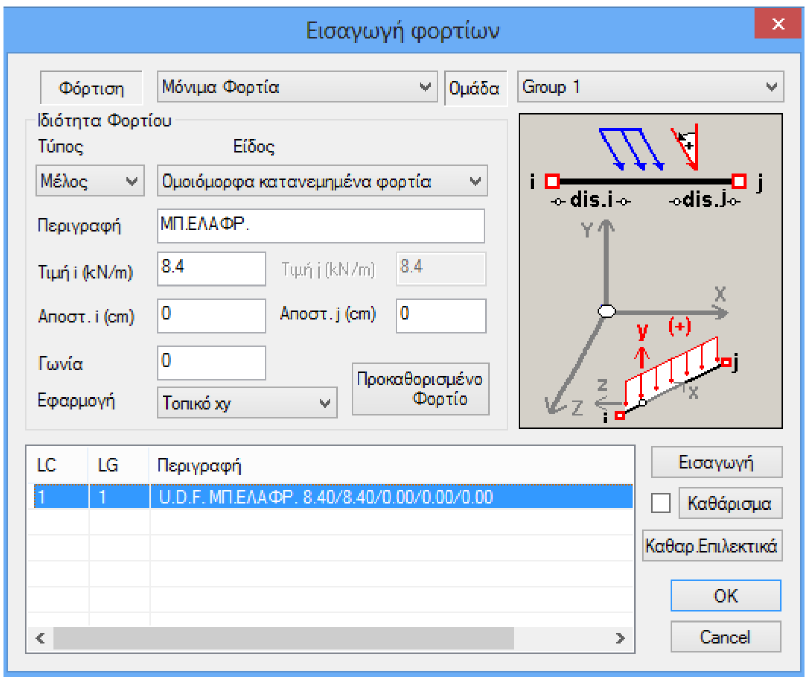


Total: attribute the loads of all plates of the active floor.
Selectively: to assign the loads of the selected plates by left-clicking inside its surface.

Homomorphism : With this option, the loads of the slabs are applied (in total or selectively, respectively) to the members, but without the geometric division of the leakage lines into rectangles and triangles, but by reducing the entire surface corresponding to the member to an equivalent rectangle.

4.4 How to import loads of members:

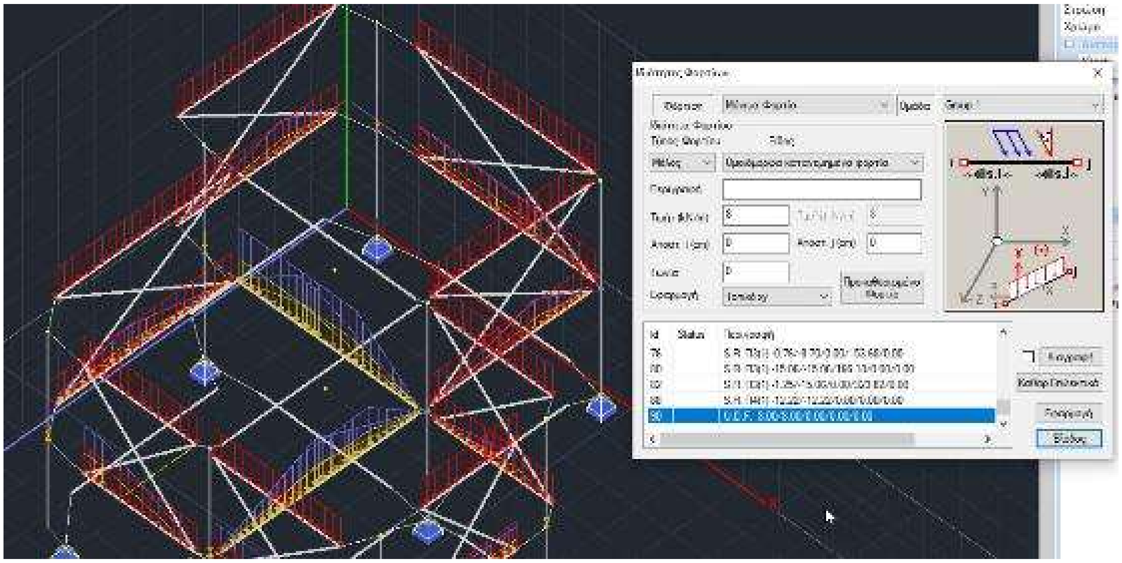
From the "**Member loads**" field select "**Insert**" and then select the elements of the member (members, nodes, finite surface) on which the loads will be applied. The selection of these elements can be done in one of the known ways  . When the selection is complete, press the right mouse button and the following dialog box appears



By selecting the "*Insert*" button, the defined load is displayed in the table with all its elements and with OK it is applied to the selected members.

OBSERVATION:

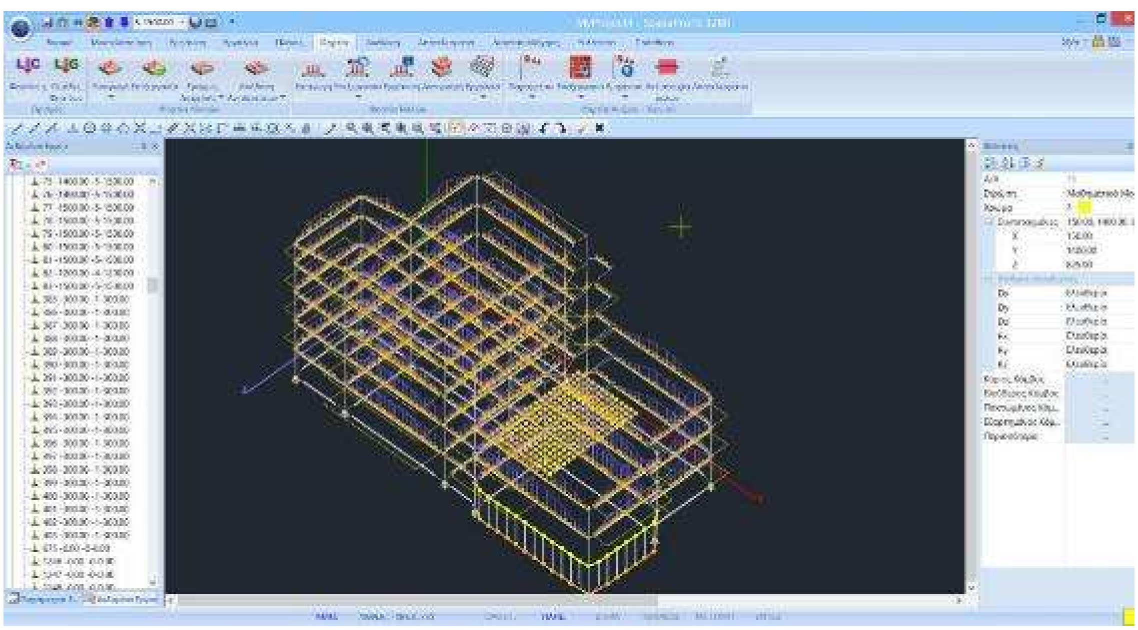
In the new version of the program, by selecting the Edit (Selective or Total) command and a load in the list, all loads belonging to the same load and having the same value are automatically reddened in the 3D representation of the vector.



In this way, you can better control the loads that have been applied to the elements of the structure and that will be affected by a possible overall modification.

⚠ With the command 

you can display all the loads on top of the study elements in 3D visualization, in total or per load and level, for visual supervision.

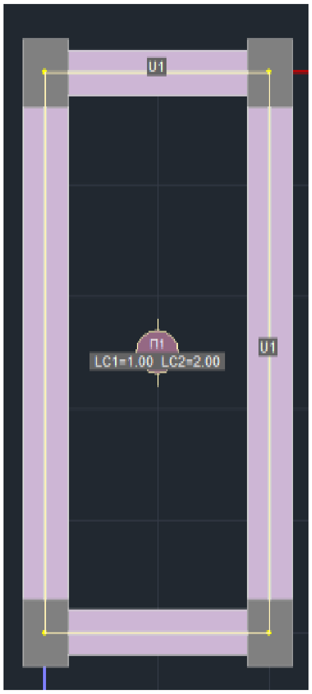


You can choose to display a vector or a number. The vector is only displayed in the 3D mathematical model. If you also check the "Value" option then values are also displayed in the graph of the loads with the vectors.

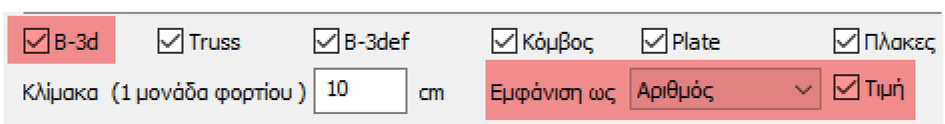
EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

In addition, by selecting 

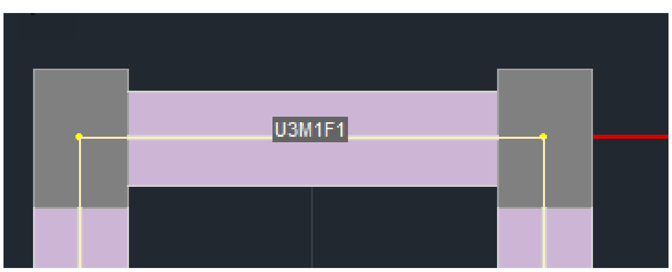
, inside plates, in the 2D visualization, the values of the plate loads are displayed.



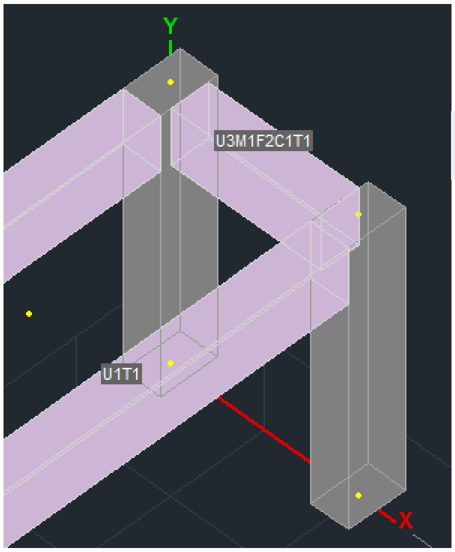
Similarly for members, with B-3d, Number and Price selected,



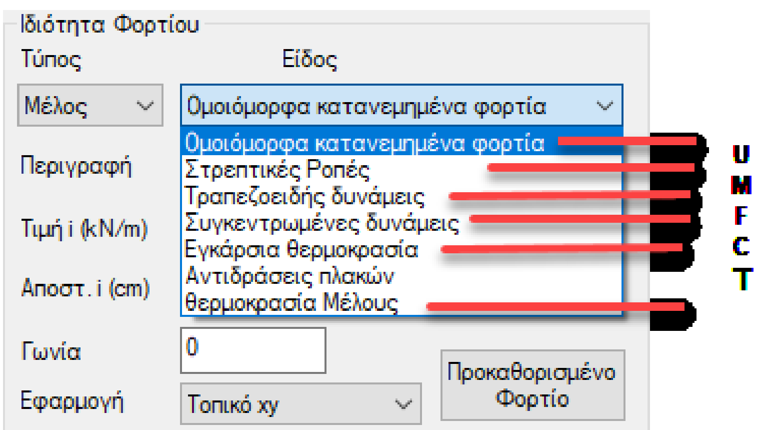
the load presence indicator is displayed on the member in letters and numbers,



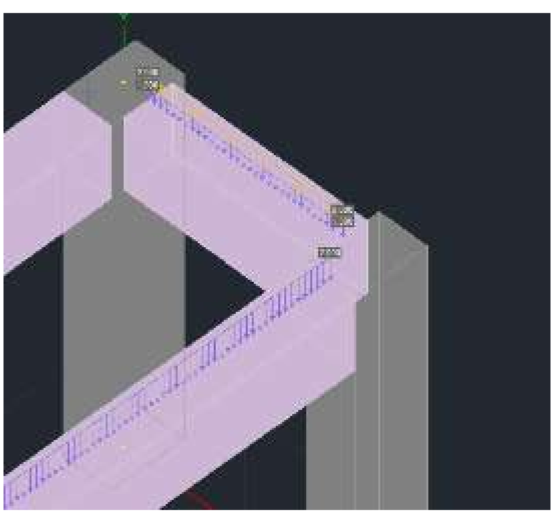
EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

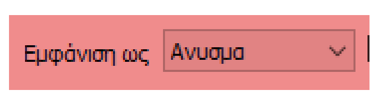


depending on the type of load (U,M,F,C,T):



And the number indicating how many shipments of that species there are.





EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

Finally, in the Filter option 

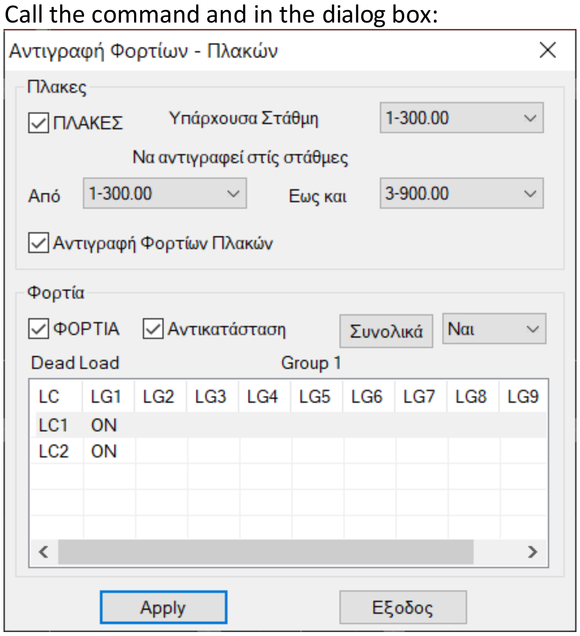
option, you can specify a range of values for the loads you want to display.

⚠ *In this example, 8.4KN/m permanent load was applied to all perimeter beams of level 1, 2, 3 and 4.*

An additional feature that can be used when you have a standard floor, i.e. when the floors are exactly the same, is *Load and Plate Copy.*

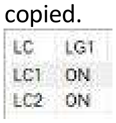
to copy plates and loads from one level to another.

Call the command and in the dialog box:



The upper part of the dialogue box is about the plates and their loads. In particular check the "PLATES" option if you want to copy the plates from one level to another. You also specify the level you want to copy ("Existing Level"), as well as the level or levels to which the copy will be made. The "Copy Plate Loads" option allows you to copy the plate loads as well.

The bottom of the dialog box is for the additional loads you have entered (masonry, linear, concentrated, etc.). Check the "Loads" option if you want the loads to be copied and select ON on the loads you want copied.



LC	LG1
LC1	ON
LC2	ON

Using the "Replace" option will replace the loads, if any, on the other floors.

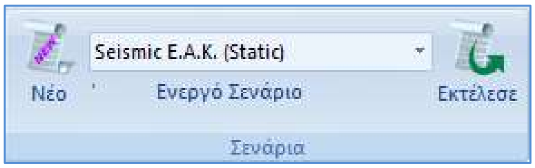
If you do not select it, the loads of the level will be added to the existing ones.

With the option "TOTAL YES OR NO" you copy the level loads in total or selectively per Group and per charge (LC).

ANALYSIS

After the completion of the modeling of the structure, the creation of the mathematical model, the insertion of the plates and the assignment of all loads to the respective members, the analysis of the design based on the regulation you will define, the creation of the load combinations and the results of the checks that will be obtained.

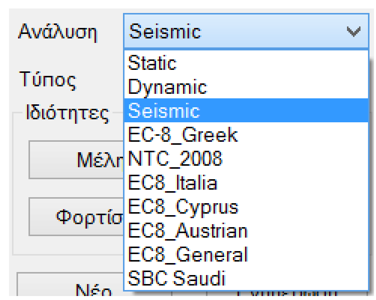
5.1 How to create an analysis script

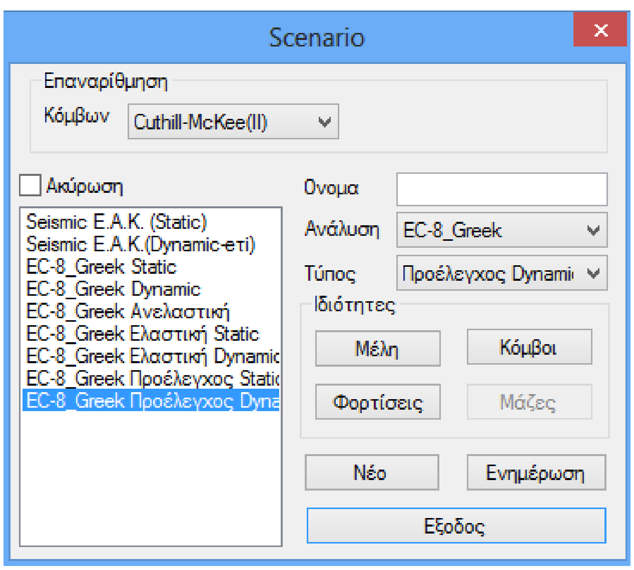


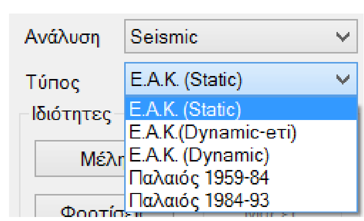
The commands of the "Scenarios" group allow the creation of the analysis scenarios (selection of regulation and analysis type) and their execution.

Select "New" to create the analysis script. In the dialog box that accompanies the selection of the New command, the possibility of creating several analysis scenarios is given, in addition to the 2 predefined ones of the current Greek regulation (Seismic EAK Static, Seismic EAK Dynamic-eti)

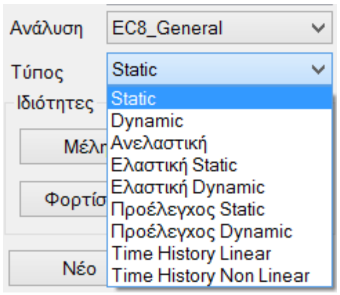
⚠ * Predefined scripts are created according to the Rules and Attachment option you make at the beginning, within the General Parameters window that opens automatically immediately after you define the file name.







EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'



Select from the "Analysis" list and the corresponding "Type" list and create 

to create a new script. Optionally, enter a name.

SCADA Pro allows you to choose between the following analysis scenarios:

For Greece:
ELASTIC - UNELASTIC

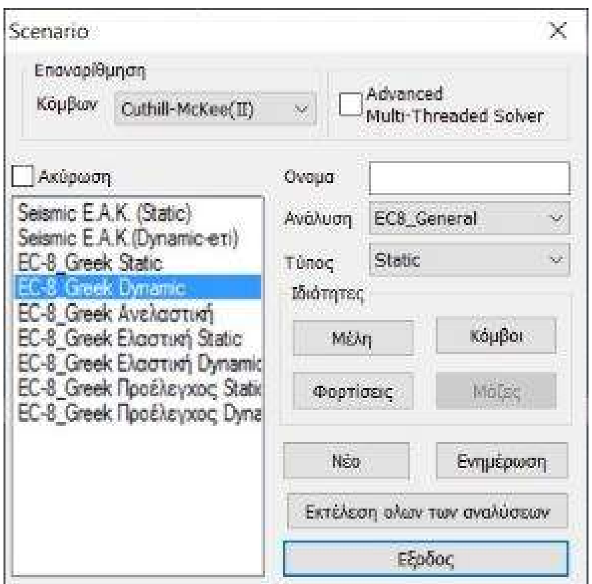
- EAK Static	Simplified spectral analysis
- EAK Dynamic-eti	Dynamic spectral analysis with homonymous torsional pairs
- EAK Dynamic	Dynamic spectral analysis with displacement of the masses
- Old 1959-84	Seismic analysis with based on the 1959 Regulation
- Old 1984-93	Seismic analysis with based on the 1984 regulation
- static	Analysis without seismic involvement actions
- EC 8 Greek static	Static analysis with based on the Eurocode 8 and the Greek Appendix
- EC8 Greek dynamic	Dynamic analysis with based on the Eurocode 8 and the Greek Appendix
- EC 8 English Pre-test Static	Pre-testing based on the CAN.EPE
- EC8 Greek Pre-Control Dynamic	Pre-testing based on the CAN.EPE
- EC 8 Greek Time History Linear	Static analysis with based on from Code 8
- EC 8 Greek Time History Non Linear	Dynamic analysis with based onfrom Code 8
- EC 8 English Elasticity	Anelastic seismic analysis based on the EDP Code 8 or the EIA Code.

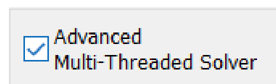
For overseas:
ELASTIC - UNELASTIC

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

- NTC 2008	Seismic analysis with based on the Italian 2008 regulation
- EC8 Italy	Seismic analysis based on the Eurocode 8 and the Italian Appendix
- EC8 Cyprus	Seismic analysis based on the Eurocode 8 and the Cyprus Appendix
- EC8 Austrian	Seismic analysis based on the Eurocode 8 and the Austrian Appendix
- EC8 General	Seismic analysis based on the Eurocode 8 without appendices (with the possibility to enter values and coefficients)
- EC 8 General Resilient	Anelastic seismic analysis based on the Eurocode 8
- SBC 301	Seismic analysis based on the code of Saudi Arabia (SBC 301)

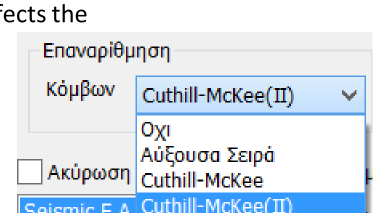
For this example, a Eurocode 8 scenario with Dynamics was chosen.





The program has now incorporated new fast analysis algorithms, using more resources, such as the graphics card, resulting in faster implementation (Parallel Processing). Activation is done through the creation of scripts.

The "Renumber Nodes" field contains a list of options. The selection affects the resolution time.

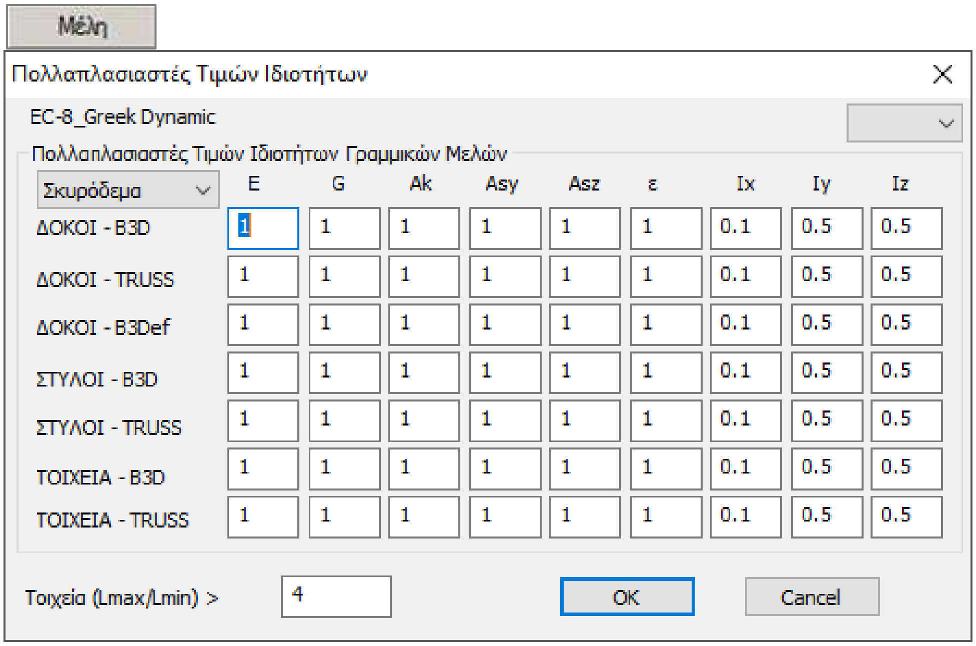


- √ The default is the option, recounting with "Cuthill-Mckee(II)".
- √ The "Cuthill-Mckee" and "Ascending Series" recounts give slower analyses, while the "No" option is not recommended.

Select the command Εξοδος to save the scripts and proceed with the analysis.

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

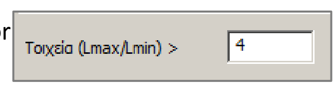
Select the script and by selecting "Members" the following dialog box appears:



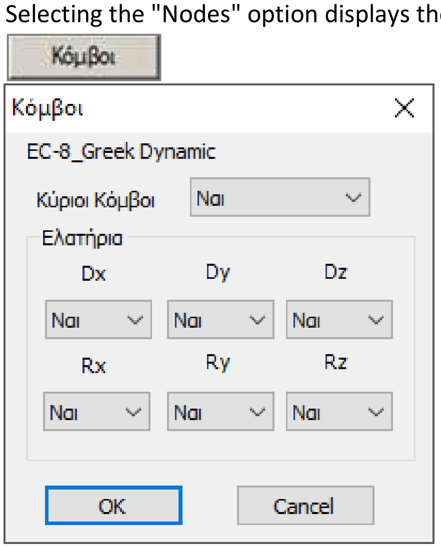
The program automatically selects, depending on the scenario regulation, the corresponding inertial multipliers so any modification is optional.

If, for example, you select "EC", the multipliers for the inertias of the linear structural elements to be taken into account in the analysis based on the provisions of the Eurocode are updated.

Also, here you can set the aspect ratio for the vertical elements in order for them to be marked as "Valley".



Selecting the "Nodes" option displays the following dialog box:



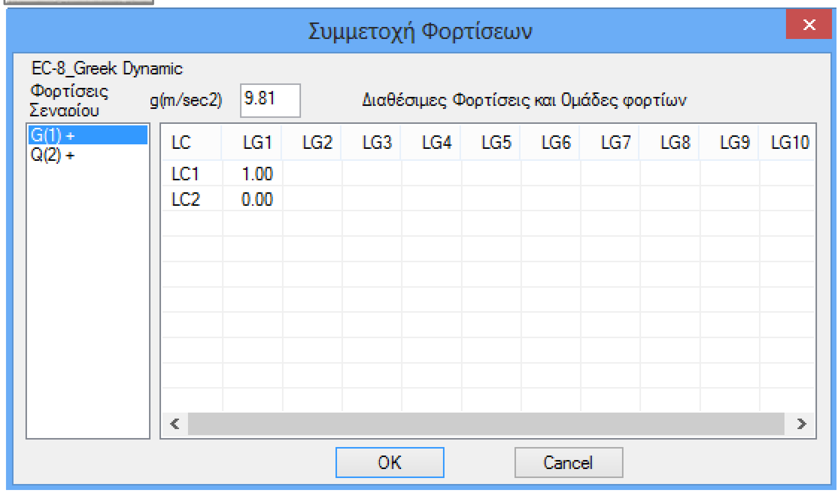
where you choose to take into account the diaphragmatic function of the plates (F.S.R.) ("Yes" default) or not ("No")

In addition, in a similar way, you choose whether or not to allow the relative movements for the foundation springs, i.e. whether you want the building to be released in a flattened state ("No") or whether you want to take into account the influence of the foundation you have introduced.

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

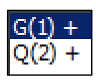
Select 

to update the script and register the changes.
Selecting "Downloads" will display the following dialog box:



where, for each scenario load, you define the corresponding load (LC) including its groups (see "Loads">>"Load groups") with the corresponding multipliers.

- For the scenarios **involving the earthquake**,
  - first select the category "Permanent Loads" - G(1), coloured blue, and set for LC1 a value of 1.00 in all subgroups and
  - then select the category "Mobile Loads" - Q(2), coloured blue, and set LC2 to 1.00 in all subgroups.

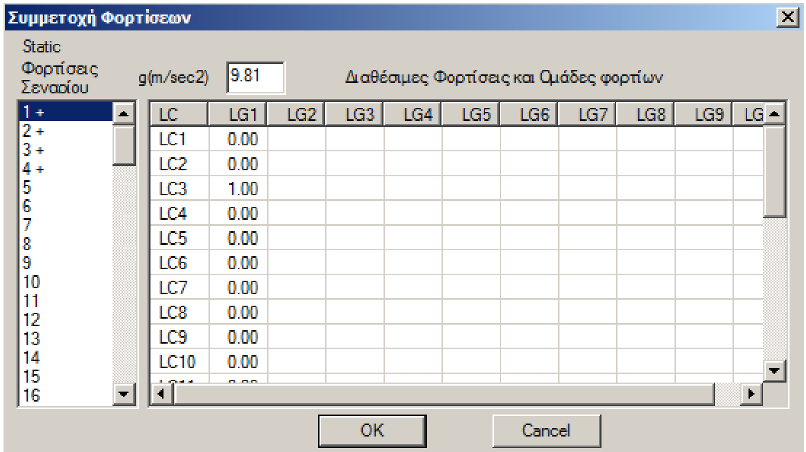
⚠ The "+" next to the charging category 

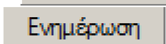
indicates that there is a load participation, i.e. a non-zero multiplier, for that particular charge.

- In scenarios where **the earthquake is not involved** (simple static, e.g. presence of wind), the loads are shown as numbers and in each load you define, with a factor of 1, the presence of the corresponding load.

⚠ Each scenario can include up to 4 loadings.

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'





to update the script and register the changes.

The new **Run All Analyses** command allows you to run all the scripts in the list with one click.

5.2 How to run an analysis script

select from the list of scenarios, the Active Scenario, i.e. the one that will be used for the analysis of the study.

In the list of scenarios, in addition to the two predefined ones, you now find all other scenarios you created before. Select one scenario at a time and continue by setting the parameters of the corresponding analysis

Selecting the "Run" button, depending on the "Active Script", opens the corresponding dialog box, which differs for:

- √ the scenarios of the **NAC**
- √ the scenarios of the **Eurocodes** and
- √ the **Anelastic analysis** scenarios

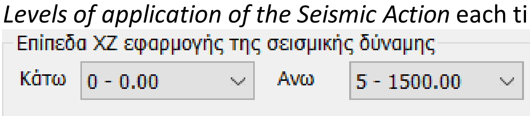
EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

First of all, select Ενημέρωση Δεδομένων to update the parameters of the active script.

Then, select Παράμετροι to set the parameters of the specific study.

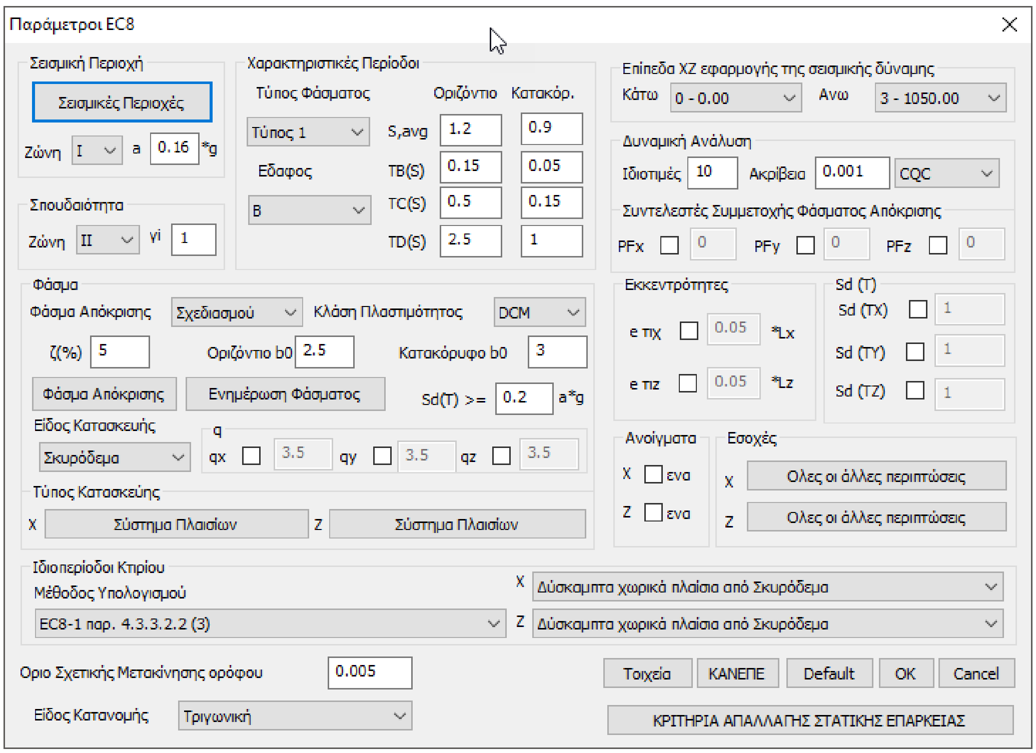
OBSERVATION Ενημέρωση Δεδομένων

After the Data Update, the Parameters you previously set are retained. However, you must set the *XZ Levels of application of the Seismic Action* each time



Depending on the script you select, the configuration dialog box varies.

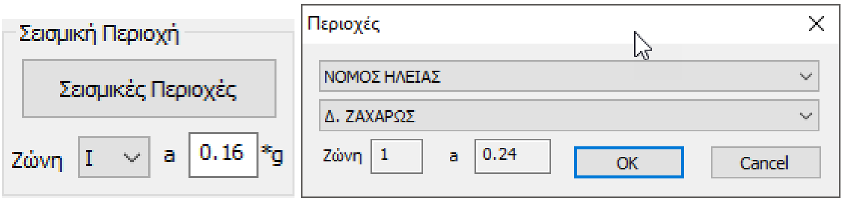
In this example, having chosen the Eurocode 8 scenario, the dialog box will have the following format:



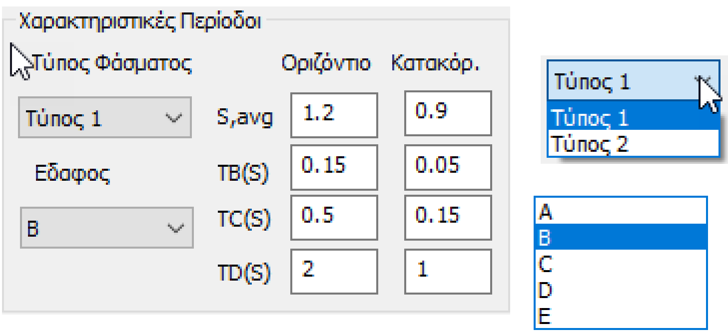
Enter the necessary information about the seismic area, the ground and building, as well as the earthquake coefficients and application levels:

Select the ***Seismic Area*** to determine the Zone and therefore the Seismic Acceleration a:

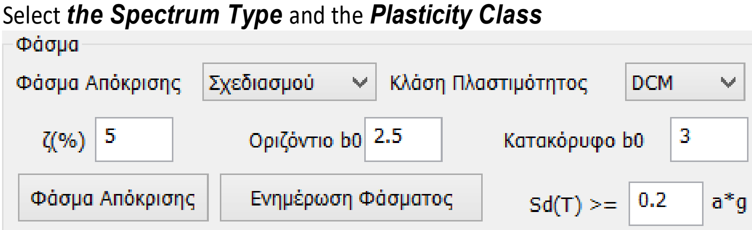
EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'



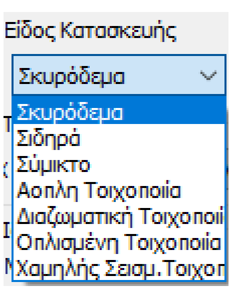
Select the ***Spectrum Type*** and Soil ***Category*** to determine the type of the ***spectrum*** and the ***Feature Periods***:



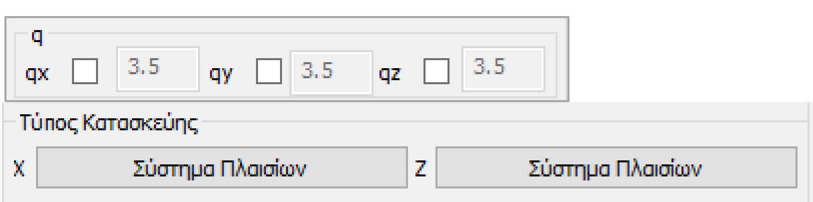
Select ***the Spectrum Type*** and the ***Plasticity Class***



Select the ***Type of Construction***



The selection of the ***Seismic Coefficient q*** and the ***type of construction*** requires complex calculations.

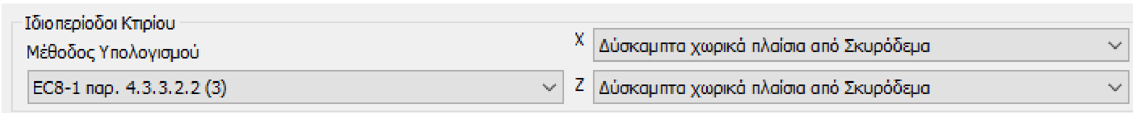


EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

ScadaPro allows the designer to get rid of them and follow the procedure described in the next chapter: "***§ How to calculate the seismic coefficient q "***

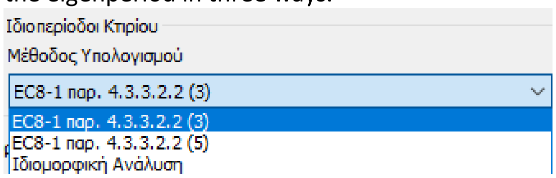
In the ***Building Properties*** field:

Where in previous versions there was the ***Building Type*** by X and Z field for the calculation of the basic eigenperiod, it has been replaced by the module:



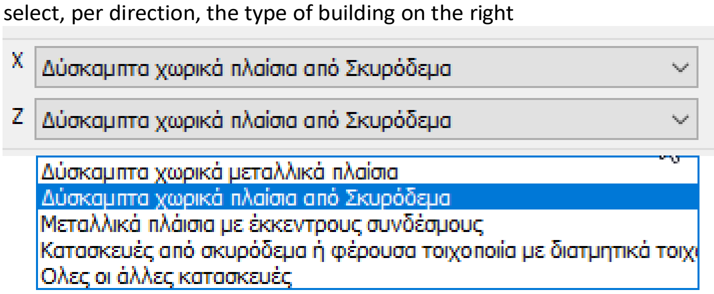
There is now the option to choose to calculate the eigenperiod in three ways.

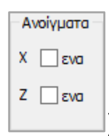
The option of choosing the



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

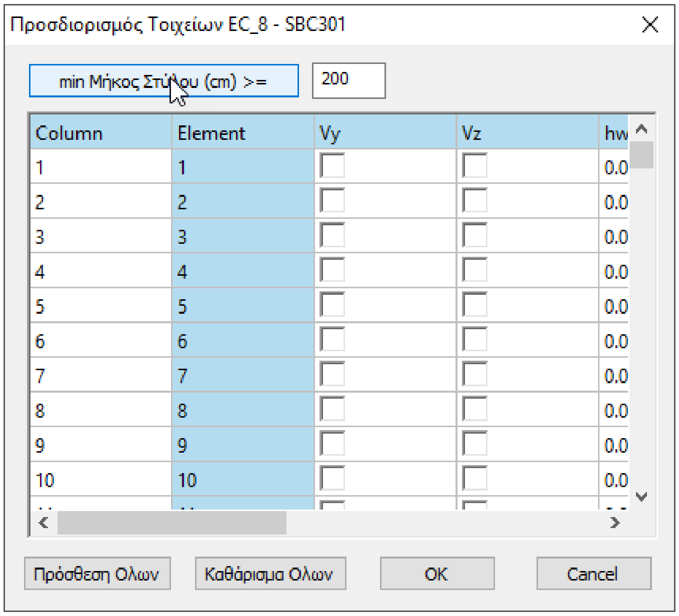
1. In the first EC8-1 παρ. 4.3.3.2.2 (3) is necessary:
   select, per direction, the type of building on the right



(in the case of X and/or Z where the structure consists of a single frame the corresponding checkbox in the "Openings" box is activated 

)

Then, select the "Vesselscommand Τοιχεία to specify based on a minimum length which of the vertical elements are defined as "Vessels".



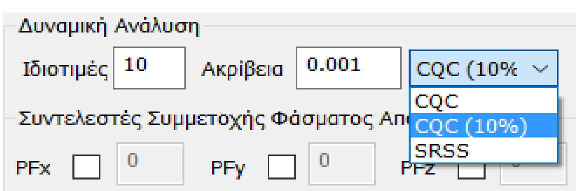
Enter the min Length (cm) and press the command "min Column Length" to automatically determine the walls per direction, so that the calculation of T1 is done according to par.4.3.3.2.2.

2. The second approximate method EC8-1 παρ. 4.3.3.2.2 (5) , is sufficient to be selected and does not require any additional action.

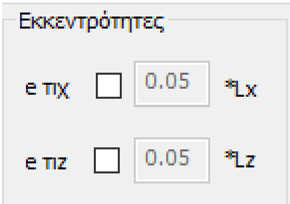
3. The third possibility is to calculate the eigenpipes by Idiomorphic Analysis.

The program takes as the building's eigenvector per direction the eigenvector corresponding to the dominant eigenmode (the eigenmode with the highest percentage of activated mass).

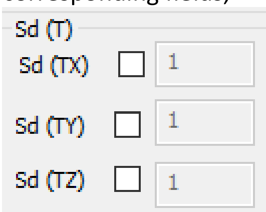
The user can increase or decrease the number of Idiosyncrasies, in case of dynamic analysis, <u>and Static, in case the user chooses to calculate the eigenvalues from Idiomorphic Analysis</u>, and the accuracy rate.



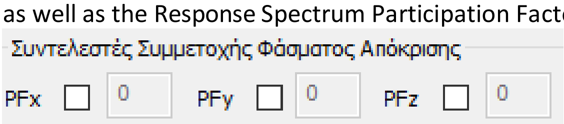
It is also possible to choose the mode of overlap of the eigenmodal responses either according to the Full Quadratic Parallelism CQC and CQC(10%) rule (3.6 EAK), or the Simple Quadratic Parallelism SRSS rule. Also, the results of the seismic action now include the results of the eigenmode analysis for the static scenarios.
To modify the coefficients for the ***Eccentricities***, select the corresponding checkbox and enter the new value on the right.



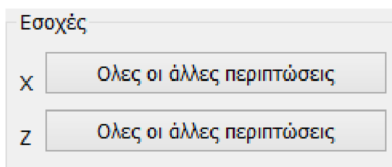
In the same way, the designer can modify the spectra by X, Y and Z by entering his own values in the corresponding fields,

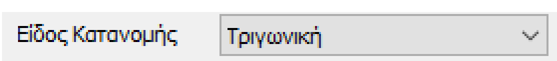


as well as the Response Spectrum Participation Factors

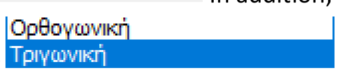


In the ***Slots*** field, select for each direction the case that is appropriate for the specific study and that is defined by the EPC.





In addition, the researcher can choose the ***Type Distribution*** of seismic force

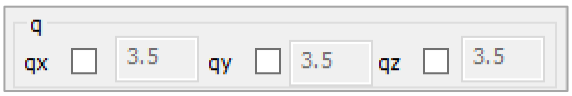


Method of calculating the seismic coefficient q

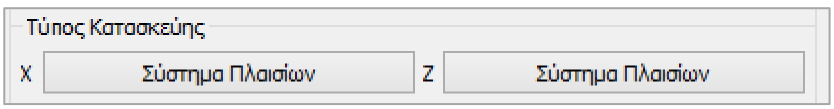
According to the Eurocode, the **"Seismic Coefficient q"** is derived from a calculation and the **"Type of Construction"** from specific criteria.

⚠ SCADA Pro automatically calculates the q and the type of construction. The procedure for automatic calculation is as follows:

- After filling in all the previous fields, leave:

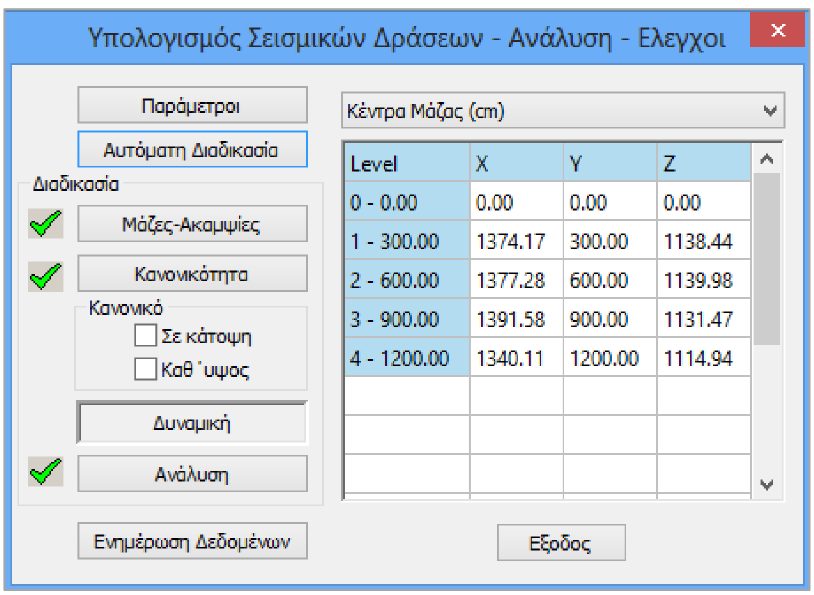


and



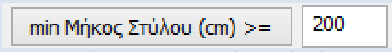
as they are.

- Select "OK" and with "Automatic Process" perform a **first analysis**.

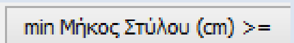


- Select the "Controls" command  and in the dialog box that appears select "OK".

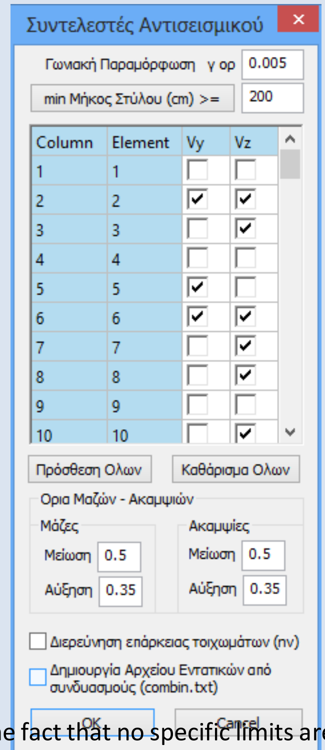
In the context of the dialogue "Earthquake Factors"



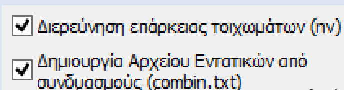
you set the minimum length that a post must have be considered a wall. By pressing the key



in the list of poles, the walls are automatically checked in each direction.

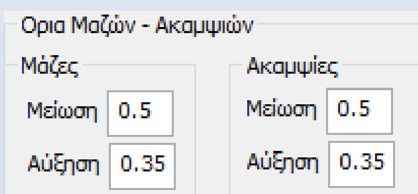


In addition, by activating the checkboxes



you indicate the creation of the corresponding .txt files, which are automatically registered in the study folder and can be printed.

The wall adequacy investigation includes a detailed analysis for each level and for each combination of the cutting force received by each wall.



In the limits field, and due to the fact that no specific limits are defined by the Eurocode (unlike the NAC), you can modify the mass and stiffness limits.

In the check file and in the calculation of the wall shear, the program "determines" the structural system of the building based on the seismic wall shear check.

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

		Σελίδα : 1
ΑΠΟΤΕΛΕΣΜΑΤΑ ΕΛΕΓΧΩΝ		
ΣΕΝΑΡΙΟ :	ΔΥΝΑΜΙΚΗ ΦΑΣΜΑΤΙΚΗ ΜΕΘΟΔΟΣ ΜΕ ΟΜΟΣΗΜΑ ΣΤΡΕΠΤΙΚΑ ΖΕΥΓΗ (EC8)	

Έλεγχος Διαφοράς Μαζών και Ακαμψιών Σταθμών Κτιρίου							(παρ.4.2.3.3.)
α/α Στάθμης	Συν/κο Υψός (m)	Συν.Μάζα KN/g	Συνολικες Ακαμψιες Ki*10^3 (KNm)		Διαφορές Μαζών - Ακαμψιων (Mi+1-Mi)/Mi - (Ki+1-Ki)/Ki		
			(Ki-X)	(Ki-Z)	(ΔMi)	(ΔKi-X)	(ΔKi-Z)
1	3.000	123.750	4867.198	2168.954			
2	6.000	57.199	3893.758	1735.163	ελ. 0.53	ελ. 0.19	ελ. 0.20
Ο Έλεγχος ικανοποιεί τα Κριτήρια Κανονικότητας						ΝΑΙ	
						ΟΧΙ	
ΣΗΜΕΙΩΣΕΙΣ:	Μάζες : Η Αύξηση πρέπει <= 0.35 - Η Ελάττωση πρέπει <= 0.50 Ακαμψίες : Η Αύξηση πρέπει <= 0.35 - Η Ελάττωση πρέπει <= 0.50						

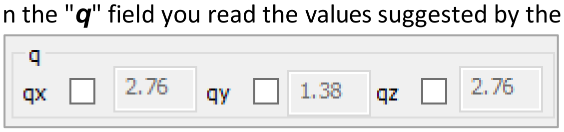
Κέντρο Βάρους - Κέντρο Ακαμψίας						
α/α Στάθμης	Συν/κο Υψός (m)	Κέντρο Βάρους		Κέντρο Ακαμψίας		Απόσταση
		X Συντ.(m)	Z Συντ.(m)	X Συντ.(m)	Z Συντ.(m)	K.B - K.A (m)
1	3.000	5.4309	6.0895	6.2884	5.6797	0.9503
2	6.000	5.3788	5.6738	6.7783	5.4379	1.4192

Σεισμική Τέμνουσα Τοιχωμάτων										Παρ. 5.1.2.
Σεισμική Τέμνουσα Τοιχωμάτων						Στάθμη Αναφοράς			0 0.000(m)	
α/α Στάθμης	Συνδ/μος	Τέμνουσα Τοιχ./Συνολική Τέμν. = nvx			ΕΠ./ΑΠ.	Συνδ/μος	Τέμνουσα Τοιχ./Συνολική Τέμν. = nvz			ΕΠ./ΑΠ.
		Τέμνουσα Τοιχωμάτων	Συνολική Τέμνουσα	nvx			Τέμνουσα Τοιχωμάτων	Συνολική Τέμνουσα	nvz	
1 ***	0	0.000	0.000	0.00	ΑΠ.	0	0.000	0.000	0.00	ΑΠ.
2	0	0.000	0.000	0.00	ΑΠ.	0	0.000	0.000	0.00	ΑΠ.
ΣΗΜΕΙΩΣΕΙΣ:	*** = Στάθμη ελέγχου nv από κανονισμό									

Καθορισμός Συστήματος Κτιρίου	
Διεύθυνση X:	Σύστημα Πλαισίων
Διεύθυνση Z:	Σύστημα Πλαισίων

Knowing the "***Construction Type***" and all the previous parameters , the program can calculate the "***Seismic Coefficient q***". Enter in the parameters the last information, i.e. the '***Type of Construction***', run the analysis a second time and enter the parameter dialogue box once more.

In the "***q***" field you read the values suggested by the program.

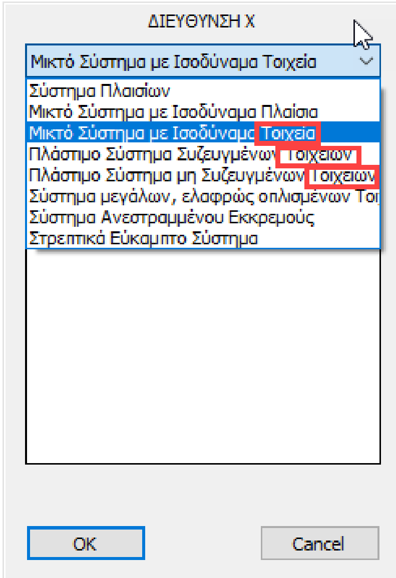


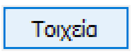
You can proceed by keeping these values or modify them by checking the corresponding checkboxes and typing your own values (which you could have done from the beginning, but then the program would receive your values without suggesting its own).

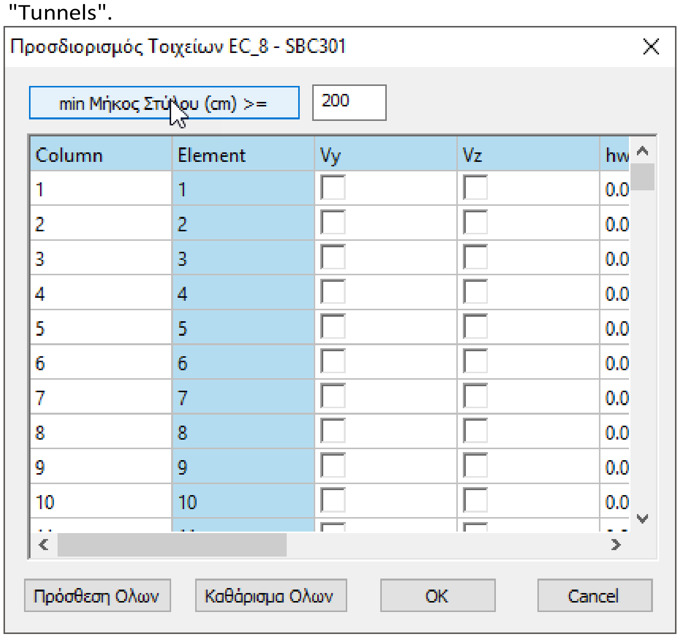
OBSERVATION:

Where the building type includes the word "walls"

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

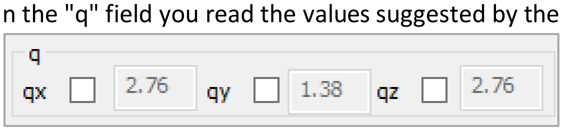


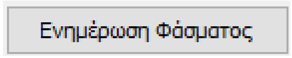
then to calculate the coefficient α0 and ultimately q you should select the "Tunnelscommand  to define, based on a minimum length, which of the vertical elements are defined as "Tunnels".



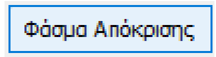
Enter the min Length (cm) and press the "min Column Length" command automatically determine the walls per direction to calculate the coefficient α0.

In the "q" field you read the values suggested by the program.

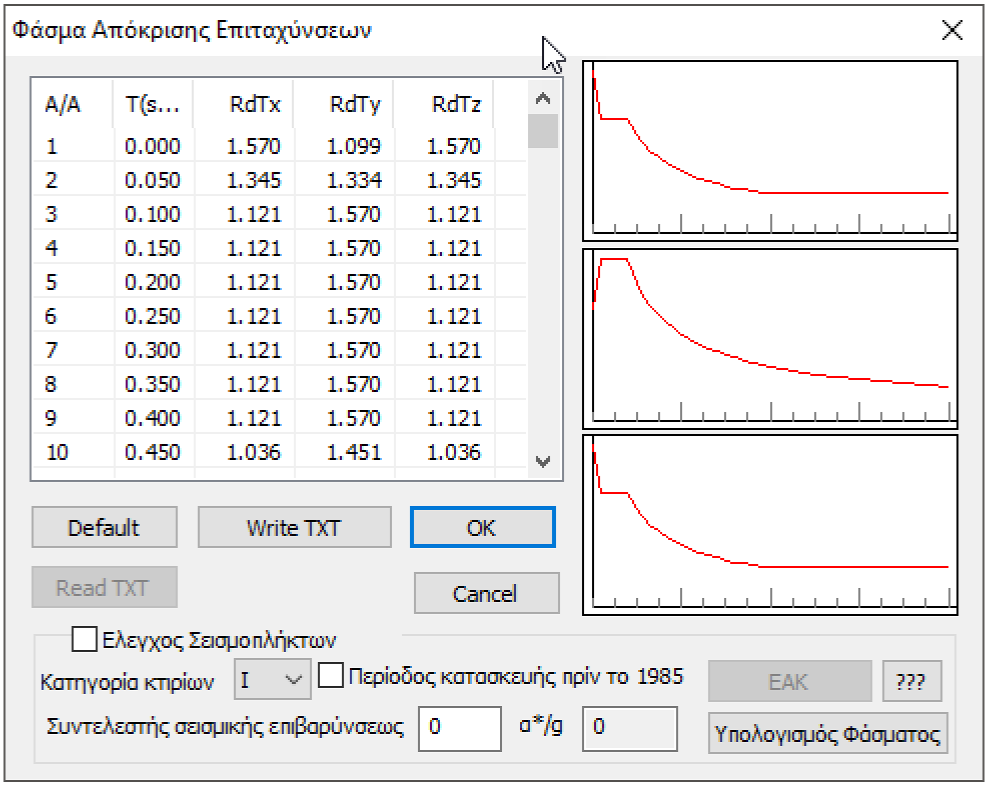


Select 

to update the spectrum with the seismic values

Coefficient q and 

to see.

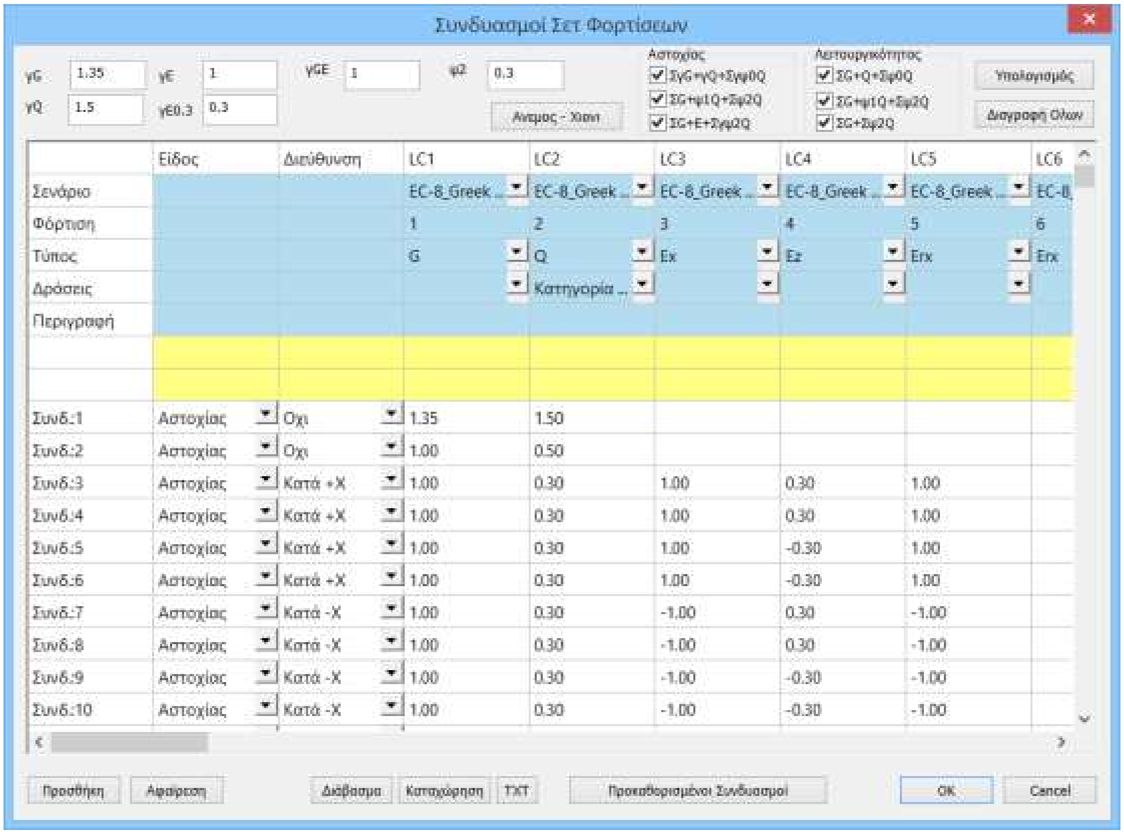


Select "OK" and with "Automatic Process" run the <u>analysis a second time take the new parameters into account.</u>

5.3 How to check the results of the analysis and create the combinations

Immediately after running the selected analysis scenario, using the commands in the "Results" field, you create the combinations (for the EC8 checks and sizing) and display the results of analysis checks:

Selecting the "Combinations" command opens the "Load Set Combinations" dialog box where you can create your own combinations or call the predefined ones included in the program.



After running an analysis scenario, its combinations are automatically generated by the program. Calling the command "Combinations" opens the table with the combinations of the active scenario.

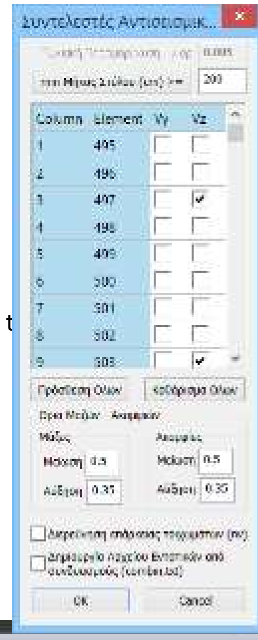
- The same is achieved by selecting the "Predefined Combinations" command, as the program will enter the combinations relevant to the active analysis scenario.

- The predefined combinations of the "running" scenarios of the analysis are automatically entered by the program.

- In addition to the predefined combinations, the designer has the possibility to create his own combination files, either by modifying the predefined ones, or by deleting all of them "Delete All" and entering his own values. The "Load Set Combinations" tool works like an Excel page offering copy, total delete capabilities in the classic ways, Ctrl+C, Ctrl+V, Shift and right-click

- The predefined combinations refer to seismic scenarios. To create combinations of scenarios that do not contain an earthquake, both **automatic** and **manual** modes are available.

5.4 Checks

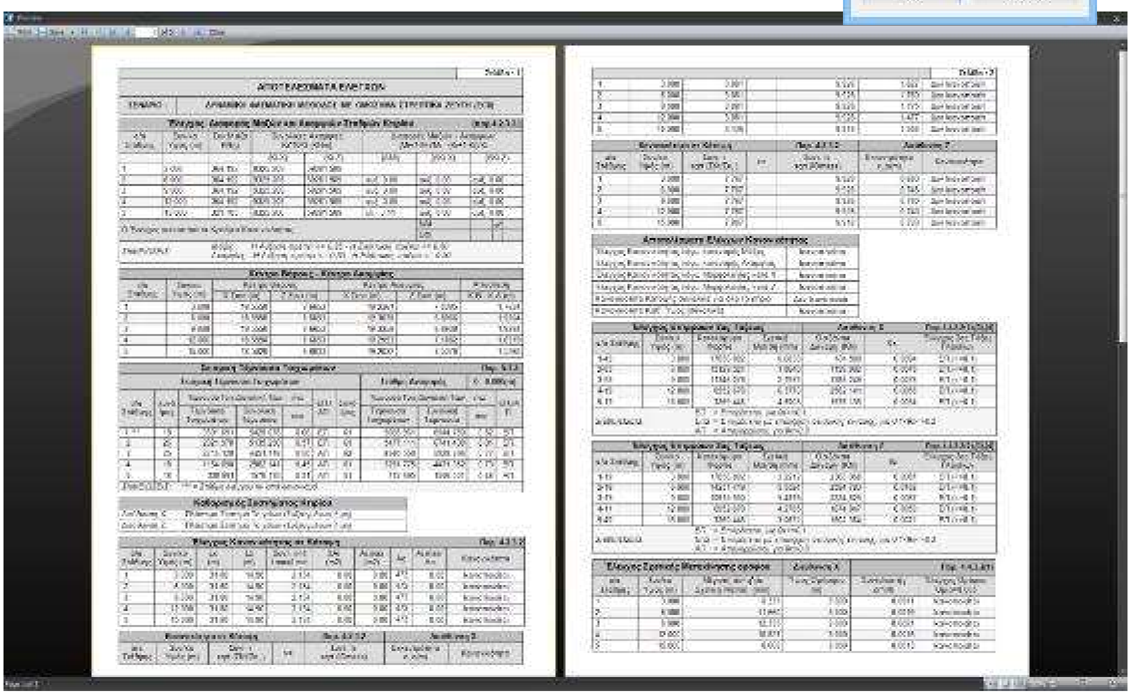


Select the "Checks" command and in the dialog box:

- √ enter the minimum length for defining the walls and click the corresponding button,
- √ set the mass and stiffness limits for the normal conditions of the building,
- √ Enable the creation of the two .txt files
- √ "OK."

Automatically opens a file that, for "active analysis". includes the results of t

- Regularity
- 2nd order influences
- Framework instability
- Floor Angular Deformation
- Wall Adequacy
- Building torsional sensitivity
- Calculation of Seismic Moment
-



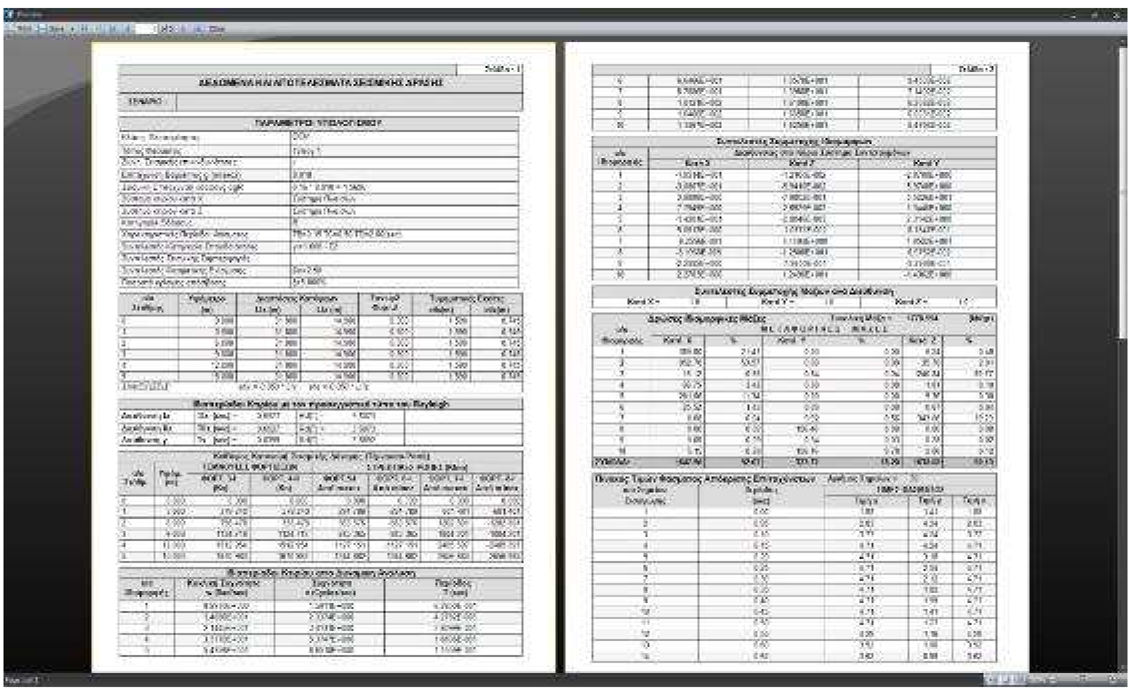
5.5 Seismic action

Select the command "Seismic Action" and automatically opens a .txt file that includes the Calculation Parameters for the seismic action, and the calculation results for the following quantities:

- √ Building Specifications
- √ Station design eccentricities in relation to the Plasmatic Axis
- √ Plumbing Distribution of the Equivalent Static Load (Torsion-Torque)
- √ Values of the response spectrum

When the analysis scenario is Dynamic analysis, the following sections are included in this file:

- √ Building Idiosyncrasies from Dynamic Analysis
- √ Idiom Participation Factors
- √ Mass Participation Rates / Address
- √ Eating Idiomorphic Masses



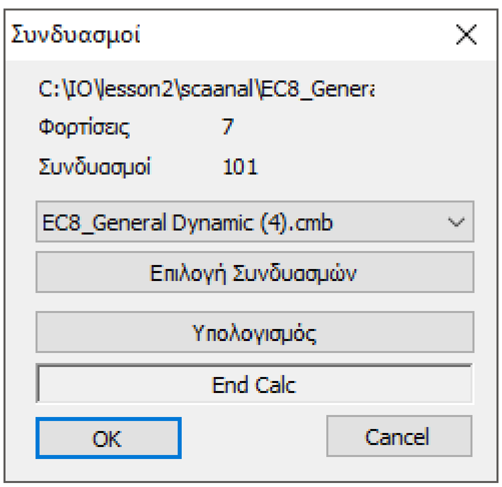
⚠ *For further clarification and details see. For further details, please refer to the User Manual § 8A. ANALYSIS*

RESULTS

6.1 How to view diagrams and deformations, as well as the reinforcement of the paving :

Go to the "Results" section to see the deformations of the beam from each load or combination under scale and the M,V,N diagrams for each member.

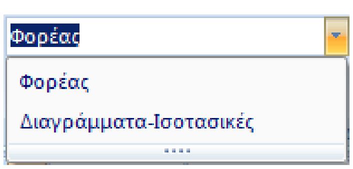
Depending on the results you want to see, from the "Combinations" command and within the dialog box:



- Select a combination from the list that includes the combinations all of "running" analyses, and let them complete their calculation automatically, or

- press the "Select File" button, select the file of combinations from the study folder and press the "Calculate" button.

⚠ To view vector deformations from eigenmodes of the dynamic analysis, select Dynamic analysis combination file.

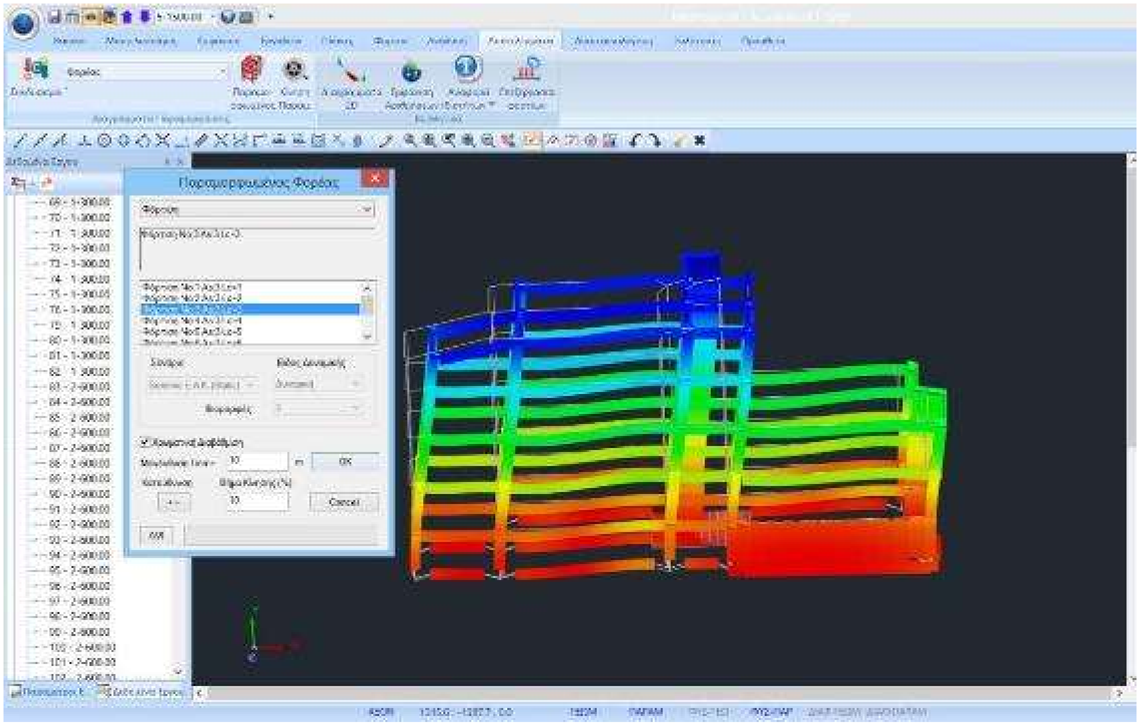


From the list on the right, depending on the results you want to see, select:

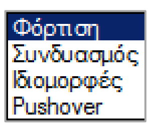
√ Institution or

√ Charts-Important

6.1.1 Body+ "Deformed Body"

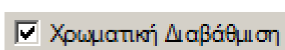


Select from the list



the type of loading for which you want to see the deformation image of the carrier and from the next list specify its number.

Activate

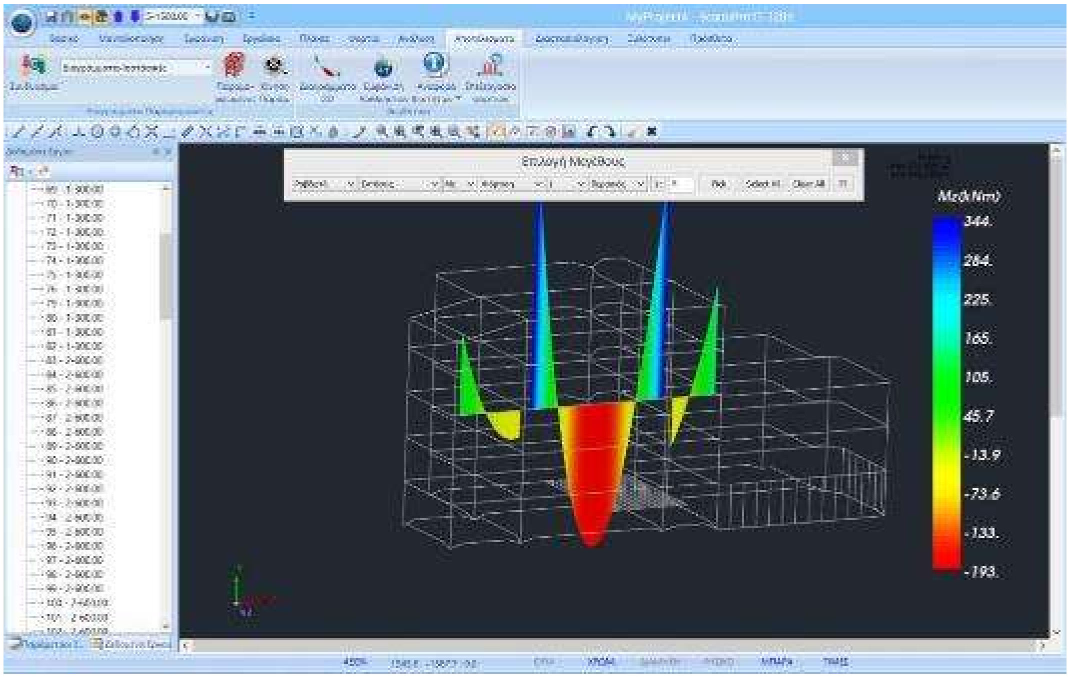


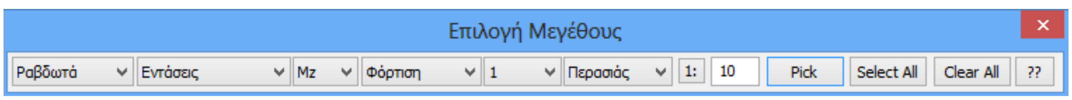
, modify the "Scale" and the "Motion Step" to see the best and most intuitive visualization.

In the "Status Bar" double-click (blue = active, grey = inactive) to select the mode of displaying the deformed vector.

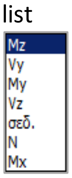
The "Motion" command is the switch that turns on and off the motion of the deformed vector, according to the choices you made in the dialog box of the previous command.

6.1.2 Charts - Equalisation

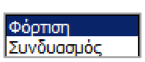




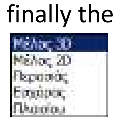
In this section you can see on the members the diagrams of the stresses for the linear members, and the isometric curves of stresses, strains and reinforcements for the finite surface elements. In particular, to see the diagrams of the stress magnitudes for the Ribbon elements, select the stress magnitude from the list



, then select the type of charge or combination or envelope and finally the way the chart is

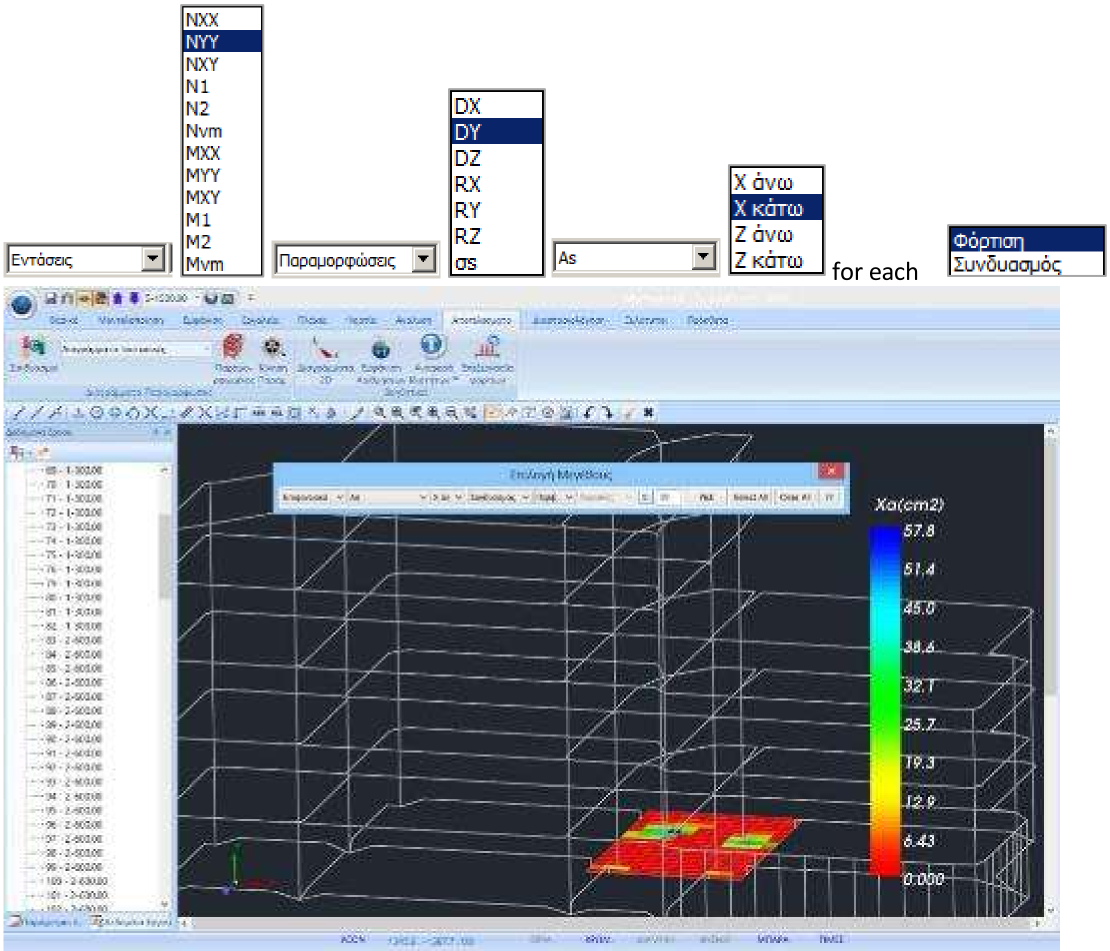


displayed

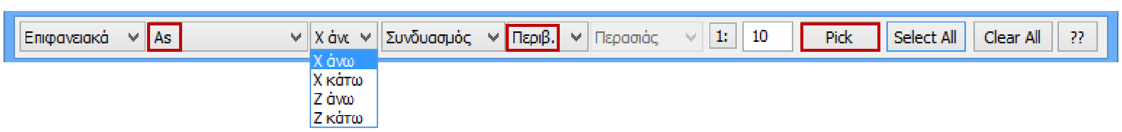


Similarly for Surface Elements select whether you want to display isometric curves for stress, strain or As reinforcement as well as loading or combination:

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'



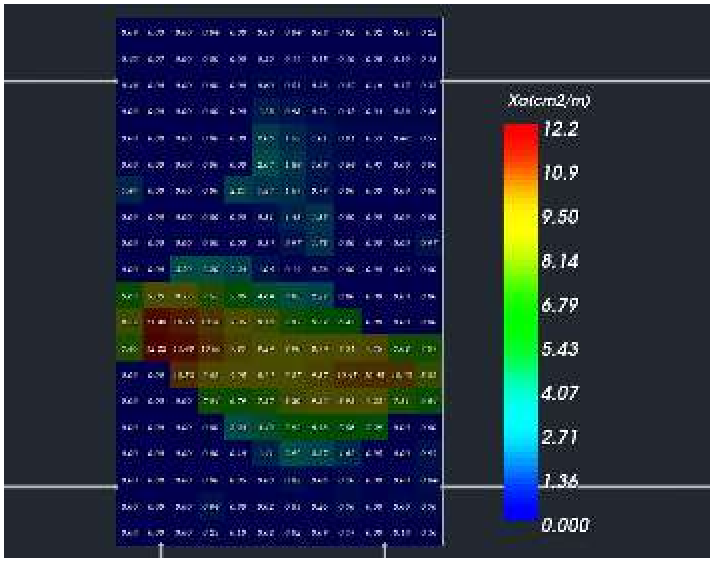
To view the reinforcement of the pavement in x and z, top and bottom, select:



The colour illustration and the bar on the right shows in colour gradation the area of reinforcement required per direction and side.

⚠ By activating "PRICES" in the bottom horizontal bar, you can see the values of the selected size on the surface of the surface element.

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'



⚠ *For more details and details see Manual Usage § 8. RESULTS*

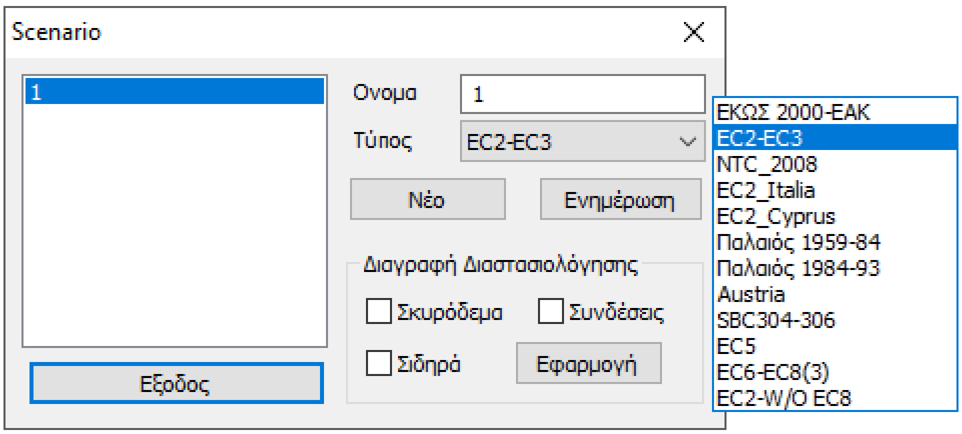
DIMENSIONING

After completing the analysis of the structure, checking the results and the deformations, the next step to complete the design is the dimensioning of the structural elements.

7.1 How to create dimensioning scripts :

Go to the "Sizing" section and select the "New" button to create the scenario you wish by selecting the regulation (EKOS, EUROCODES, Old regulations, for Greece).

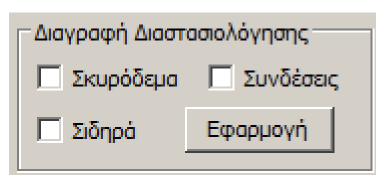
...defined scripts are created according to the Rules and Attachment option you make at the beginning, within the General Parameters window that opens automatically immediately after you define the file name.



Type a name, select a type and New to populate the list of scenarios.

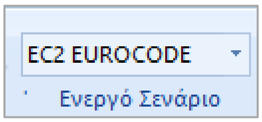
In this example, a Eurocode scenario was used.

In the "Delete Dimensioning" field, activate the corresponding checkbox and "Apply", to delete the results of a previous dimensioning (concrete elements, steel sections, or connections respectively), in order to dimension from scratch using other combinations, or parameters, or scenario, etc.

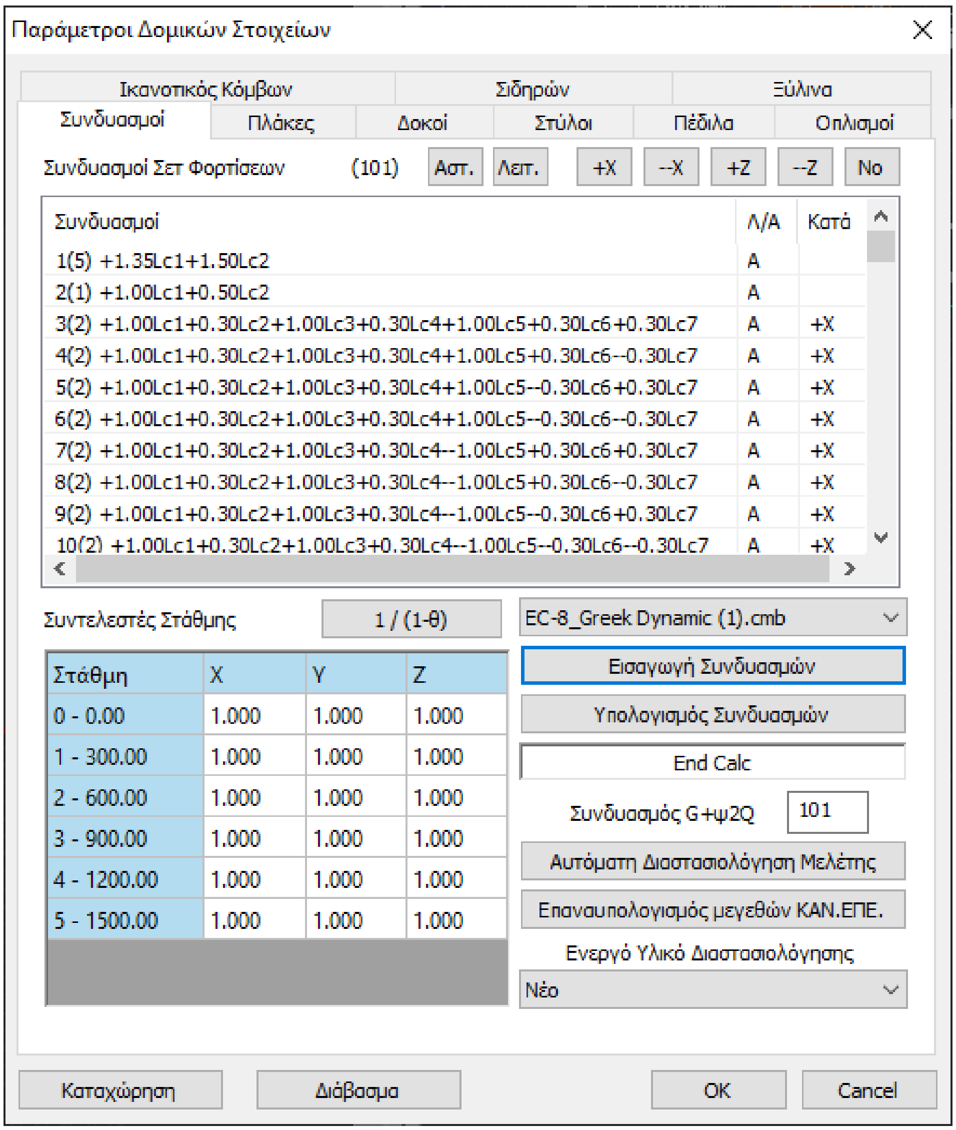


7.2 How to define the parameters of dimensioning, per building element :

From the list of scenarios you have created, select the scenario you will use for sizing.
(**See Internal Use Chapter 9 "Sizing"**)

With active 

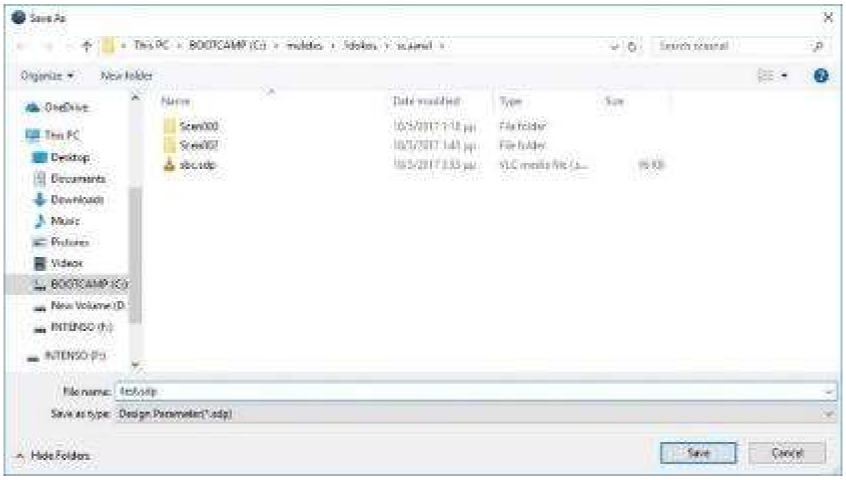
the selected scenario, you open the Parameters



EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

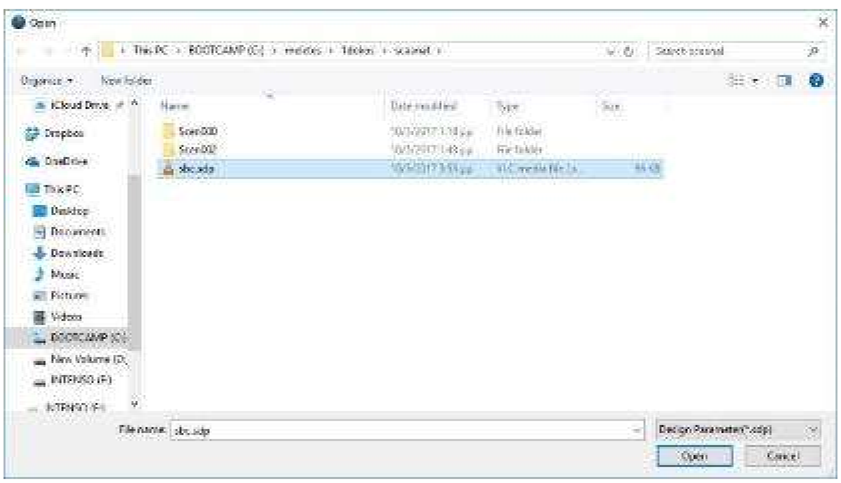
Two new commands concerning the storage of the sizing parameters of the active scenario.

Once you have configured the sizing parameters, you can now save them in a file to use them in your next study.
Pressing the "Register" button opens the storage box



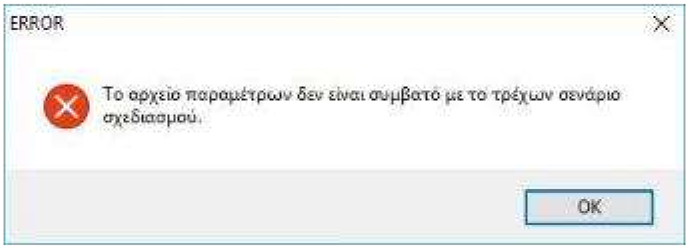
where you type a name (it is good to be relevant to the sizing scenario).

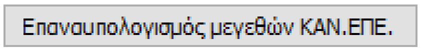
The extension of these files is sdp scenery design parameters.
Similarly, with the "Read" option, you can load a previously saved parameter file into a study.



ATTENTION

A prerequisite for calling a configuration file is that the current sizing script is the same as the configuration script you are calling. Otherwise you will see the message

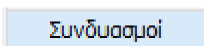




A new command that allows the recalculation of all the sizes provided by the CEE for all members of the design and is used in cases where the strength of the materials is changed while the reinforcement has been placed according to the existing situation.

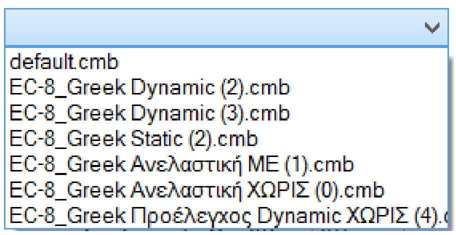
Combinations

Regardless of the material, a prerequisite for sizing is the calculation of combinations.



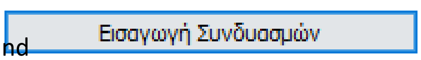
The selection of the .cmb file of the combinations registered by the analysis is either:

- from the list

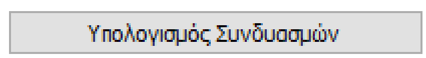


with automatic calculation

- through the command



where, from within the study folder, you select from the registered the file the of combinations of with the which will dimension and then use the button to make the



calculation.

Depending on the case and the conditions that are met, you can use either the combinations of static or dynamic to dimension the superstructure (as long as in the analysis you have "opened" the springs (not the footing)). You can also have run analyses with scenarios of different codes (e.g. EAK and EC8) and by dimensioning with the respective combinations you can see the differences that result.

In the "Combinations" field the list of all combinations is displayed. In the field "Level

coefficients"

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

Συντελεστές Στάθμης 1 / (1-θ)

Level	X	Y	Z
0 - 0.00	1.000	1.000	1.000
1 - 400.00	1.000	1.000	1.000
2 - 700.00	1.000	1.000	1.000
3 - 1000.00	1.000	1.000	1.000
4 - 1300.00	1.000	1.000	1.000
5 - 1600.00	1.000	1.000	1.000

You can increase or decrease, by manually entering coefficients different from 1, the seismic actions per direction and level.

⚠ The 1 / (1-θ) button, if selected, will do the 2nd order influence check, <u>with automatic increase of the when 0.1 < θ < 0.2</u>, at the levels required.

To take into account any modifications to the combinations, select the command again .

Υπολογισμός Συνδυασμών

The field Συνδυασμός G+ψ2Q 99 applies only to the scenarios of the Greek regulation (EKOS).

⚠ **OBSERVATION:**

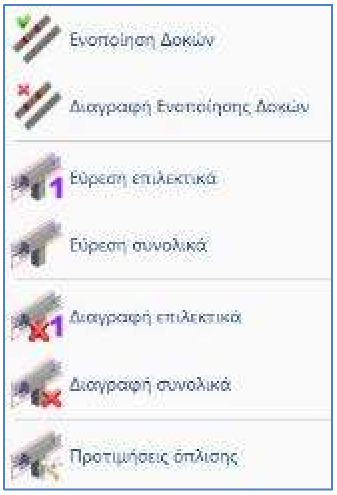
The order Αυτόματη Διαστασιολόγηση Μελέτης is an automation related to **Concrete** studies and allows you to dimension the whole study with a simple "click".
Set the parameters in the fields below and select "Automatic Study Sizing". The program will automatically carry out the entire sizing process that you include in the following groups and that is otherwise followed "Step by Step".

All the parameters of the layering depending on the structural element and the material of the carrier are located in the corresponding tabs and are explained in detail in ***Ench. Chapter 10A "Dimensioning".***

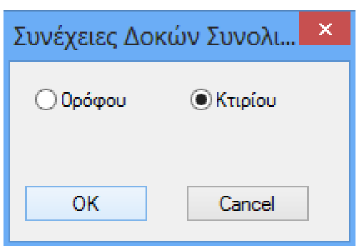
Ικανοτικός Κόμβων | Σιδηρών | Ξύλινα
Συνδυασμοί | Πλάκες | Δοκοί | Στύλοι | Πέδιλα | Οπλισμοί

7.3 How to size the beams:

The "Beams" field contains the commands for finding Beam Continuity, Dimensioning, Reinforcement Check and Beam Continuity Results.



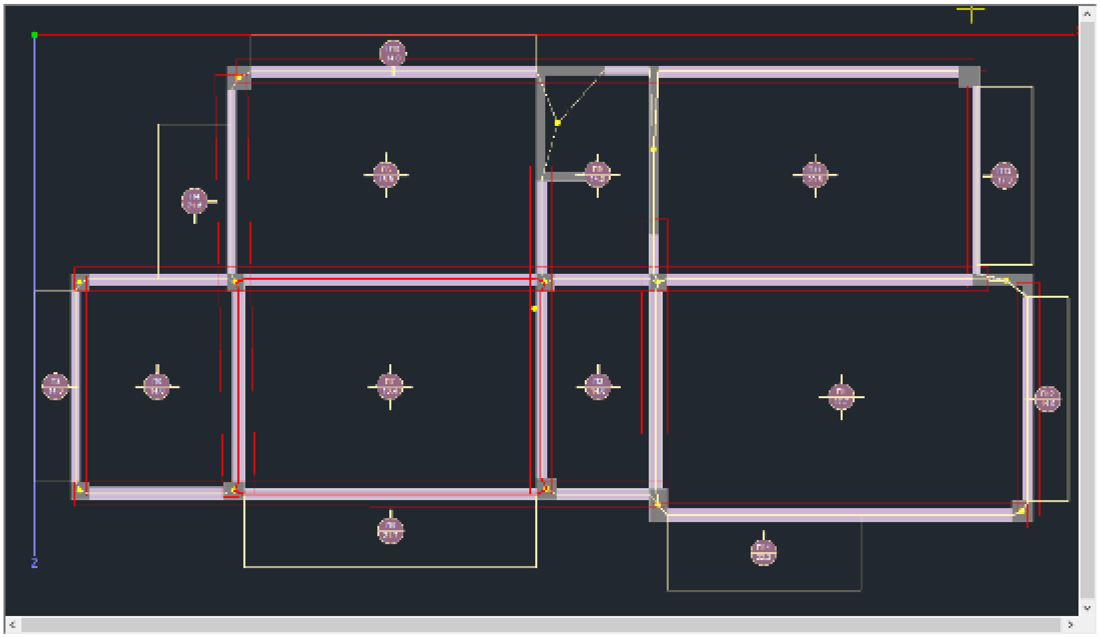
Select the command "*Beam Continuums>Find Total*"



to automatically determine the continuity of the beams of the whole building.

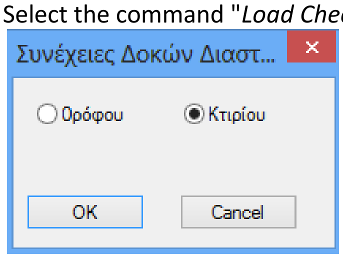
The program automatically creates all the beam passes.

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'



In *"Arming Preferences"* you specify whether one or two reinforcement bars will be placed as common support reinforcement in the beams, whether you wish to take into account the bars of both openings in the support reinforcement, as well as the anchorage length by varying, if you wish, the support width of the beam.

Select the command "*Load Check>Total*" dimension the beams in total for the whole building.



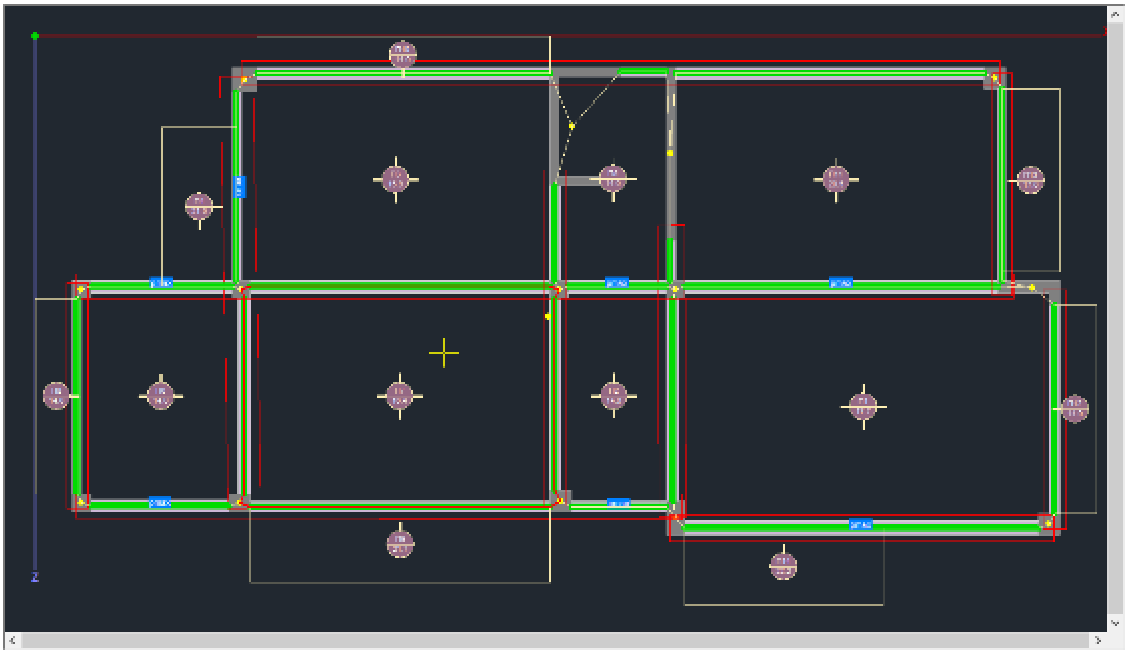
The beams are coloured with the corresponding colour indicating the type failure and the initials K, D, S, d, pmax are written on their axis to indicate the type failure (K: Bending, D: Shear/Torsion, S: Connectors).

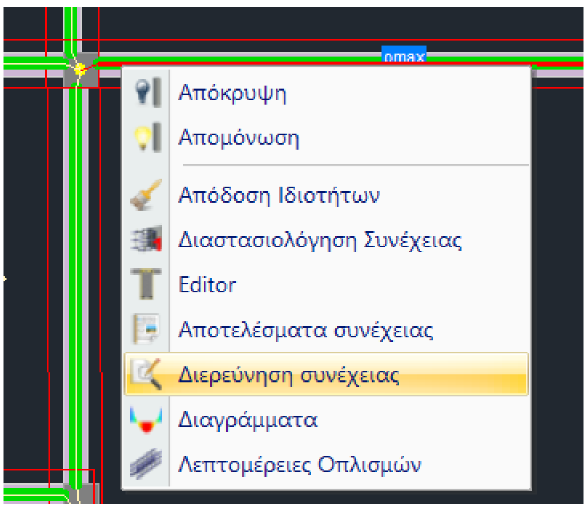
- **Red**. Bending failure. Maximum reinforcement rate ρmax exceeded. Dense Connectors.
- **Pink**. Shear/Torsion failure ..
- **Cyan**. The beam was dimensioned without any problem.

Initials indicating the type of failure also appear on the beam.

For this particular example, the dimensioning of the beams showed some failures for exceeding the maximum reinforcement percentage in the supports marked "p".

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

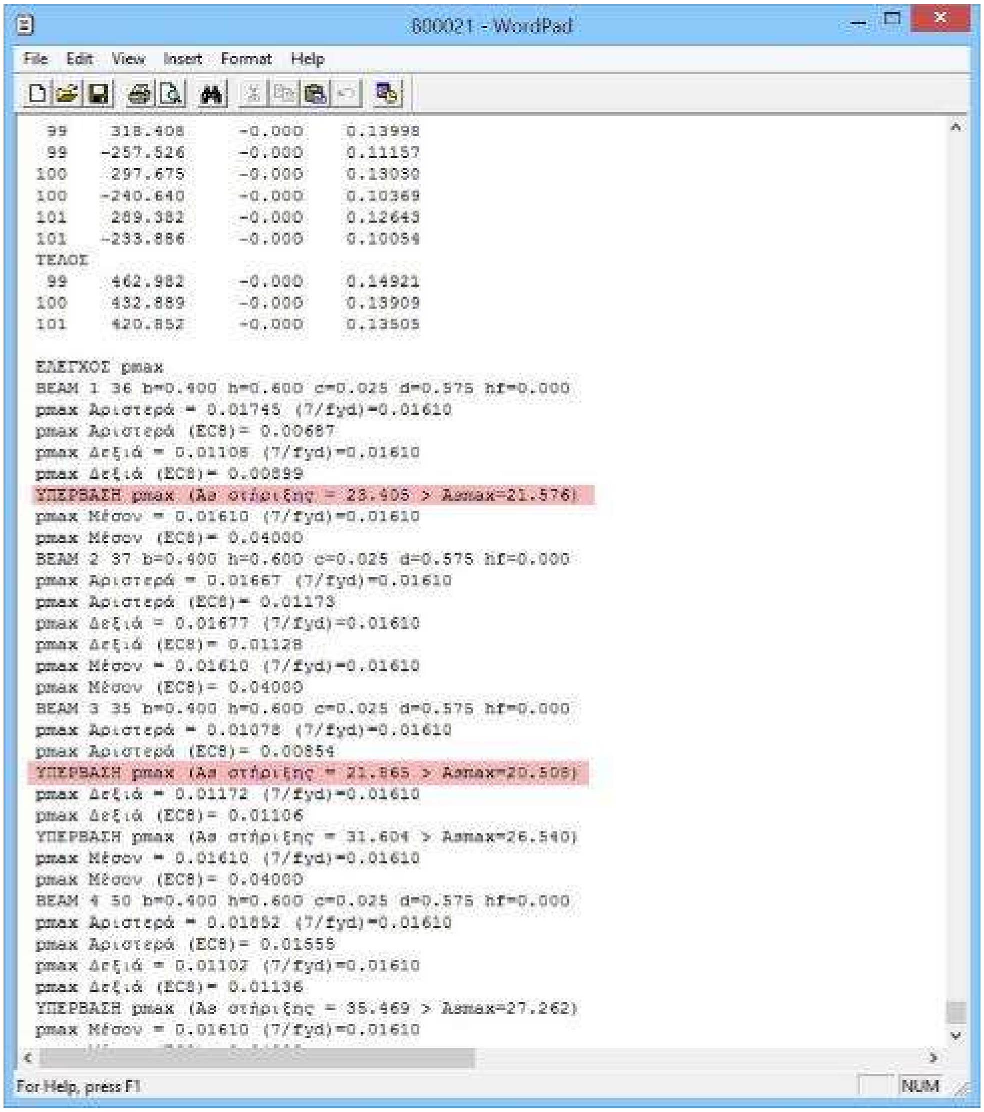




Right-clicking on the member of the beam that fails opens a list of commands related to the dimensioning of the continuity.

Select "**Continuity Investigation**" check the failure from analytical results file that opens:

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

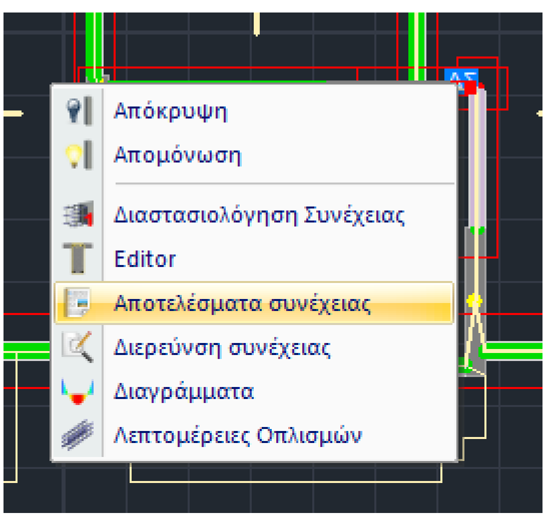



```
 99   318.408    -0.000   0.13998
 99  -257.526    -0.000   0.11157
100   297.675    -0.000   0.13030
100  -240.640    -0.000   0.10369
101   289.382    -0.000   0.12643
101  -233.886    -0.000   0.10054
ΤΕΛΟΣ
 99   462.982    -0.000   0.14921
100   432.889    -0.000   0.13909
101   420.852    -0.000   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****: information about most forms of failure in the receive via the "**Continuity Results**" file*

*For example, for a failure marked "**D**": Select "**Continuity Results**" to check the failure from the summary results file that opens:*



EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

	Σελίδα : 1
Δ14 Δ1	

ΧΑΡΑΚΤΗΡΙΣΤΙΚΑ ΔΟΚΟΥ									
Δοκός	Μέλος	Κόμβος αρχής	Κόμβος τέλους	Μήκος L(m)	Είδος	Πλάτος b_w(m)	Ύψος h(m)	Πάχος h_f(m)	Πλάτος b_m(m)
14	124	31	32	1.30	Ορθογων	0.45	0.60		

ΣΚΥΡΟΔΕΜΑ							
Ποιότητα	f_{ck} (MPa)	γ_{cu}	γ_{cs}	max ε_c (N,M)	max ε_c (N)	f_{ctm} (MPa)	τ_{rd} (MPa)
C20/25	20.00	1.50	1.00	0.0035	0.002	2.20	0.25

ΧΑΛΥΒΑΣ ΟΠΛΙΣΜΟΥ							
	Ποιότητα	E_s (GPa)	f_{yk} (MPa)	γ_{su}	γ_{ss}	max ε_s	Επικάλυψη c(mm)
Οπλισμός κάμψης	B500C	200.00	500	1.15	1.00	0.02	25
Συνδετήρες	B500C	200.00	500	1.15	1.00	0.02	

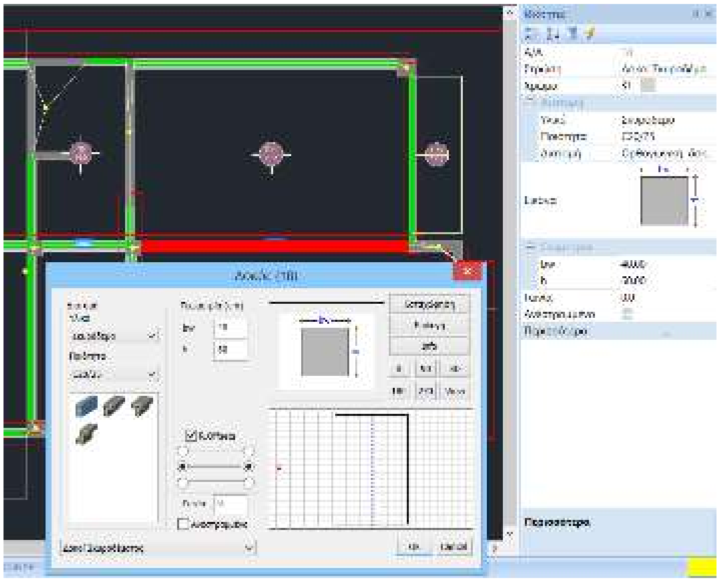
ΕΛΕΓΧΟΣ ΣΕ ΚΑΜΨΗ ΜΕ ΑΞΟΝΙΚΗ							
		ΣΤΗΡΙΞΗ ΑΡΧΗΣ		**ΑΝΟΙΓΜΑ**		**ΣΤΗΡΙΞΗ ΤΕΛΟΥΣ**	
		Άνω	Κάτω	Άνω	Κάτω	Άνω	Κάτω
Συνεργαζόμενο Πλάτος	b_{eff} (m)	0.45	0.45	0.45		0.45	0.45
Αξονική Υπολογισμού	N_{sd} (kN)						
Ροπή Υπολογισμού	M_{sd}	679.90	-568.60	163.04	-126.33	403.47	-441.68
Καθοριστκοί Συνδυασμοί		39(A)	64(A)	39(A)	64(A)	37(A)	62(A)
Απαίτηση Οπλισμού	A_s (cm^2)	35.94	29.27	6.93	5.32	18.84	21.04
ανά Παρειά/Καθοριστ.Συνδ	(cm^2)						

ΕΛΕΓΧΟΣ ΣΕ ΔΙΑΤΜΗΣΗ						
ΧΩΡΙΣ ΙΚΑΝΟΤΙΚΗ ΜΕΓΕΝΘΥΣΗ ΤΕΜΝΟΥΣΑΣ	Αρχή			Τέλος		
Τέμνουσα Σεισμού (kN)	minV_{sd}	maxV_{sd}	ζ	minV_{sd}	maxV_{sd}	ζ
	30.3	872.6	0.03	18.5	860.8	0.02

		ΑΡΧΗ (Κρίσιμο)		ΑΝΟΙΓΜΑ		ΤΕΛΟΣ (Κρίσιμο)	
Τμήματα Δοκού l(m)		0.60		0.10		0.60	
Συμμετοχή Σεισμού		Όχι	Ναι	Όχι	Ναι	Όχι	Ναι
Τέμνουσα Υπολογισμού	V_{Ed} (kN)		872.6		862.3		860.8
Στρ. Ροπή Υπολογισμού	T_{Ed} (kNm)		7.0		7.0		7.0
Αντοχή ΧΩΡΙΣ οπλισμό	$V_{Rd,c}$ (kN)		151.6		95.0		121.8
Αντοχή θλιβόμενων διαγώνιων	V_{Rdmax} (kN)		857.0		857.0		857.0
Στρεπτική Αντοχή θλιβόμενων διαγώνιων	T_{Rdmax} (kNm)		143.4		143.4		143.4
$T_{Ed} / T_{Rdmax} + V_{Ed} / V_{Rdmax}$ <= 1.0			1.1		1.1		1.1
Καθοριστικοί Συνδυασμοί			39(A)		39(A)		39(A)
Απαιτουμενη Διατομη							
Συνδετήρες,Δισδιαγώνιοι	A_{sw}/s (cm^2/m)	κ39.31		κ38.85		κ38.79	
Πρόσθετα Λοξά	(cm^2)						

Having identified the failures, you should make the necessary modifications.
Select the beam with the left mouse button on the floor plan. The list of "Properties" opens on the left and "More" opens the geometry of the cross-section.

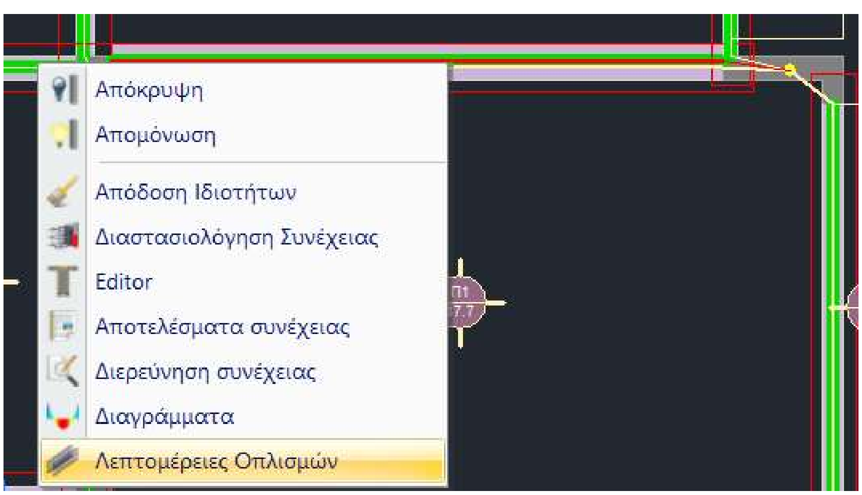
EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'



Enlarge the cross-section

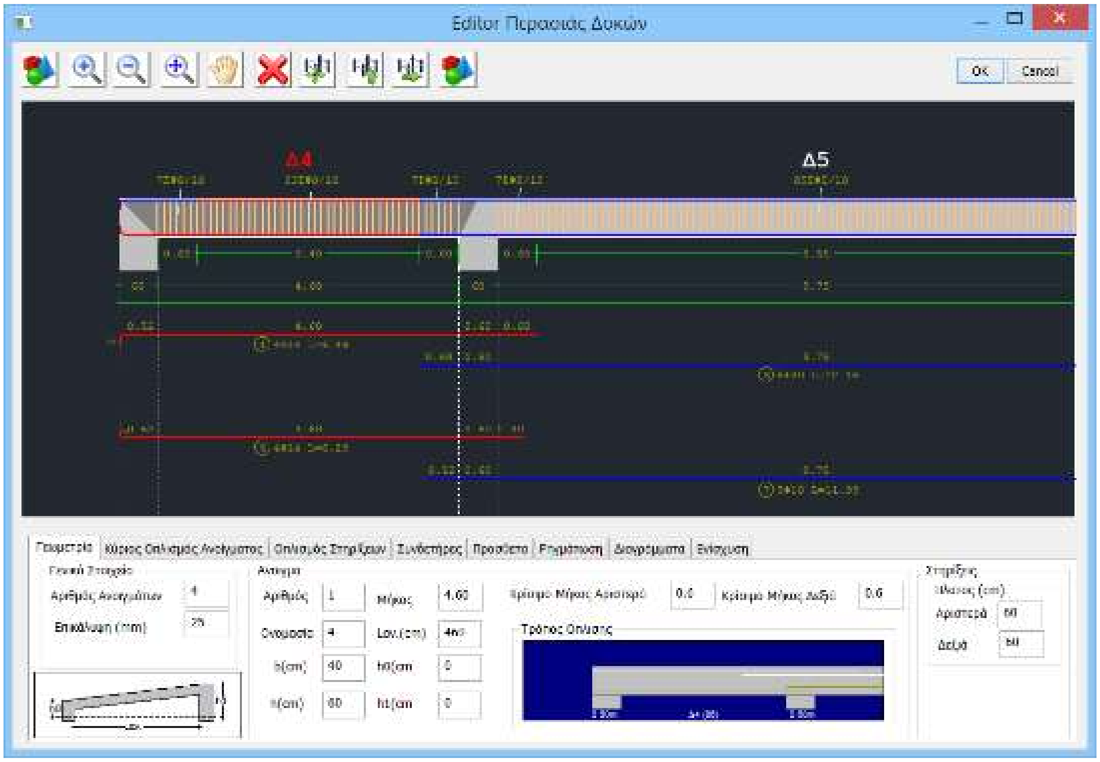
and

resize the with continuity.



By right-clicking on the beam member and selecting "**Armouring Details**", the window of details concerning the armouring of the continuity as derived from the dimensioning opens, depicting the continuity according to the local axes.

⚠ Attention, beams belonging to the same continuum must have the same direction.

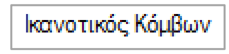


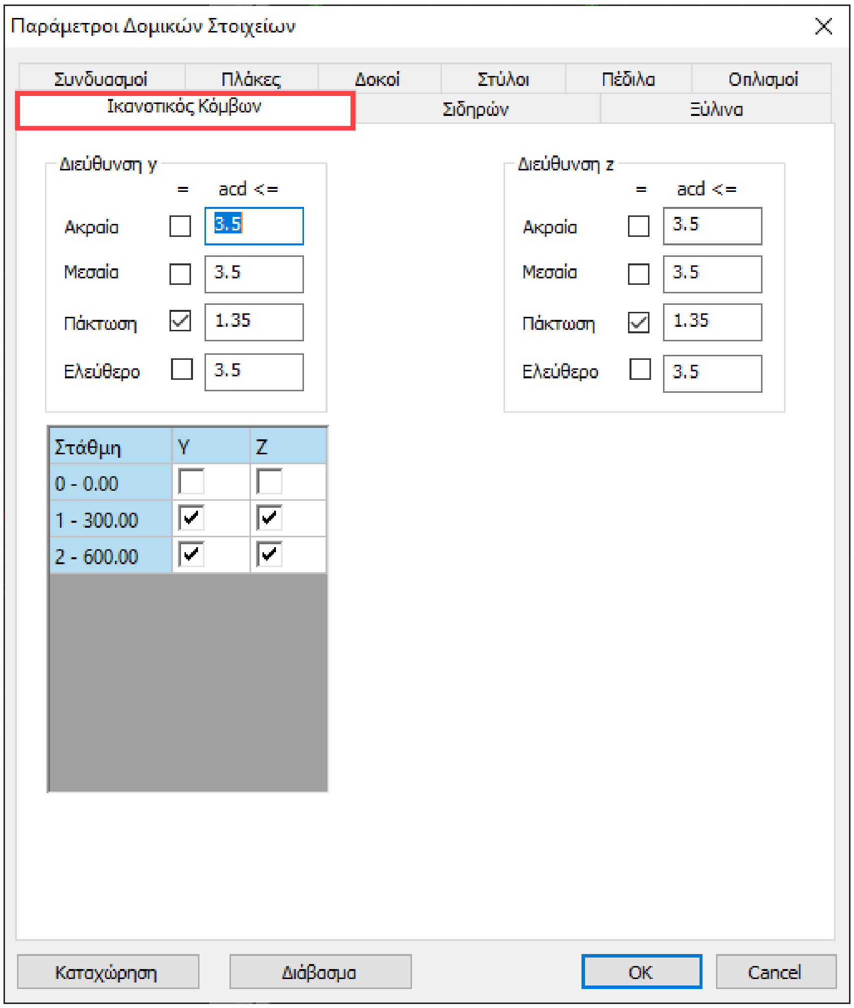
Here you can make all changes to the main and secondary armament.

⚠ *Detailed instructions on how to use this command can be found in the corresponding user manual.* (***See Instructions for Use Chapter A "Details of Beam Armouring"***)

7.4 How to do the satisfaction check:

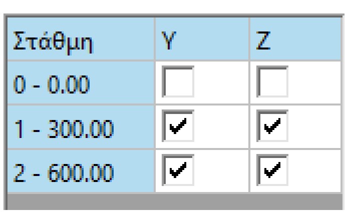
Having set in the "**Node capacity**" section of the **Parameters**





specify by x and z the parameters to be used in the satisfaction check.

At the bottom



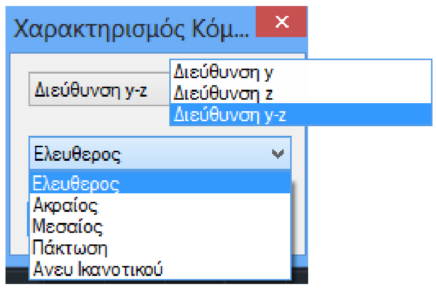
you select the level or levels and the direction where you wish to perform the level check.

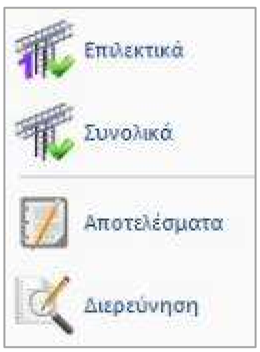
Specify the upper bound on the satisfactory node enlargement factor acd.

In general, the value of $_{acd}$ is defined to be less than or equal to the value of the seismic behaviour coefficient q.
For the footing positions of the columns, $_{acd}$ shall be taken as 1,35.
Check the corresponding option and enter the value you want.
If you do not check any option, the program will take into account the value of acd it will calculate.

OBSERVATION:
The definition of the node type will then be done with the "Node Characterization" option.
Not "Node Characterization" by the user means that all nodes are taken as free in both directions, except for the packed nodes.

With the "**Node characterization**" option, you specify the type of node per direction.





The solution of the satisficer is done either Selectively or Overall per level.

Select the **Results** command and a column or wall node to open the test results file for that node for each seismic combination and direction.

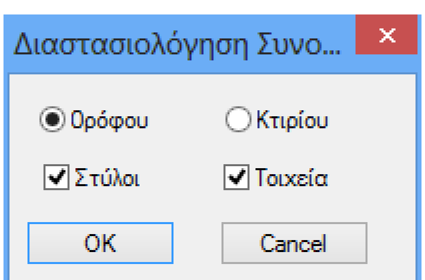
Κόμβος = 33
Στύλος Κάτω = 19
Στύλος Πάνω = 33

ΣΥΝΔ.	SMRby	SMEby	acdy calc	acdy	SMRbz	SMEbz	acdz calc	acdz
3	142.500	177.826	1.042	1.042	140.600	24.609	7.427	3.500
4	142.500	104.563	1.772	1.772	140.600	18.346	9.963	3.500
5	142.500	176.740	1.048	1.048	140.600	23.446	7.796	3.500
6	142.500	103.477	1.790	1.790	140.600	17.183	10.637	3.500
7	142.500	181.446	1.021	1.021	140.600	28.486	6.416	3.500
8	142.500	108.183	1.712	1.712	140.600	22.223	8.225	3.500
9	142.500	180.360	1.027	1.027	140.600	27.323	6.690	3.500
10	142.500	107.097	1.730	1.730	140.600	21.060	8.679	3.500
11	142.500	121.847	1.520	1.520	107.600	1.066	131.240	3.500
12	142.500	48.584	3.813	3.500	107.600	7.094	19.718	3.500
13	142.500	122.933	1.507	1.507	107.600	0.905	154.592	3.500
14	142.500	49.670	3.730	3.500	107.600	5.931	23.585	3.500
15	142.500	125.467	1.476	1.476	107.600	3.046	45.921	3.500
16	142.500	52.204	3.549	3.500	107.600	3.217	43.483	3.500
17	142.500	126.553	1.464	1.464	107.600	4.209	33.232	3.500
18	142.500	53.290	3.476	3.476	107.600	2.054	68.107	3.500
19	108.600	48.584	2.906	2.906	140.600	7.094	25.766	3.500
20	108.600	121.847	1.159	1.159	140.600	1.066	171.490	3.500
21	108.600	49.670	2.842	2.842	140.600	5.931	30.819	3.500
22	108.600	122.933	1.148	1.148	140.600	0.905	202.005	3.500
23	108.600	52.204	2.704	2.704	140.600	3.217	56.818	3.500
24	108.600	125.467	1.125	1.125	140.600	3.046	60.004	3.500
25	108.600	53.290	2.649	2.649	140.600	2.054	88.995	3.500
26	108.600	126.553	1.116	1.116	140.600	4.209	43.424	3.500
27	108.600	104.563	1.350	1.350	107.600	18.346	7.624	3.500

7.5 How to size poles and walls:

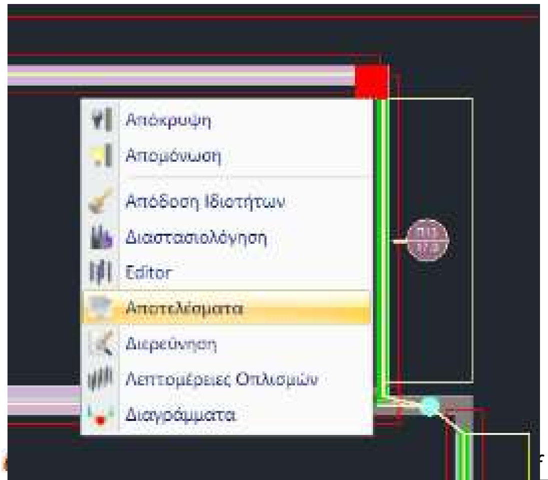
The "Columns" field contains the commands for Dimensioning, Reinforcement Check and Column and Wall Results.
(**See Internal Usage Chapter 9 "Sizing"**)

Select the command "***Arming Control > Total***" to perform a total sizing of the columns and/or walls in the design, per floor or throughout the building.
Selecting the command displays the following dialog box:



where you choose whether to dimension the columns and/or the walls of the floor or the whole building.

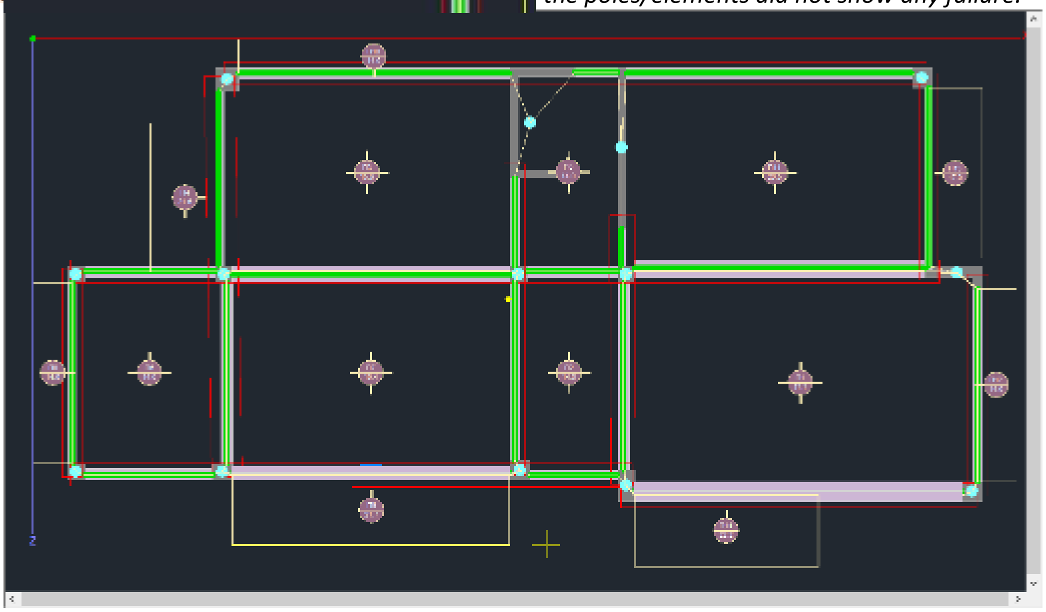
After sizing, coloured dots appear in the centres of the poles. The colour changes according to the <u>type of failure</u> as follows:



- ❖ **Red**. Failure from biaxial bending. Exceeding maximum 4% reinforcement percentage. Dense Connectors.
- ❖ **Pink**. Failure by shear/torsion or by exceeding the plasticity limit. In the results you can see the failure ratio.
- ❖ **Cyan**. The pole was dimensioned without any problem.

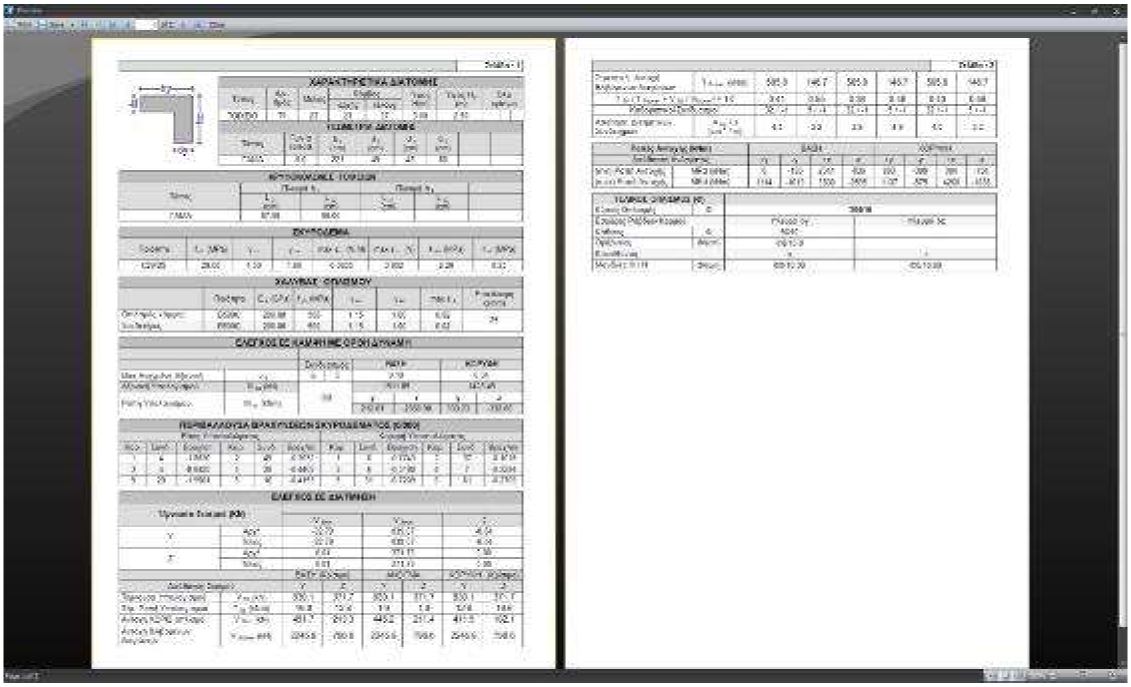
Initials indicating the type of failure also appear on the pole.

f the poles/elements did not show any failure.

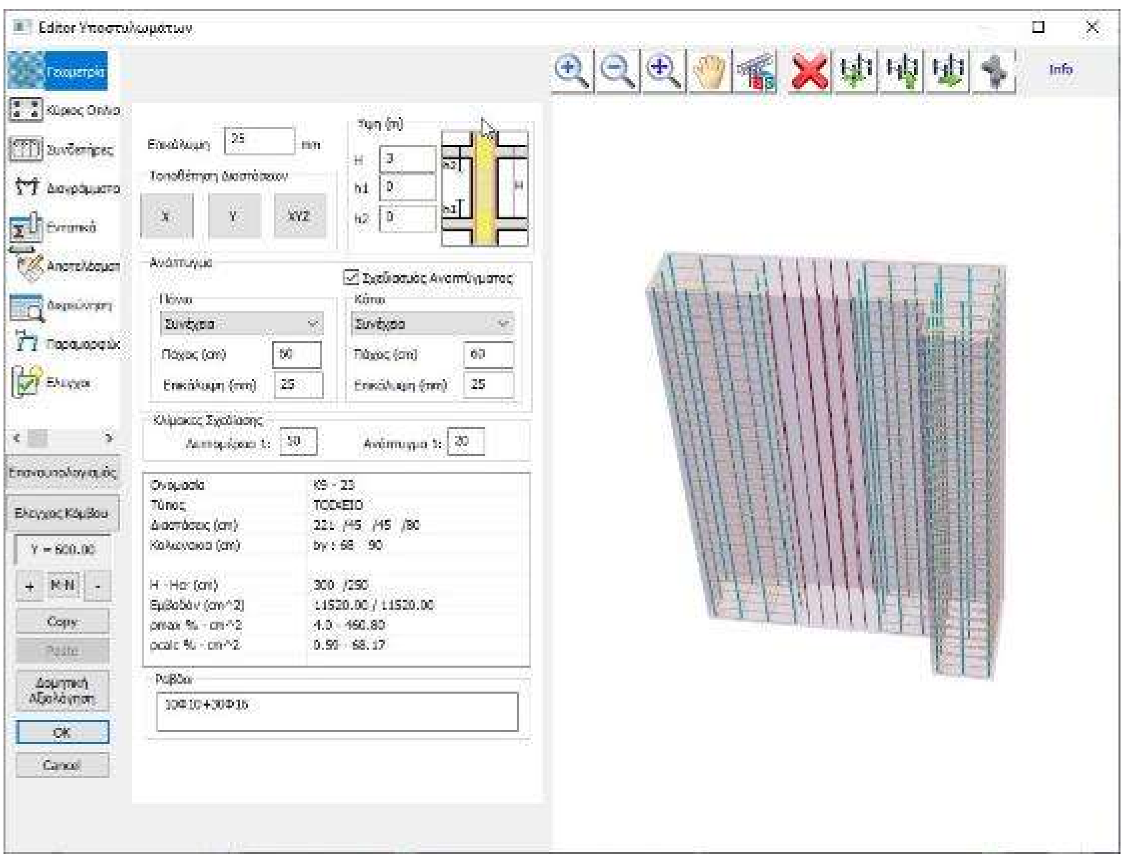


Right-clicking on the cross-section of the column opens a list of commands related its dimensioning. Select "**Results**" to read the checks from the summary results file that opens:

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'



Select "**Armouring Details**" to open the window of details concerning the armouring of the column/element as derived from the dimensioning:

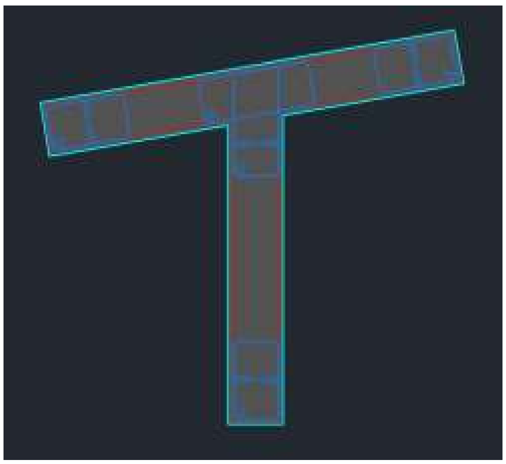


EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

With this option you can edit the reinforcement of the column or wall, perform local checks and recalculate the interaction diagrams, within an integrated calculation and design environment.

OBSERVATION:

It is now possible to change the vertical and horizontal trusses in the walls, a very useful function especially in the valuation of existing buildings.
For the vertical bars the change is made in the Reinforcement Details of the columns with the familiar bar correction tool. With these changes the results in the design book are automatically updated and obviously these bars are also taken into account in the overall strength of the wall.
The vertical bars are listed in the results by y and z direction. There is the possibility of two entries per direction as in the following table.

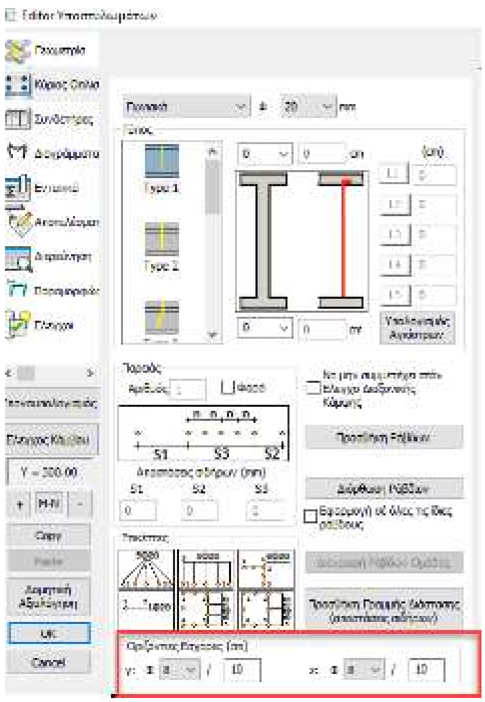


Where by y we can have differentiation of the vertical trunk bars

```
|ΚΥΡΙΟΣ ΟΠΛΙΣΜΟΣ               | 43Φ14                                           |
|ΚΑΘΕΤΕΣ ΕΣΧΑΡΕΣ ΡΑΒΔΩΝ ΚΟΡΜΟΥ | 2Φ10+ 2Φ10 (πλευρα by)   5Φ10     (πλευρα bz)  |
|ΟΡΙΖΟΝΤ.ΕΣΧΑΡΕΣ ΡΑΒΔΩΝ ΚΟΡΜΟΥ |Φ 8/10.0 (πλευρα by)      Φ 8/10.0 (πλευρα bz)   |
|Μανδυες Φ / Hcr.      (cm)    |(y)Φ 8/10.00              (z)Φ 8/10.00          |
```

Regarding the modification of the horizontal armament, a new field was added to the editor in the "Main Armament" section to change it.

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'



The definition of horizontal bars is done per y and z direction. The directions are only meaningful when there are T or C-shaped columns. For rectangular columns, define by y or z depending on the direction of the wall.

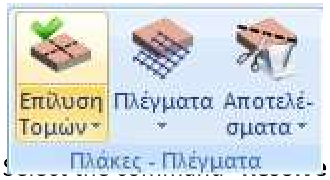
```
|-------------------------------------------------------------------------|
|ΚΥΡΙΟΣ ΟΠΛΙΣΜΟΣ              | 43Φ14                                     |
|ΚΑΘΕΤΕΣ ΕΣΧΑΡΕΣ ΡΑΒΔΩΝ ΚΟΡΜΟΥ | 2Φ10+ 2Φ10 (πλευρα by)  5Φ10    (πλευρα bz) |
|ΟΡΙΖΟΝΤ.ΕΣΧΑΡΕΣ ΡΑΒΔΩΝ ΚΟΡΜΟΥ |Φ 8/10.0 (πλευρα by)    Φ 8/10.0 (πλευρα bz) |
|Μανδυες Φ / Hcr.      (cm)   |(y)Φ 8/10.00            (z)Φ 8/10.00        |
|Περισφιγξη ωwd               |(y)απ.: 0.08 υπ.: 0.17  (z)απ.: 0.08 υπ.: 0.17 |
===========================================================================
```

OBSERVATION:

Recall that the horizontal trunk bars are obtained by testing the trunk in shear. If the need for denser bars than the column connectors arises, these bars are indicated. Otherwise, the horizontal bars shall be placed the same as the column connectors.

⚠ *Detailed instructions on how to use this command can be found in the corresponding user manual.*
*(**See Instructions for Use Chapter B "Details of Pillar Armament"**)*

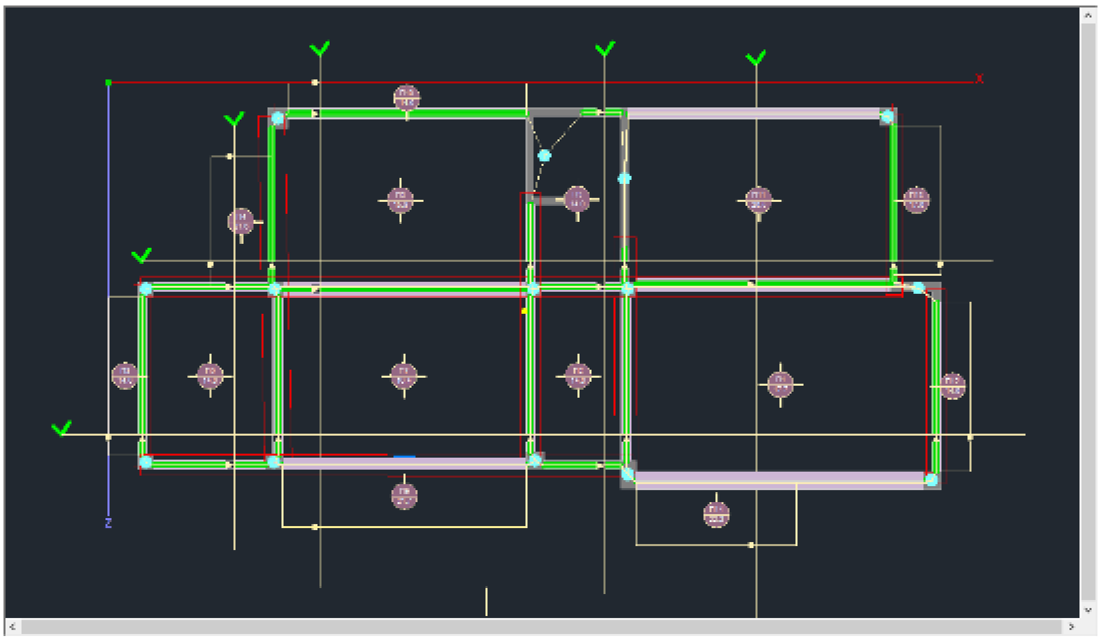
7.6 How to size the plates:



The "Slabs-Mesh" field contains the commands for Sizing, Reinforcement Check and Slab Results.

Sections > Total" for total resolution of all sections of the specified level.

By solving the sections, the intensive quantities are calculated and the plates are dimensioned. The program calculates <u>tensile</u> (E) reinforcement (Fe), <u>compressive</u> (TH) reinforcement (Fe') in $cm^{(2)}$. Similarly, it calculates reinforcement bars in the spans, <u>distribution</u> reinforcement in the amphibious slabs, <u>separation</u> reinforcement, <u>additives</u> in the supports and <u>connectors</u> if the slabs are ribbed beams.



OBSERVATION:
In the new versions of SCADA Pro has been added the Deformation Control on the plates.

The deformation check is based on 7.4.2 and 7.4.3 of EC2 and is presented at the end of the results for each plate and if the scenario is not ECOS. The results of the two checks are shown separately.

ΥΠΟΛΟΓΙΣΜΟΣ ΠΑΡΑΜΟΡΦΩΣΕΩΝ (EC2 παρ.7.4.2 & παρ.7.4.3)								
l/d	l/d επιτρ.	Επάρκεια	Προτειν.ελάχ. πάχος hs(mm)	Max. M (kNm)	du1 (mm)	a	l/a (επιτρ) (mm)	Επάρκεια
34.59	80.10	NAI	77	-7.64	0.42	250	18.40	NAI

In the first check, a minimum proposed thickness is obtained, but it cannot be proposed in the initial identification of the plate because its calculation requires its reinforcements.

In the calculation of the first check, no intensive quantities are used, while the second check is carried out with the functional combination(s).

7.7 How to size the sandals:

The field "Peds" contains the commands for sizing the peds and the corresponding results.

Select the command "**Check Arming>Total**" to perform a total sizing of the level pedestals. Select the command and all the level pedestals are sized.
The node of the pedestal, depending on the type of failure, shall be painted in the corresponding colour according to the following

 The skirt was sized and armed without any problems.

The skirt missed. The type of failure is also indicated as a symbol above failure indication. The failure indications are respectively the letter "Z" which means failure at limit load, the letter "e" which means failure due to load eccentricity and the letter "s" which means exceeding the developing stress.

- A prerequisite for the dimensioning of the pedestals is dimensioning of the level 1 poles.

OBSERVATION:

In some cases it is suggested that the dimensioning of the footings be done with combinations of statics because the dynamic quantities are unmarked and not suitable for the dimensioning of the foundation.

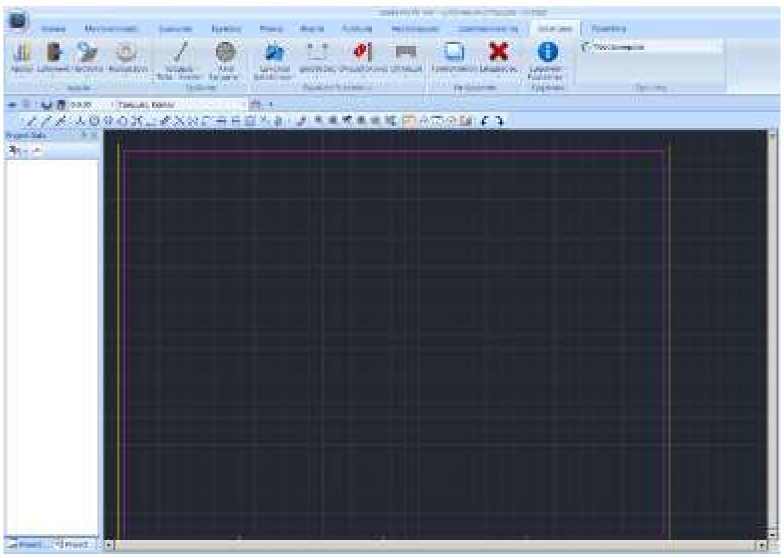
As is well known, seismic intensities derived from dynamic analysis are unlabeled because they result from superposition of the eigenmodal responses. In the diagrams wherever there is a necessity to superimpose them, they are always used with positive values. And for the dimensioning of the elements there is no problem because the combinations include them with both signs but in cases such as the dimensioning of the pedestal where magnitudes are used for each combination from each element the situation may turn out unfavourable.

For this reason I recommended you to solve the sandals with static combinations.

WOODEN DOORS

After completing the dimensioning of the structure and the modifications of the reinforcement through the "Editor" and "Reinforcement details" commands for concrete designs, or the creation of the connections for metal ones, within the Timber Formwork Module you enter, modify and finally create the drawings of the formwork and its details.

By selecting the "Wooden Forms" section, the drawing paper frame is displayed on the desktop.



8.1 How to import formwork and beam expansions into the design environment:

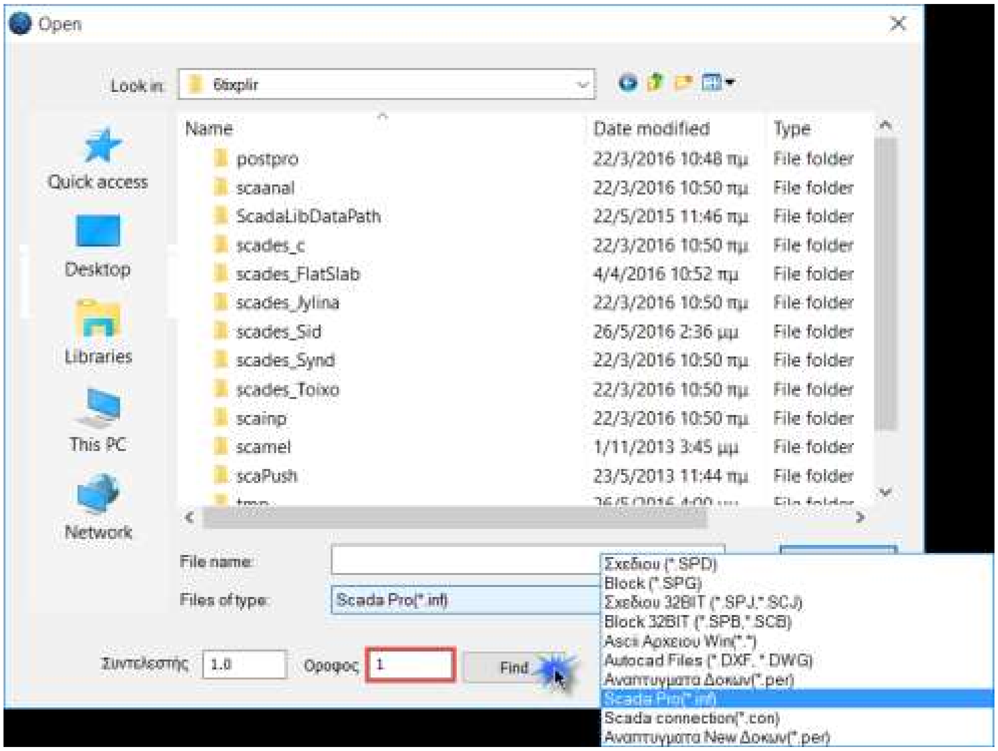
The Import command opens the window for selecting the study folder.

You choose:

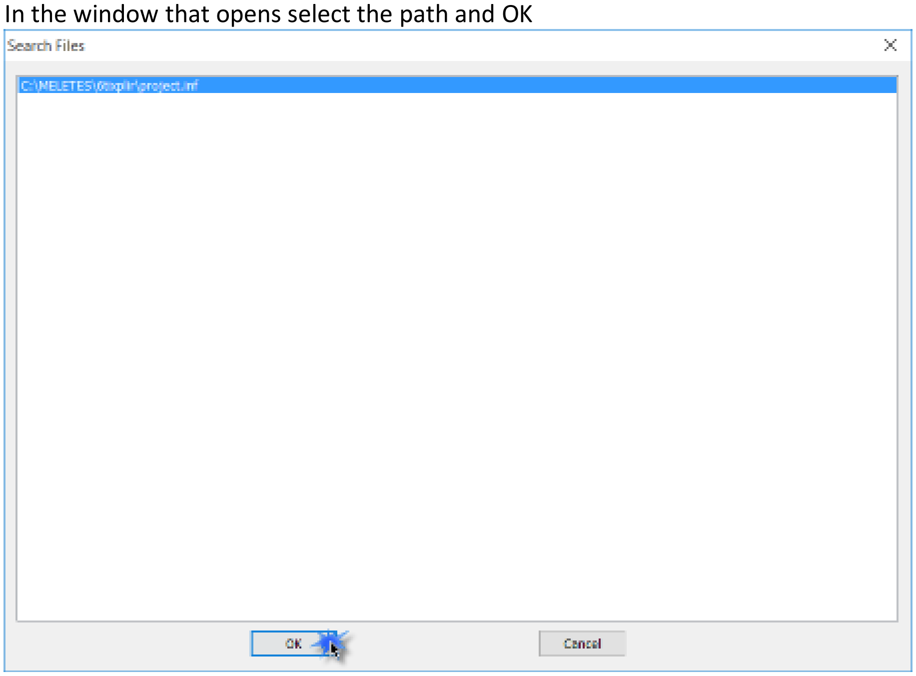
- the type of design from the **Files of Type**
- the number of the floor and
- the coefficient

you press the **Find**
command.

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

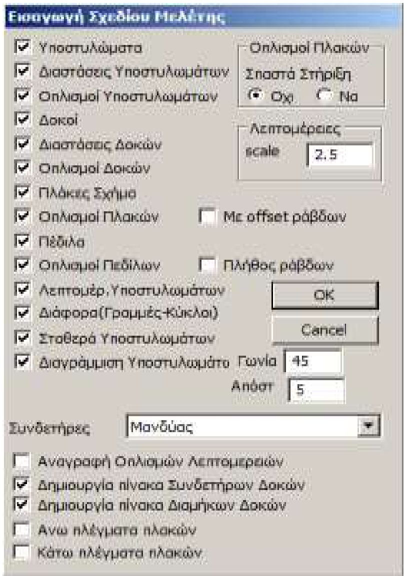


In the window that opens select the path and OK



The following dialog box appears on the screen from which:

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

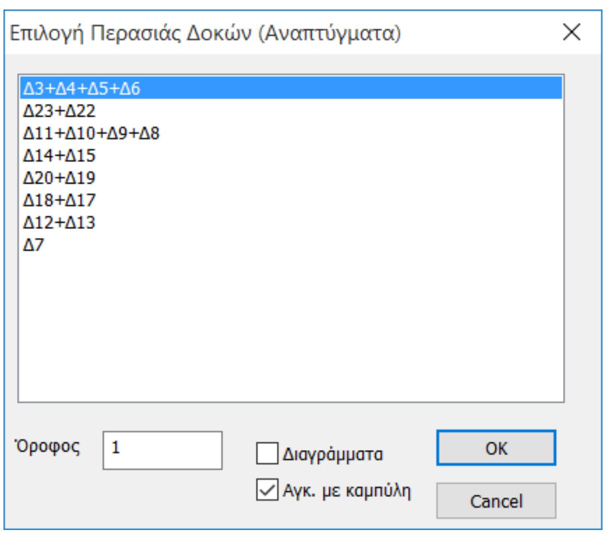


- You select the entities to be inserted in your design by activating with "v" the corresponding checkboxes.
- In "**Plate reinforcements**" you will select whether the additional bars of the plate supports are to be designed broken or not.
- In the "**Details - Scale**" field you will enter the scale factor for the details of the columns to be inserted on your paper.
- Example : If you are drawing a 1:50 scale woodcut and 1:20 scale post details, you would enter the factor 50/20 = 2.5.

From file Beam Expansions (.per) :*

Enter in your design the reinforcement expansions for the beam span that you will choose from those available in our study.
This option is for expansions created with the existing beam editor, while the "New Beam Expansions" option with the same format (*.per) is for expansions created with the new "Armor Details" editor.

Selecting the Beam Expansions (old and new) the path in Find takes you to a new window to select the passes one by one.



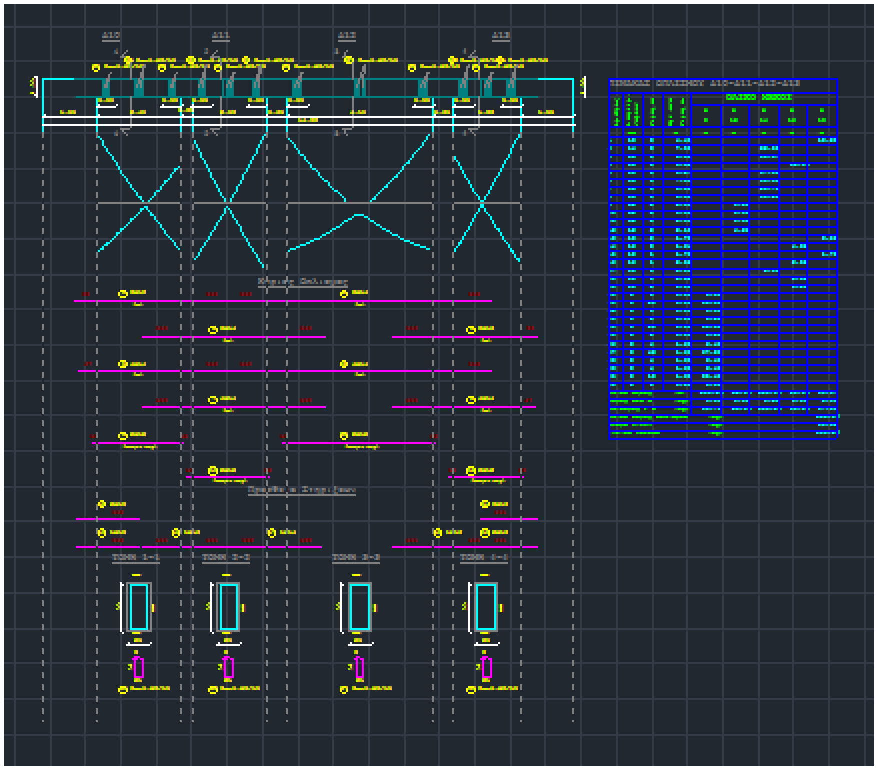
In the "Floor" position, enter the number of the level where the terrace is located, of which you want to draw the developments.
Activating with "v" the indicator:

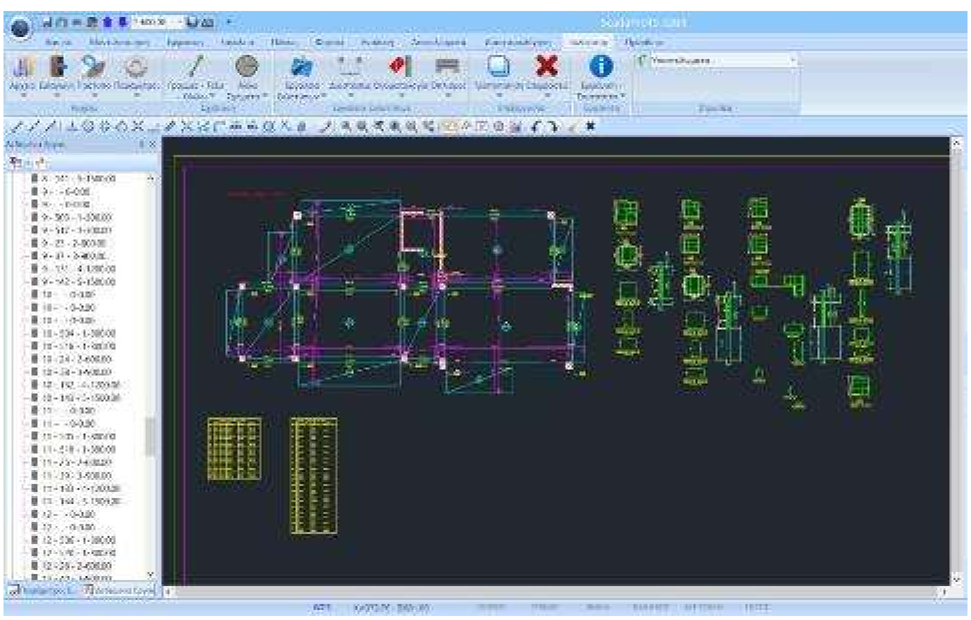
- "Diagrams": the expansion you enter will be accompanied by the corresponding torque diagram.
- "Curved anchorage": the anchorages will be closed with a curve.

You select one of the available passes that open and by pressing the "OK" button you are invited to place the design of the growths on your paper.

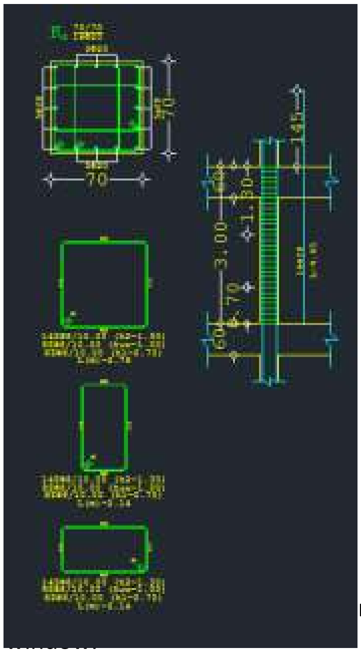
EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'

Point to the insertion point and insert the drawing of the selected level, repeating the process for all levels and all details.





8.2 How to import into the drawing environment detailed post details with the possibility to modify them directly from within the editor:



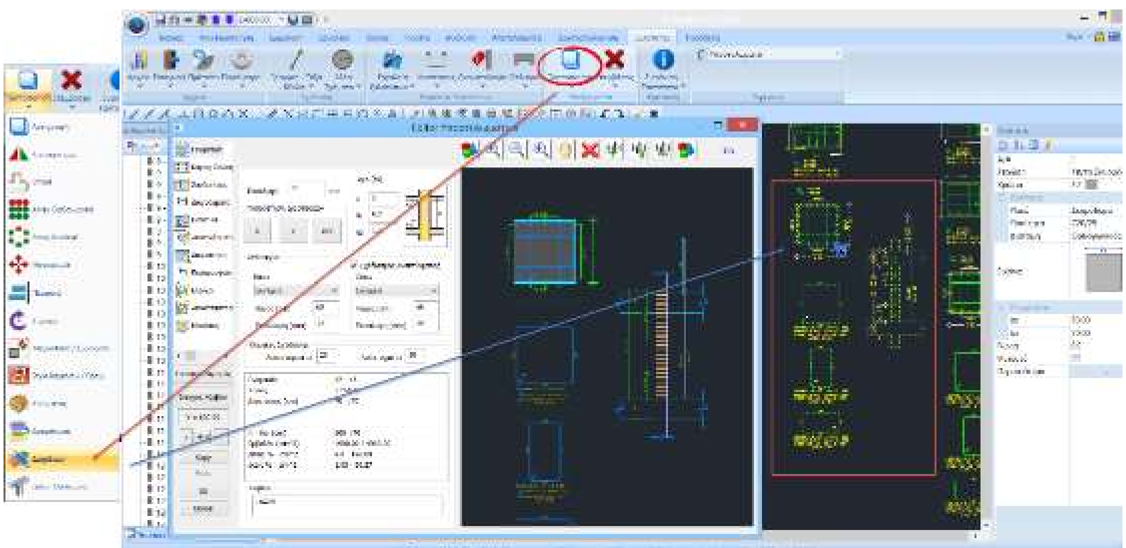
A prerequisite for the introduction of the detailed details of columns and walls within the design environment is:

* the "Armament Details" command for the corresponding poles and walls has been selected beforehand, and

* in the respective windows to press the "OK" button.

Then, the import of the design plan "project.inf" will include the detailed details of columns and walls.

mmand allows the detail to be corrected directly within the editor

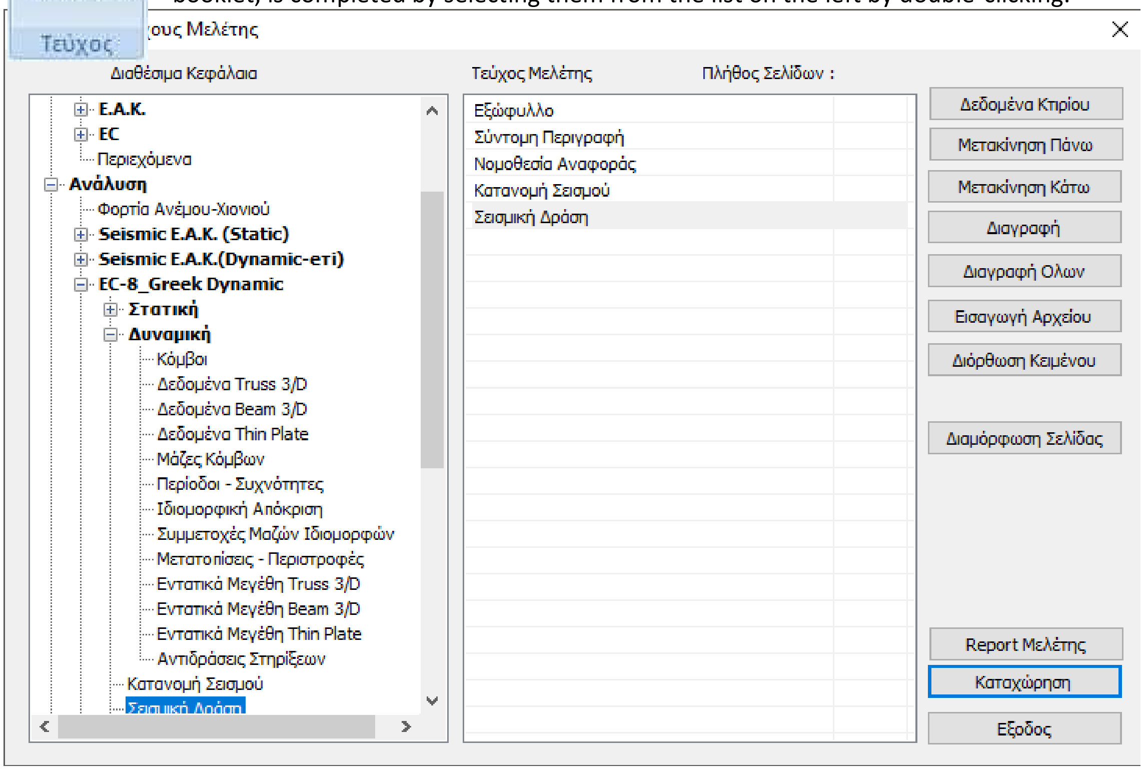


Select the "**Fix**" command and left click on the detail. The corresponding editor window opens automatically where you can make the necessary modifications. By pressing the OK button you save the changes which automatically update both the drawing and the issue.

COPY

9.1 How to create the study issue:

To create the study booklet open the "Extras" section and select the Print command. In the "Create Study Sheet" dialog box, the list of chapters available for printing is displayed on the left. The list on the right, with the chapters to be included in the booklet, is completed by selecting them from the list on the left by double-clicking.



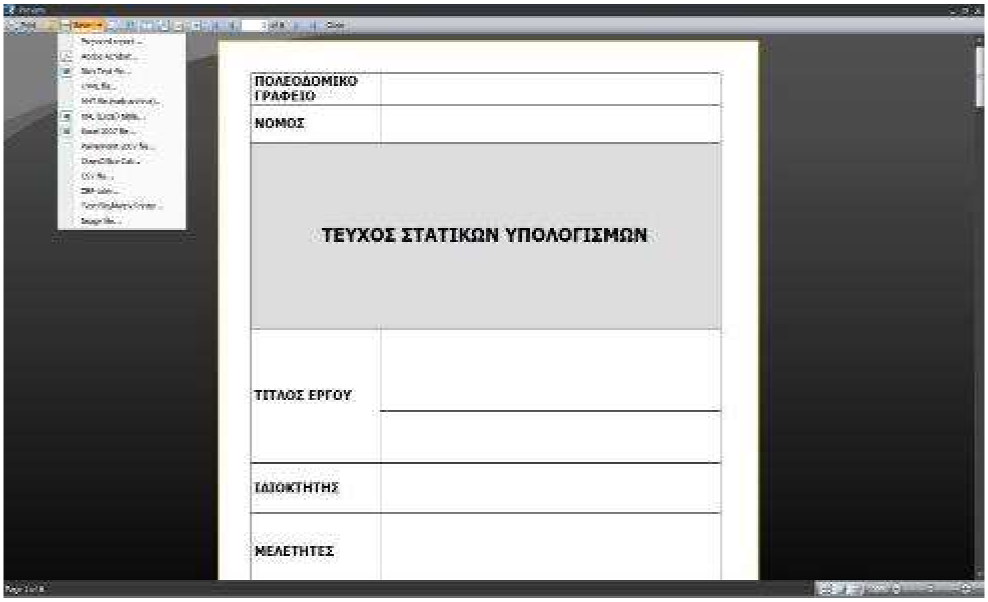
Click on the "Study Report" button to display the preview of the issue.

OBSERVATION:

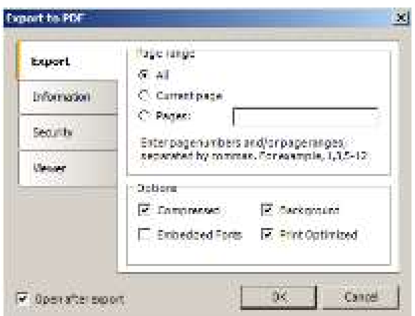
In the new version of SCADA Pro all the printouts of the study results booklet have been redesigned and implemented with modern tools in order to offer you a new tabular, easy-to-read study booklet with the addition of diagrams and images. You also now have a full preview of your issue as well as the ability to export and edit the file in more than ten different file formats including pdf, docx, rtf, xml, CSV, PowerPoint, etc.

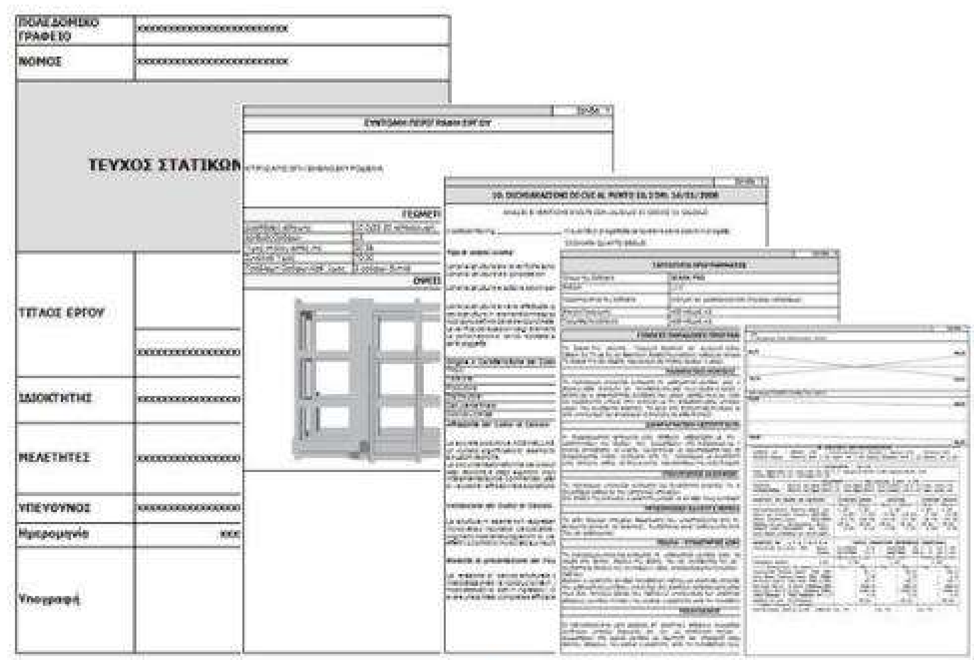
In addition, the ability to "break" the study book into individual sections has been added, a useful and practical feature mainly for the easy management of multi-page studies.

EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'



You can save the issue as a .pdf, or .doc file, .excel, .xml and edit it.





EXAMPLE 1: 'REINFORCED CONCRETE CONSTRUCTION STUDY'