SCADA Pro™
Structural Analysis & Design

Example 3
Non-standard Study
Metal Construction

ACE HELLAS
INTEGRATED SOLUTIONS

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

CONTENTS

FOREWORD **4**

INTRODUCTION **4**

THE NEW ENVIRONMENT **4**

1. GENERAL DESCRIPTION **6**

- *Geometry* *6*
- *Materials* *6*
- *Regulations* *6*
- *Cross sections* *7*
- *Loading - analysis assumptions* *7*
- *Comments* *7*

2. DATA IMPORT - MODELLING **8**

- *How to start a new study* *8*
- *New Study* *11*
- *Creating the Physical Model of the construction* *11*
- *Setting levels* *12*
- *Introduction of auxiliary drawing and physical sections* *13*
- *Copying layers and configuring floor plans* *17*
- *Foundation level configuration* *20*
- *Creating a mathematical model* *22*

3. IMPORTATION OF GOODS **24**

- *How to enter wind and snow loads in the automatic way based on Eurocode 1 24*

3.1.1 PARAMETERS 24
3.1.2 WALL TREATMENT 25
3.1.3 ROOF TREATMENT 28
3.1.4 WIND SHOW 30
3.1.5 SNOW SHOW 30
3.1.6 MATCHING MEMBERS 32
3.1.7 RESULTS 34

4. ANALYSIS **38**

- *How to create an analysis script:* *38*
- *How to run an analysis script:* *44*
- *How to create combinations of charges* *49*

5. RESULTS **52**

- *How to view charts and deformations* *52*
- *Body+ "Deformed Body"* *53*
- *Charts - Equalisation* *54*

6. DIMENSIONING OF METALLIC SECTIONS **55**

- *How to create dimensioning scripts* *55*
- *How to determine the parameters of the dimensioning of metal sections:* *56*
- *Dimensioning of steel sections:* *62*

6.3.1 Checking of metallic sections *62*
6.3.2 Buckling control of metallic sections *66*
6.1.1.1 USER CONSOLIDATION 73

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

7. CONNECTION SIZING **74**

How to dimension the connections of the metal members: *74*

8. FIELD SIZING **79**

How to size the sandals: *79*

THE NODE OF THE FIELD, DEPENDING ON THE TYPE OF FAILURE, SHALL BE PAINTED IN THE CORRESPONDING COLOUR ACCORDING TO THE FOLLOWING 79

9. PROMOTION **80**

10. DESIGN **82**

How to import the plans of the links *82*

11. COPY **86**

How to create the study issue: *86*

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

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) 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 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 is a program that is constantly upgraded, evolving and adapting. The technical department of ACE-Hellas in permanent cooperation with Metsovio Polytechnic University engaged in its continuous development and its adaptation based on new data, applications and needs. A "living organism" that matures!

INTRODUCTION

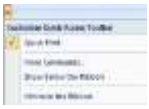
This manual was created to guide the designer in his first steps in the new SCADA Pro environment. It is divided into chapters and based on a simple example guide.
Each chapter contains information useful for understanding both the program commands and the procedure to be followed in order to insert, check and dimension a metal structure.

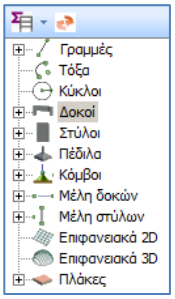
THE NEW ENVIRONMENT

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 of menus, toolbars and tables, and make it easier to find the command you want to use.

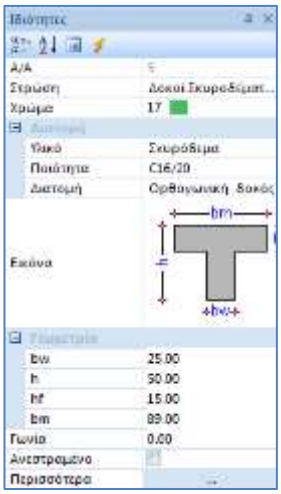
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

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 "quick access toolbar customization".





new SCADA Pro environment displays on the left side of the screen all entities of the construction categorized in a tree format either per level or for the whole building as a whole. This categorization 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 executed bidirectionally, i.e. the selection can be made graphically on the vector and the element will automatically appear in the tree with its properties on 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.



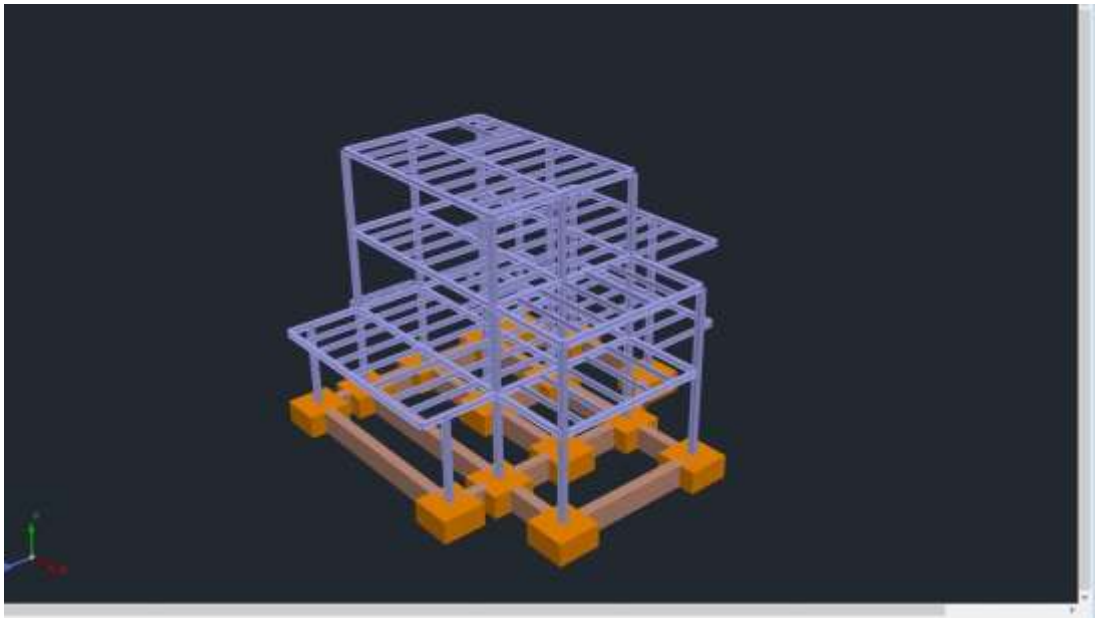
The "Properties" list on the right is automatically populated by selecting an element of the operator. It informs the user of its attributes, as well as allowing changes to them.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

1. GENERAL DESCRIPTION

1.1 Geometry

The metal structure under study consists of three levels with different floor plans
(1^{h} Level 3.80m, 2^{h} Level 6.70m, 3^{h} Level 9.50m).
The frame structure is formed by a steel frame, while the foundation is formed by individual reinforced concrete footings and connecting beams in both directions. For the complete geometry see the figure below:



1.2 Materials

For the construction of all the members of the structure, steel grade S275 (Fe430) will be used. The modulus of elasticity is E=21000kN/cm2 and Poisson's ratio n=0.30. The specific gravity of the steel is taken to be 78,5 kN/m3.

1.3 Regulations

Eurocode 0 (EC0, ENV 1990), for the determination of load combinations.
Eurocode 3 (EC3, ENV 1993), for the dimensioning of the metallic members of the carrier.
Eurocode 8 (EC8, EN1998), for seismic loads.
Eurocode 2 (EC2, EN1992), for the dimensioning of the foundation.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

4 Cross sections

Pillars:	HEA220
Main beams:	HEA200
Secondary Beams:	IPE200

5 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 ± (epicentric torsional moment loads resulting from the epicentric 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). To these, for this example, we will add the 3 below:
(8) S (snow)
(9) W0 (wind in direction x)
(10) W90 (wind in the z direction)

In the seismic analysis only the permanent and mobile loads are involved, not snow and wind loads which are taken into account in another "simple" static analysis scenario without earthquake (see Analysis).
The values of snow and wind loads are arbitrarily taken without the exact calculation as required by Eurocode 1, for the sake of simplifying the example.
On the contrary, the action coefficients ψ0, ψ1, ψ2 are determined exactly as prescribed by Eurocode_0.

6 Comments

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

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

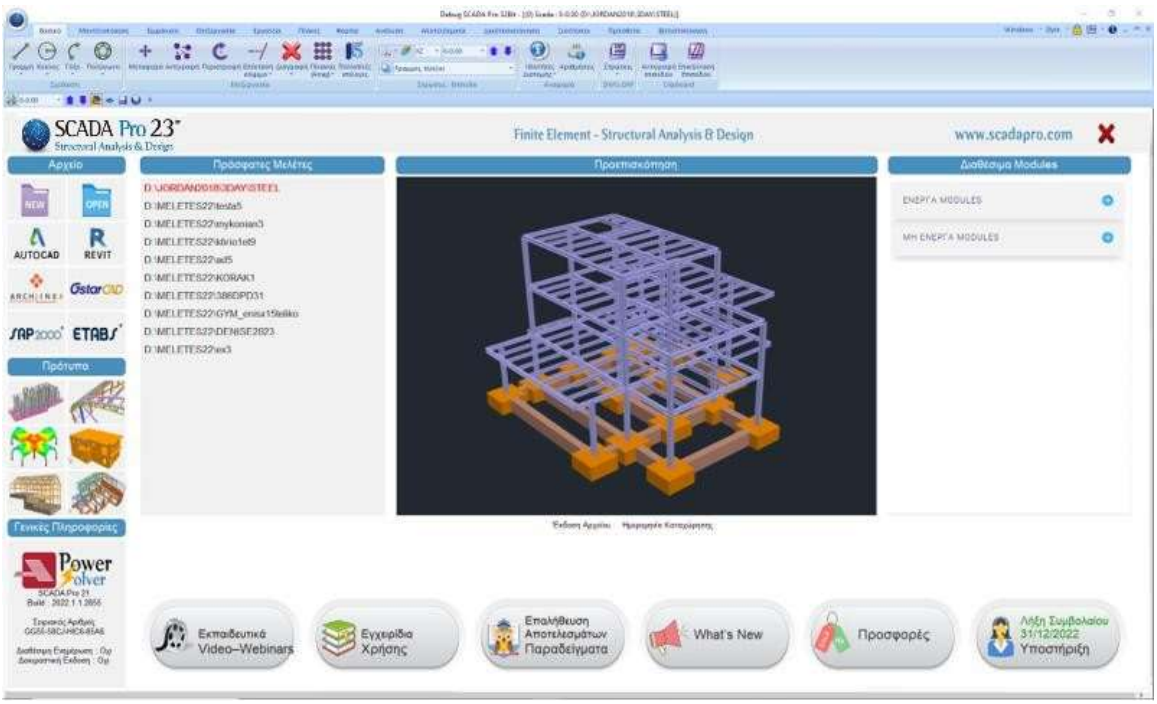
2. DATA IMPORT - MODELLING

How to start a new study:

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

> ⚠ In this example we will analyse in detail how to use
> **auxiliary dwg file** for the modelling of a metal carrier.

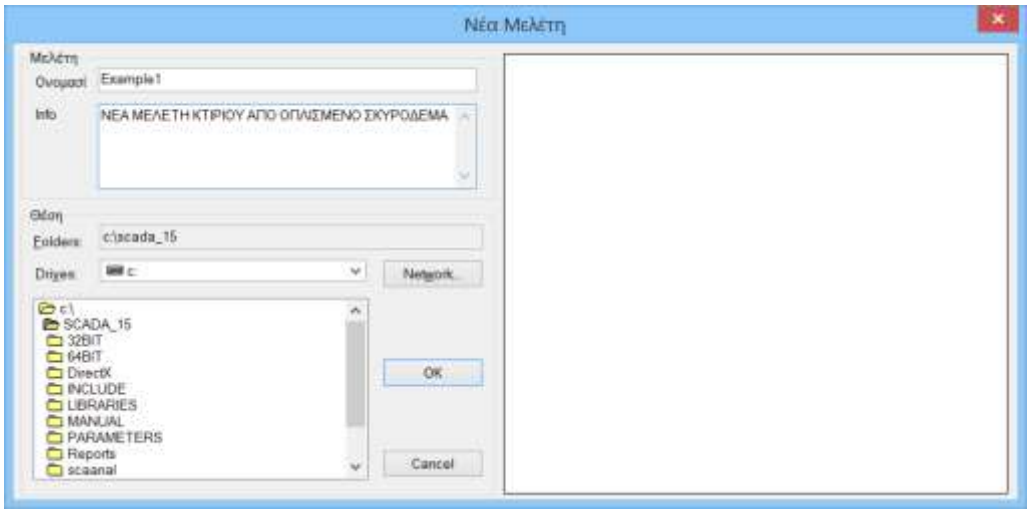
Upon opening the program, the startup window appears on the screen, which includes a set of commands to start the program:



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

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

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

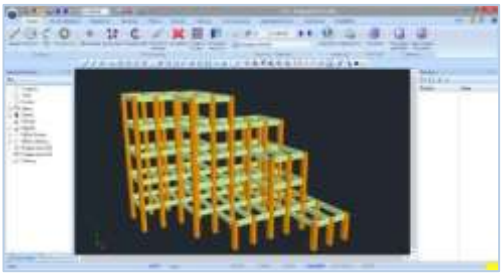


⚠ *The file name must consist of* ***a maximum of 8 Latin characters and/or numbers, without spaces and without the use of special characters (/, -,)*** *(e.g. ARXEIO1). The program automatically creates a folder where it enters all your study data. The "Location" of the folder, i.e. the place where this folder will be created, should be on the hard disk. We suggest you 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.

"**new**": usually used when there is no auxiliary file in electronic form. The startup is done in a blank interface. O
the designer starts by defining the stations and inserting the cross-sectionsusing the modelling commands and with the help of the cantilever pulls.

"**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.

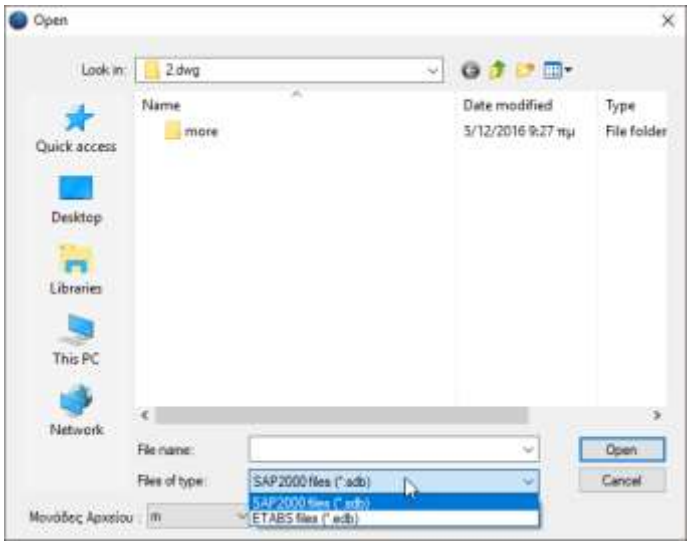


"**ArchlineXP**": read xml files from ArchlineXP.

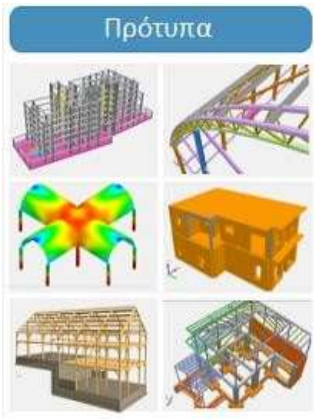
"**ETABS, SAP2000**": reading .edb & .edb files .sdb from the static programs ETABS & SAP2000 .

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

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)
For direct access to the "standard constructions" menu:



concrete metal

surface connections(IDEA StatiCa)

wooden masonry

OBSERVATION:

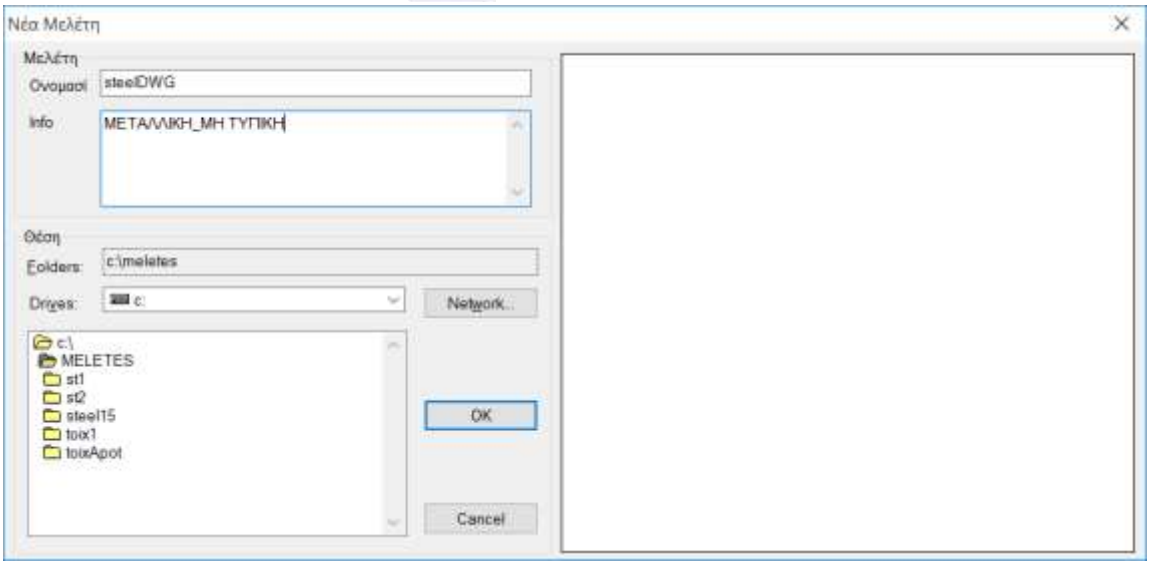
⚠ *The usual metal carriers are typical structures with continuous frames in one or both directions with a pitched roof. They may include mullions and purlins, windbreaks and frontal columns. In cases using the standard structures you manage to model the girder in one go! But also in cases of more complex structures, the use of standard structures can*
provides the background on which to build a complex institution.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

New Study

Select the relevant icon 

and in the dialog box



give a "**Name**" to the study. If desired, write some information about the study in the "**Info**" field and specify the location of the registration within the local disk.

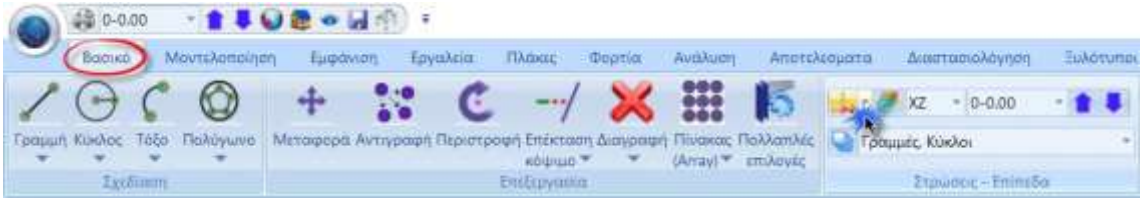
Creating the Physical Model of the construction

The layout of the commands of the new SCADA Pro facilitates the user and guides him through the study. Starting from the "Basic" for the definition of the stations, the "Modeling" for the introduction of the physical cross-sections, "Tools" for the necessary interventions, up to "Dimensioning" and "Design", you manage the elaboration of each study. In this example you will see the path followed, and many of the commands and tools of the program, in order to better educate the user.

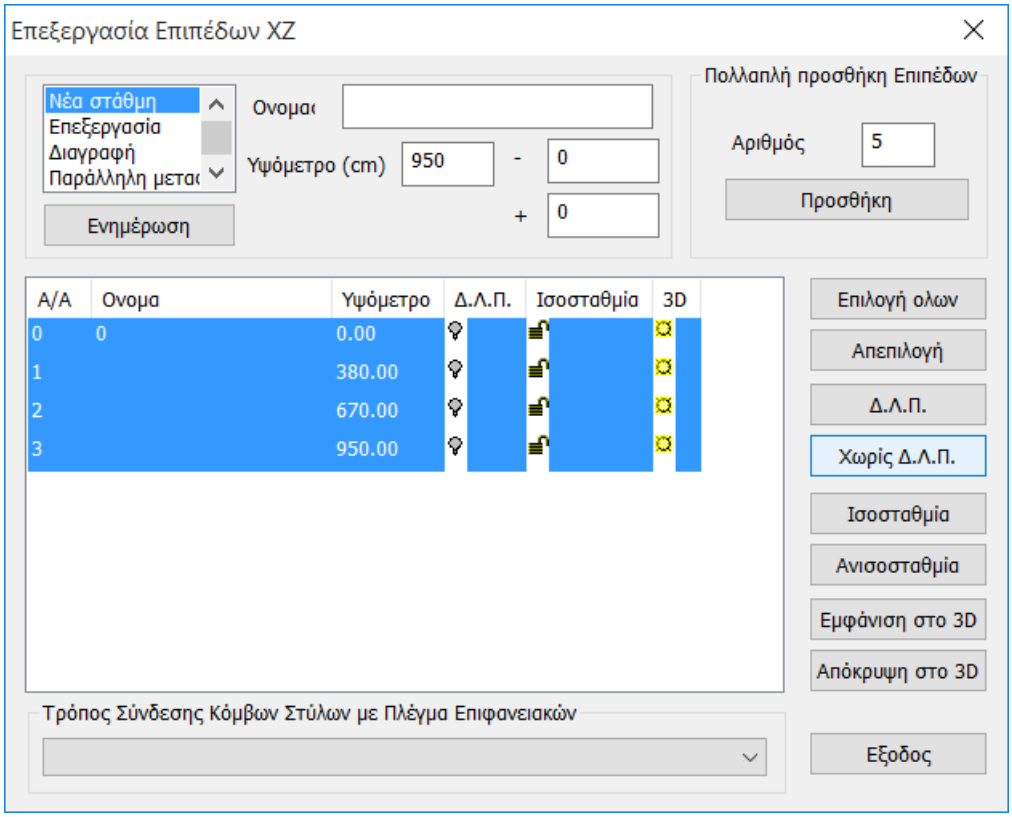
In detail, to create the Physical Model, the following steps are followed:

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

Setting levels



Press the  button and in the dialog box set the three levels of the anode by entering the altitude of each level and pressing "Update".
Then, with "Select all" and "Without I.D.S." disable the diagrammatic function of the plates in cases where they are absent.

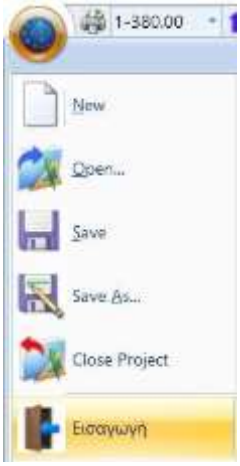


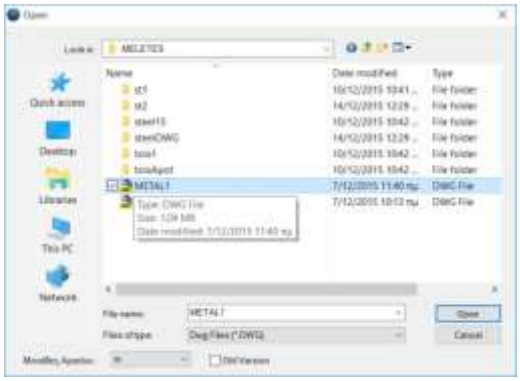
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

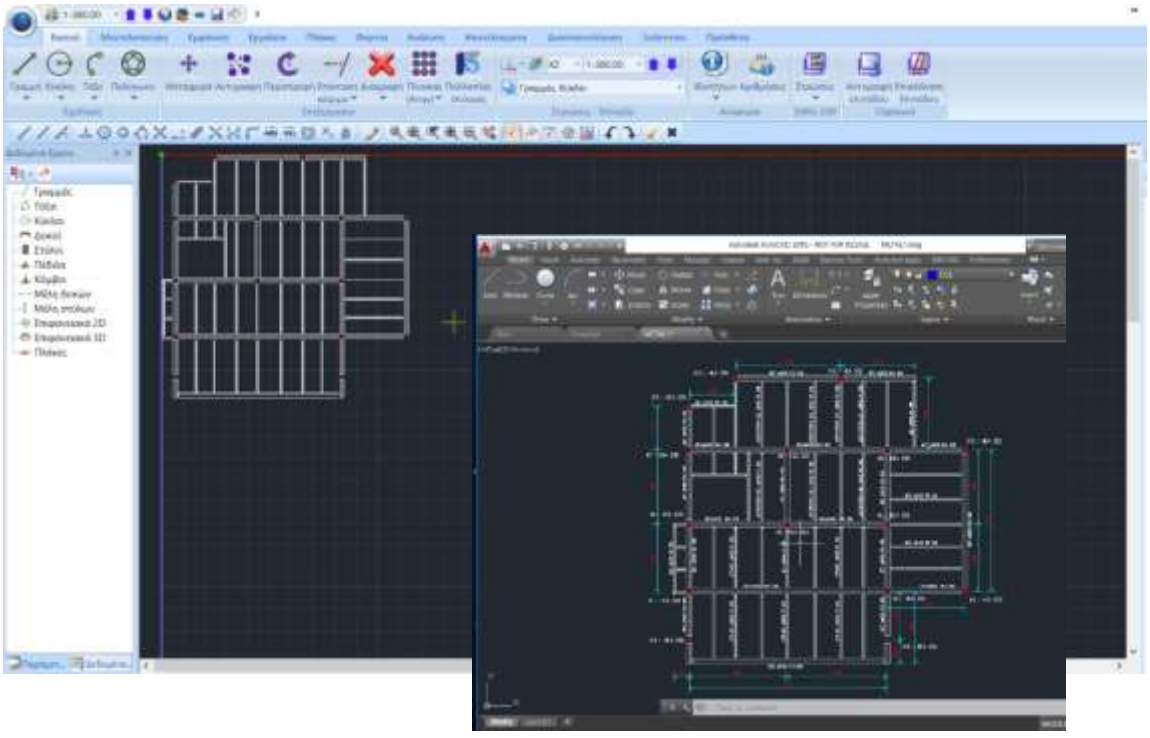
Introduction of auxiliary drawing and physical sections

With the help of the arrows 

, activate level 1 where you will insert the auxiliary file.
The import of the utility is done via the "Import" command

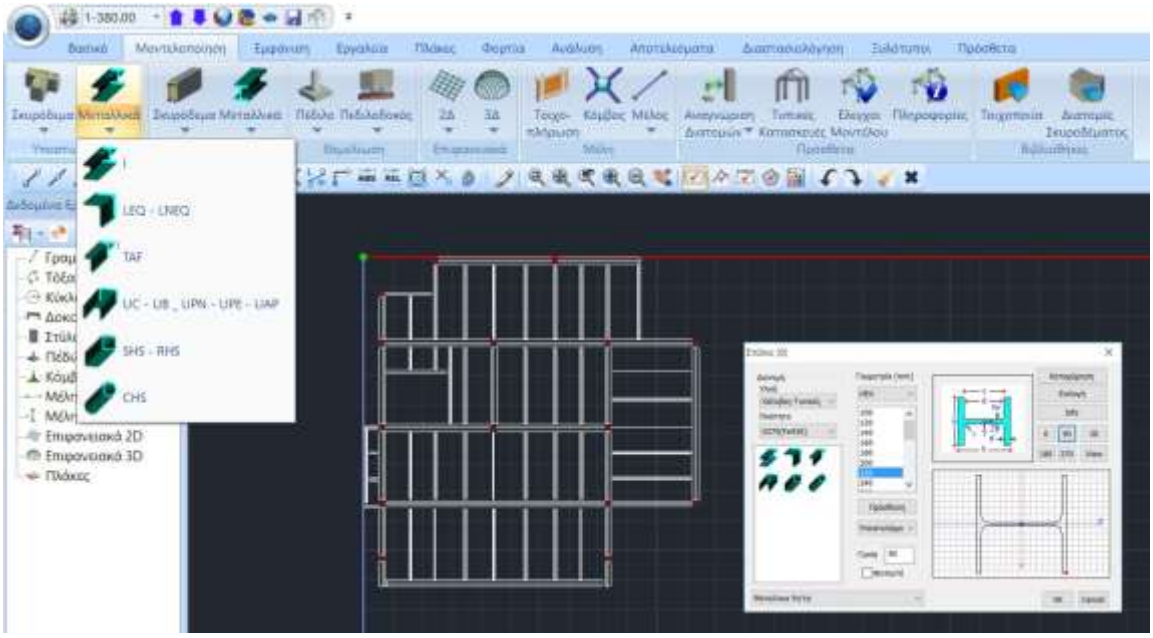




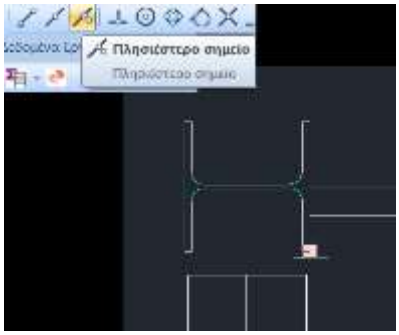


The drawing is displayed on the desktop allowing for pulls on the points.
Through the "Modeling" field you start the process of inserting the metal sections.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



Select the Metal columns and set the cross-section HEA220 with angle 90 quality S275. Then with the help of the pulls insert all the cross-sections of the columns of level 1.

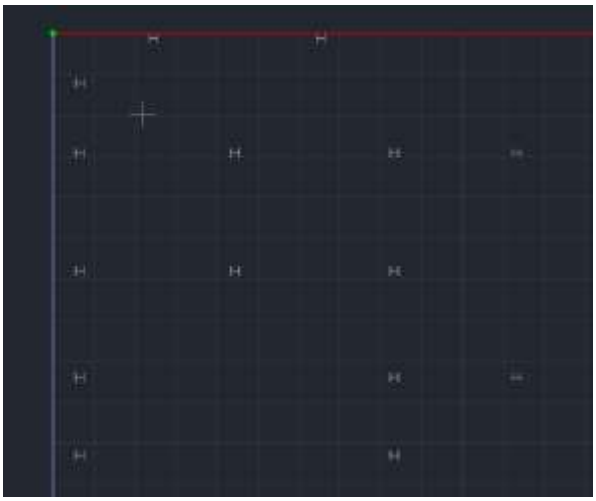


Using the commands 

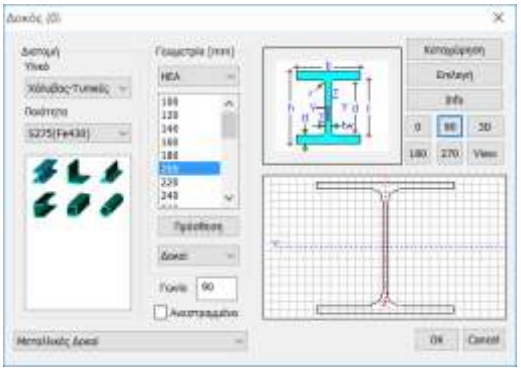
, copy to foundation level 0 the cross-sections of the columns.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

STATION 1

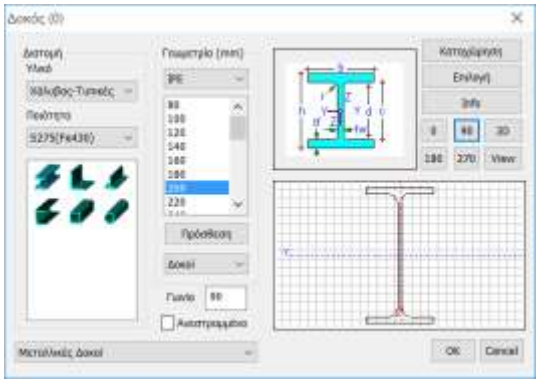


Return to level 1 and the "Modeling" field to enter the beams.



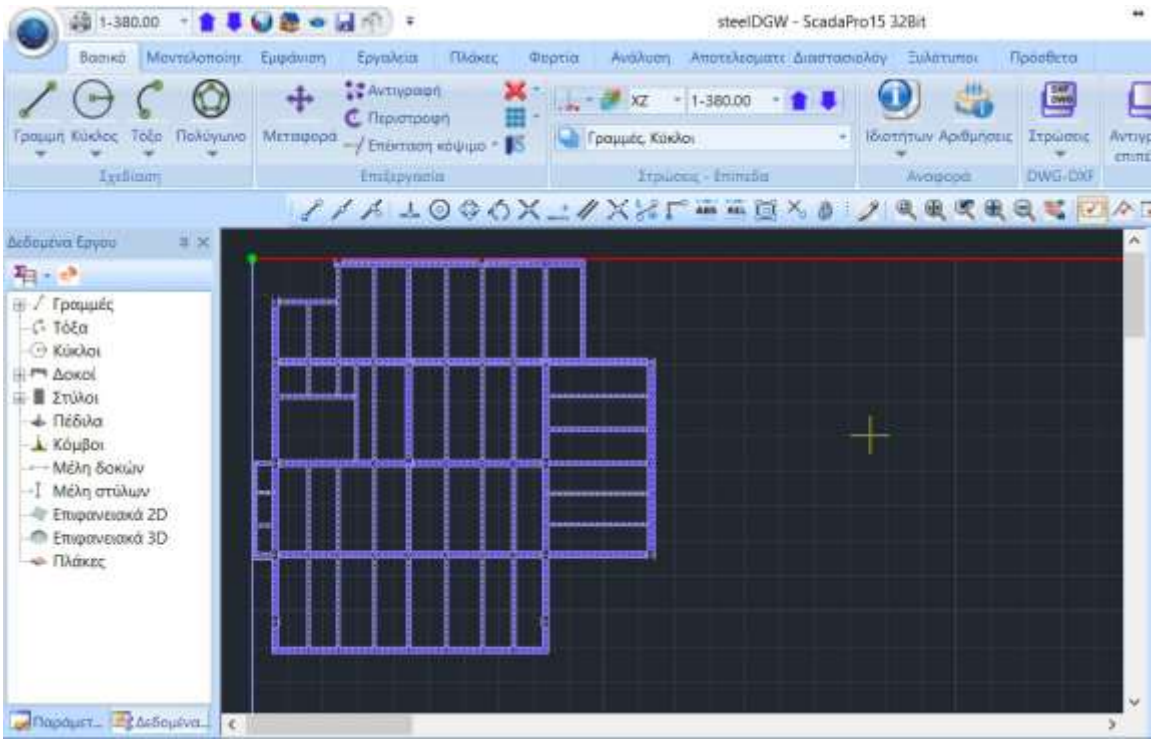
First set the HEA200s at an angle of 90 and with the help of the pulls insert them into the respective positions.

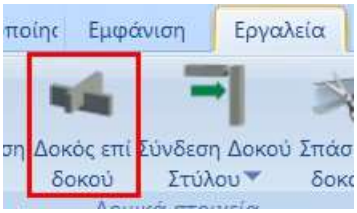
⚠ **NOTE:**
"**AutoTrim**" allows you to insert the beams without interrupting them where they an intermediate post.



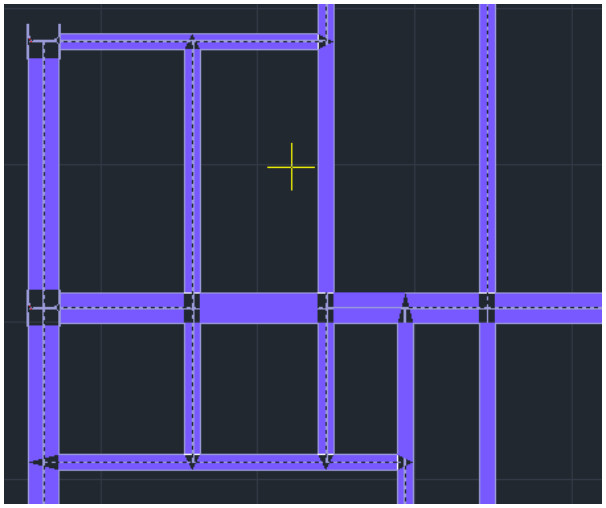
Then, set the IPE200s at an angle of 90 and with the help of the pulls, insert them in their respective positions.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



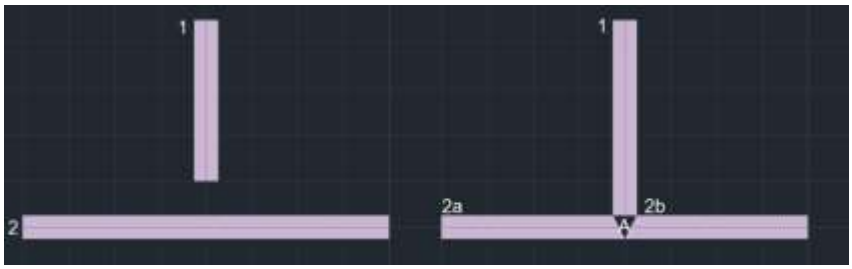


After the import of all the physical cross-sections of the plan view is completed, through the "Tools" group and the "**Beam on Beam**" command, you create the intermediate supports, where the intermediate nodes will be created with the subsequent creation of the mathematical model.

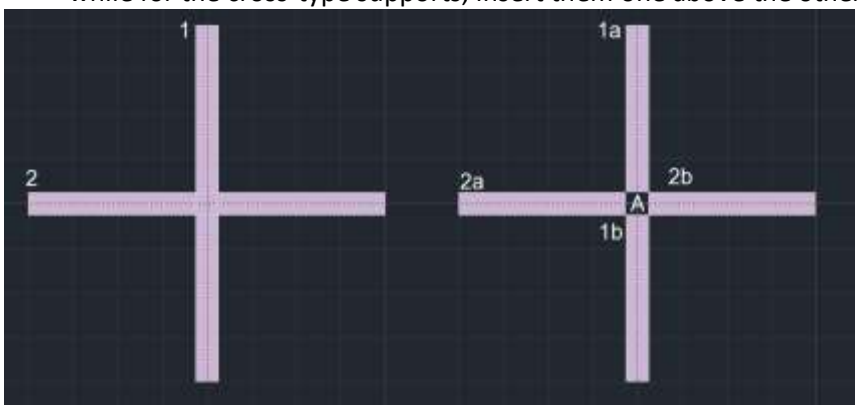


⚠ **NOTE:**
For T-type supports, make sure that the two beams do not intersect during insertion,

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



while for the cross-type supports, insert them one above the other.



2.6 Copying layers and configuring floor plans

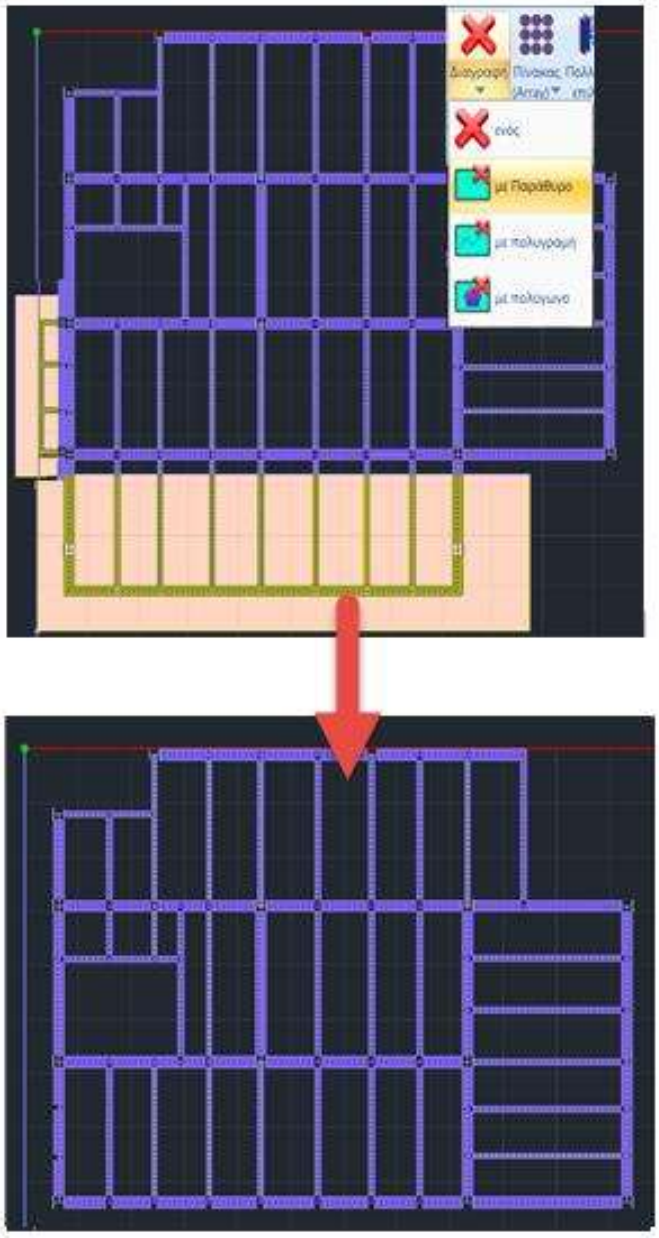
'Having completed the physical model of level 1, using the commands 

you copy the cross-sections to level 2.

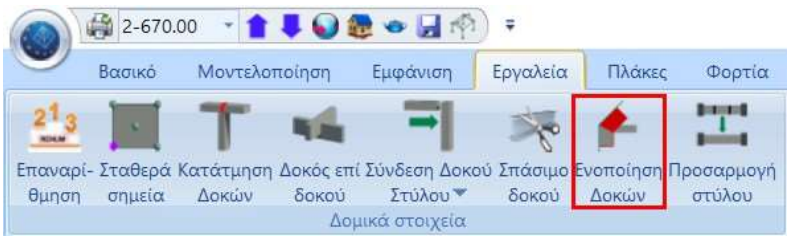
Then you shape the floor plan according to the study, making the necessary deletions:

STATION 2

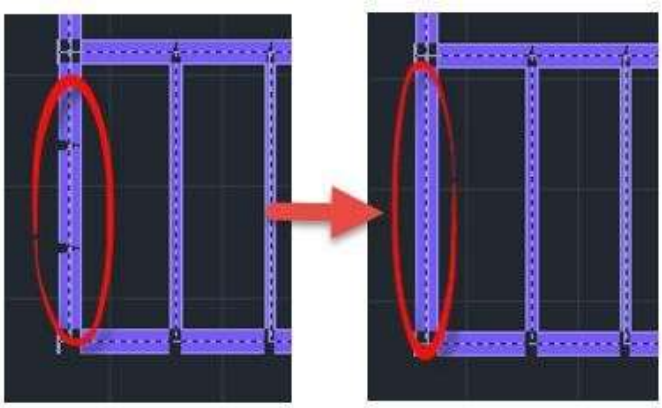
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



From this modification, the need arises to unify the parts of the beam where the indirect supports are no longer present.
Select the command "Consolidate Beams" and then one by one the beam sections to join the beam:

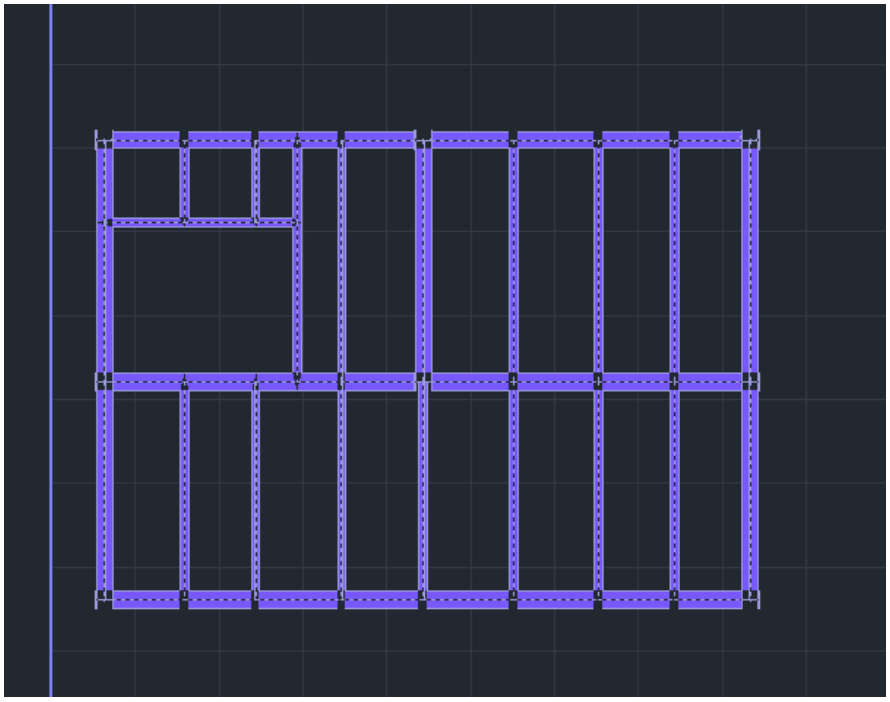


EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



In the same way you configure level 3. Copying from level 2, deleting the extra sections and consolidating the broken beams until the following physical model is formed:

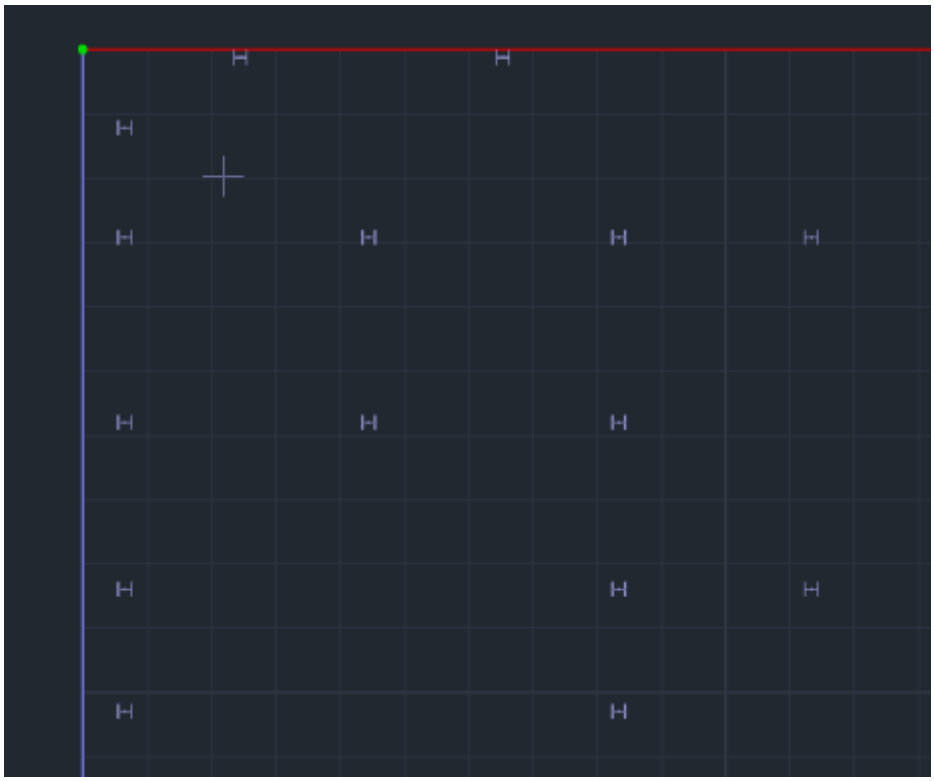
STATION 3



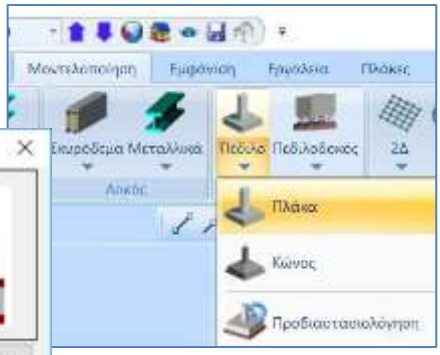
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

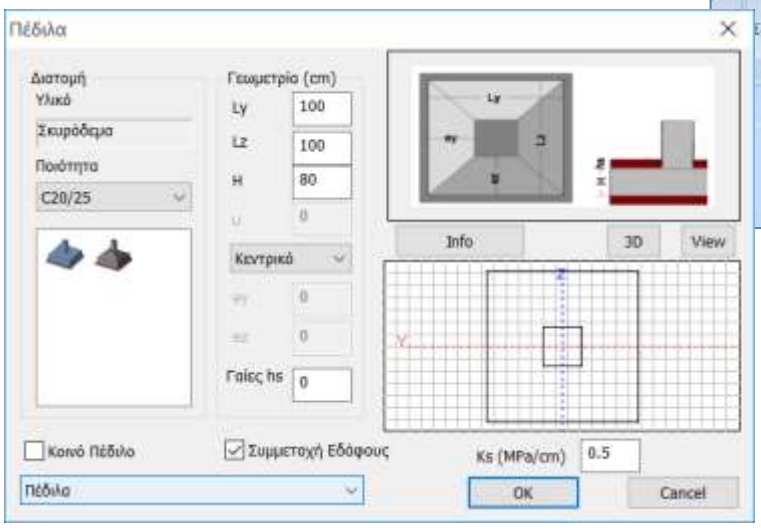
Foundation level configuration

To configure the foundation level, go to Level 0:



Select the command "Pediment-Plate" and set the characteristics of the pediments.

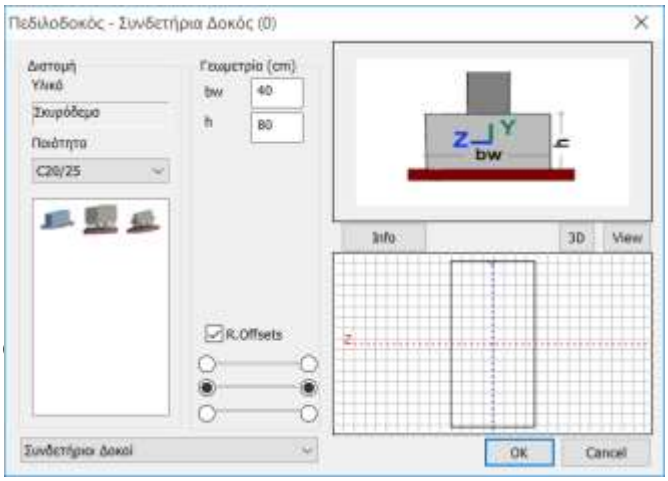




Then, insert the central skirts into the bases of the posts with the help of the pulls.

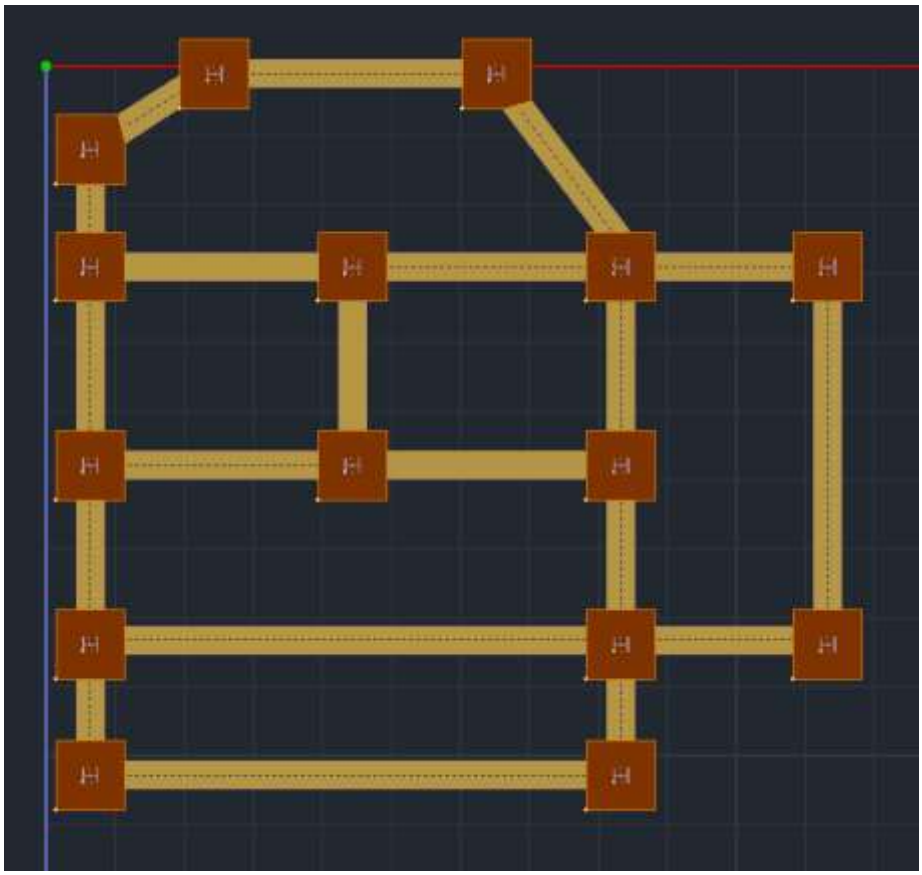
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

Finally, select



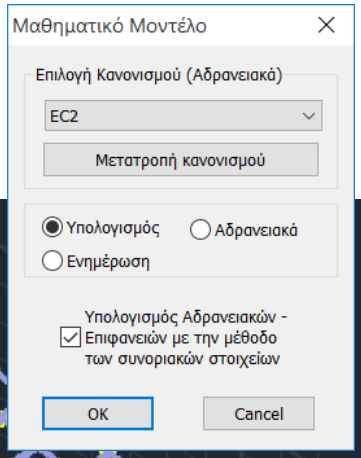
and after setting the attributes, enter th
in both addresses:

STATION 0



2.8 Creating a mathematical model

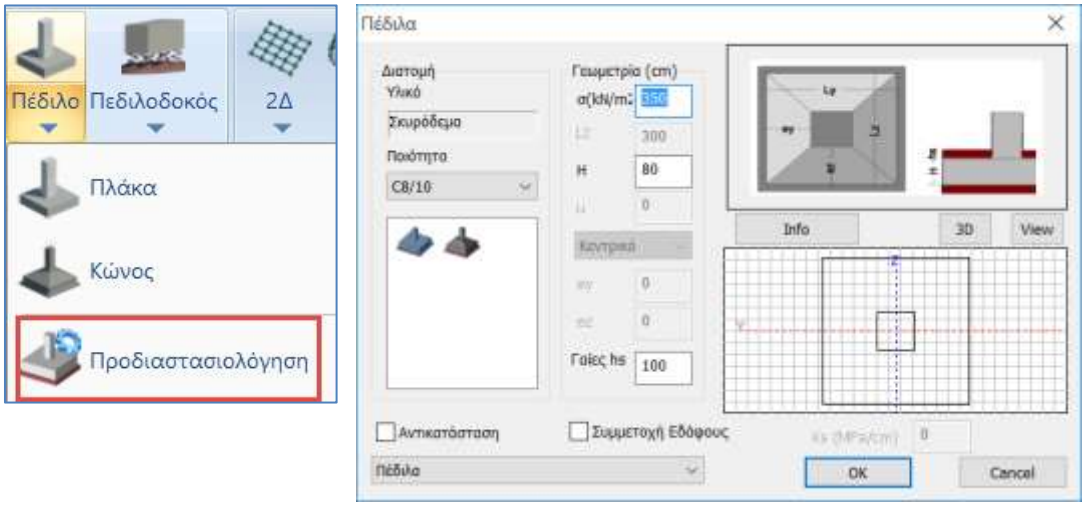
After creation of the Physical Model of the whole vector is completed, the creation of its Mathematical Model follows, which is automatically calculated through the corresponding command:



EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

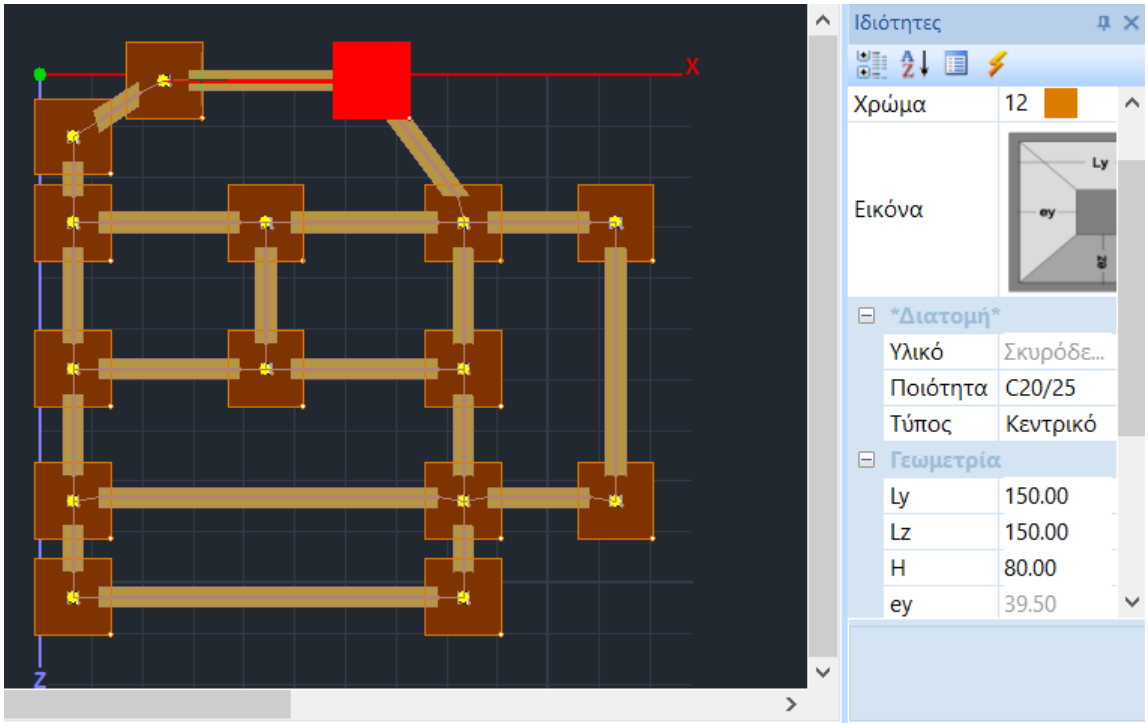
NOTE:

A new feature offered by SCADA Pro, after the creation of the Mathematical Model, is the "Prescaling" of the fields:



which, according to the soil tension σ(KN/m2), the height of the footings H and the overlying soil hs, pre-standardizes the existing footings, possibly modifying their dimensions:

as in this example where from 100x100cm they were modified to 150x150cm:



3. IMPORTATION OF GOODS

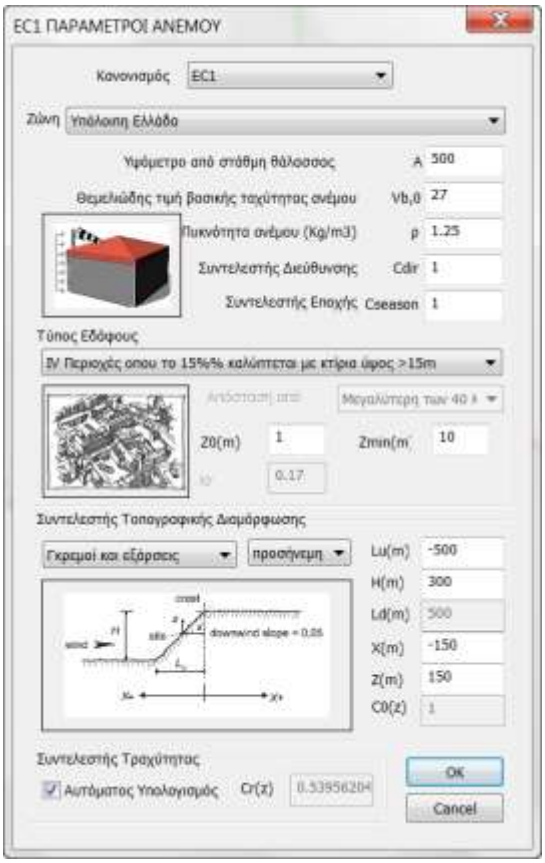
3.1 How to enter wind and snow loads in the automatic way based on Eurocode 1

In steel structures the influence of wind and snow loads is particularly important and must be taken into account.

Through the "Loads" module and the "Wind - Snow Loads" command group, the appropriate tools for calculating and distributing the loads on the individual walls and roofs of the structure can be found.
The process you follow starts with setting the wind and snow parameters according to the location of the structure.

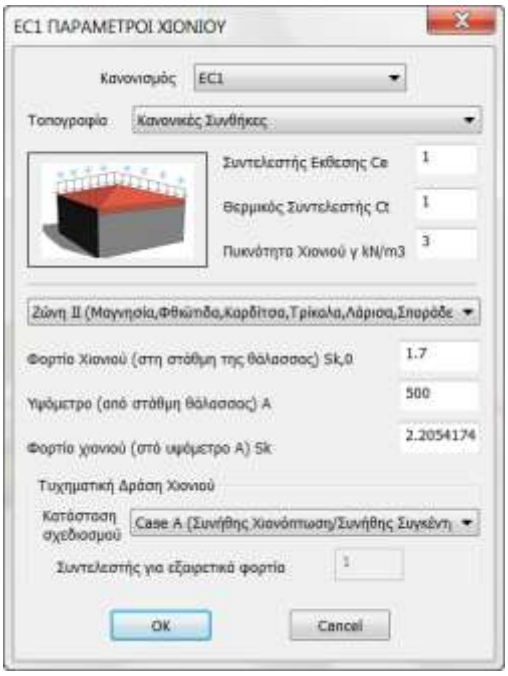
3.1.1 Parameters

In the wind parameters window:



You select EC1 and indicate the Zone, Land Type, Topographic Configuration, and the necessary wind values.
The Speed Factor is calculated automatically if Auto Calculation is selected or set by the user otherwise.

Similarly, in the snow parameters window:



Specify the Topography that determines the values of the coefficients Ce and Ct, the Zone and select for Greece Planning Condition A.

3.1.2 Wall Treatment

Then, via "Edit> "**Walls**", we define the walls per direction for the calculation of the Equivalent Wall.

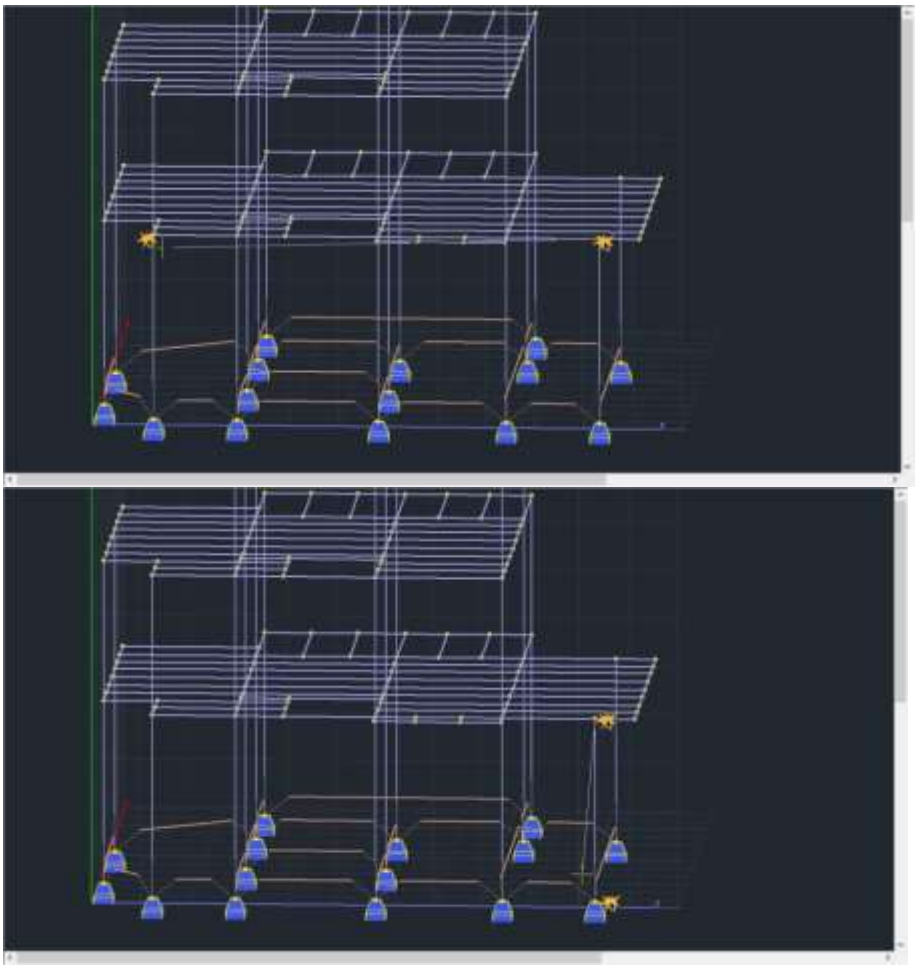
We start, for example, from the wall on the left, perpendicular to wind direction 0.

give the program the length (b) and height (h) for each wall (Left, Front, Right, Back), simply by clicking on Pick and selecting each time with the mouse the 2 ends of the wall in the corresponding direction, viewing the vector in 3D.

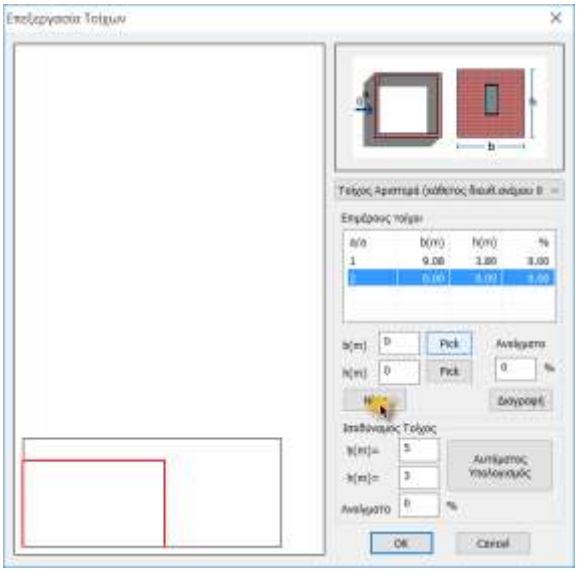
Our goal is to define all the wall sections that are perpendicular to the wind direction 0, in a graphical way, using Pick and showing the wall points to determine the length (b) and height (h) of each section, per level.

- For fixing the first part of the first floor:
  press the Pick next to b(m) and point with the mouse to the 2 ends of its length, then press Pick next to h(m) and point with the mouse to the 2 ends of its height.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

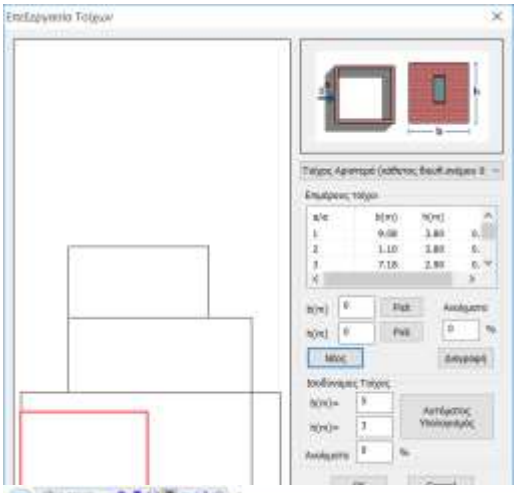


- For fixing <u>the second part of the first floor:</u> Press "New" and repeat the previous procedure.

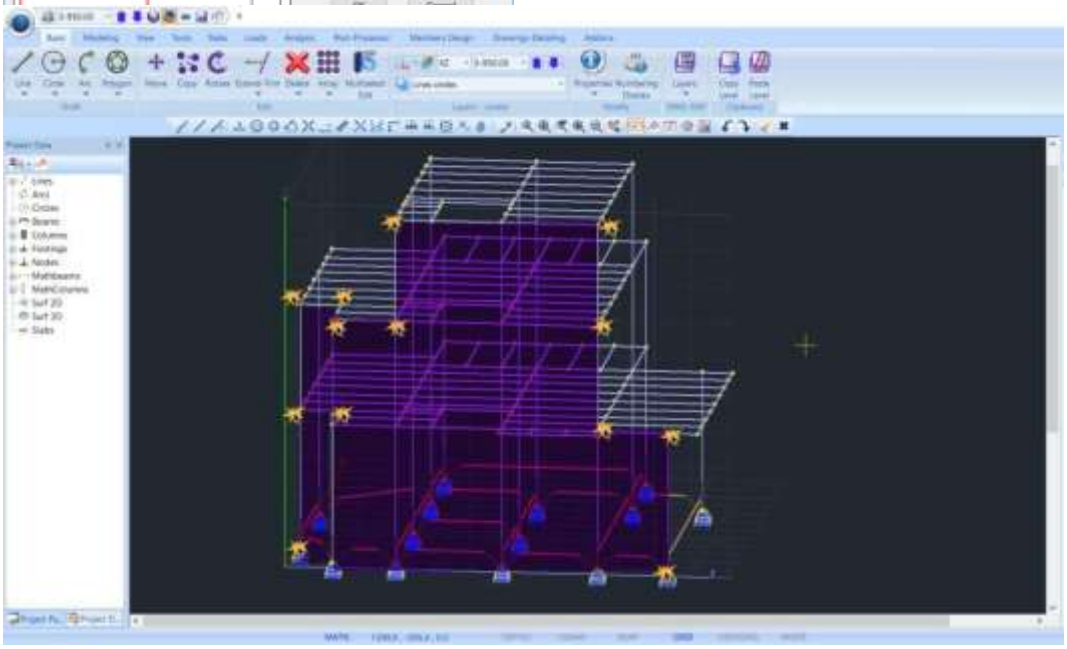


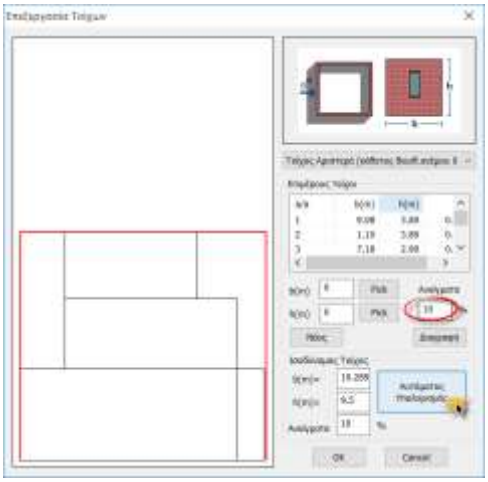
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

- For fixing the wall sections on the 2th and 3th floor:



Press "New" and repeat the previous procedure until you have defined the entire view.



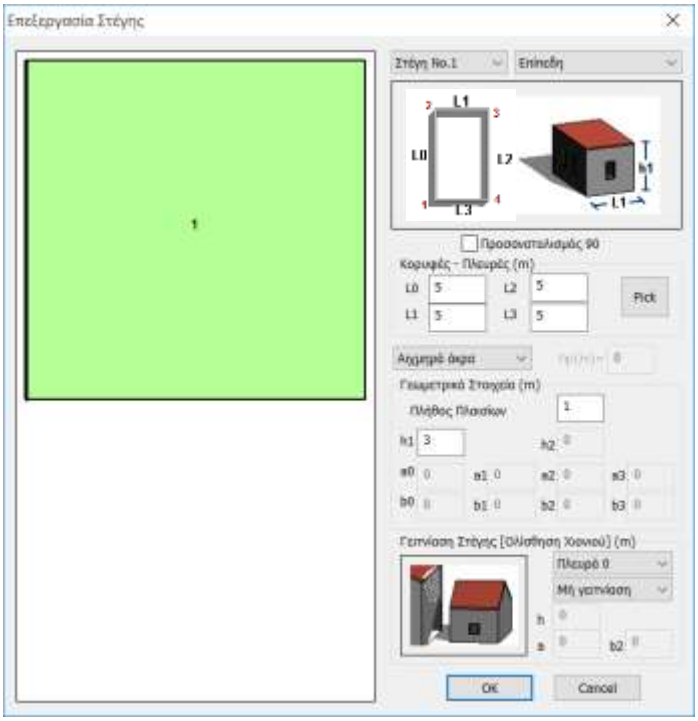


Then, set the percentage of openings and press the Αυτόματος Υπολογισμός button so that the program calculates the dimensions of the equivalent wall.

Repeat for all four directions of the walls.

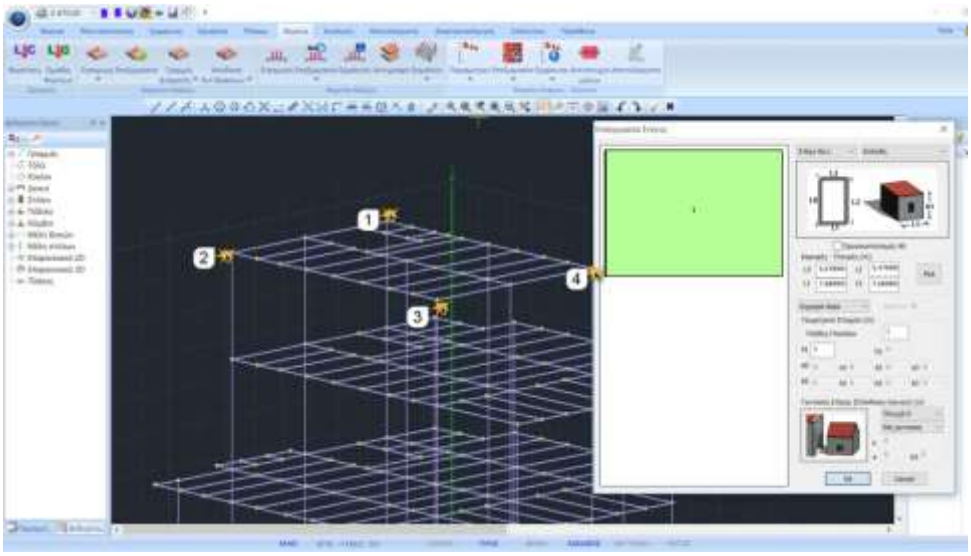
3.1.3 Roof treatment

Similarly, from "Edit> "**Roofs**",

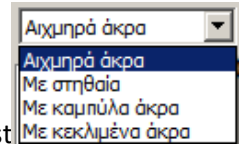


define the type of roof, its orientation and the dimensions Lo,L1,L2,L3, by pressing Pick and selecting each time with the mouse the 4 edges of the roof.

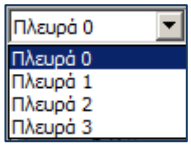
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

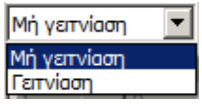


NOTES:

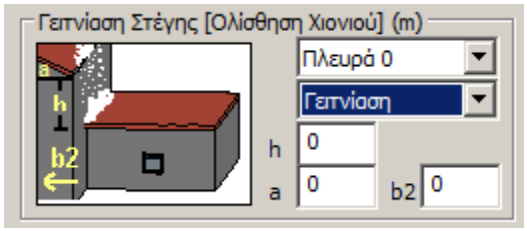
- If your roof has an obstacle (snow accumulation point) select the type of obstacle from the corresponding list 

and enter its height in m.

- If a structure is adjacent to another taller one, in the "Roof adjacency" field select the adjacent side 

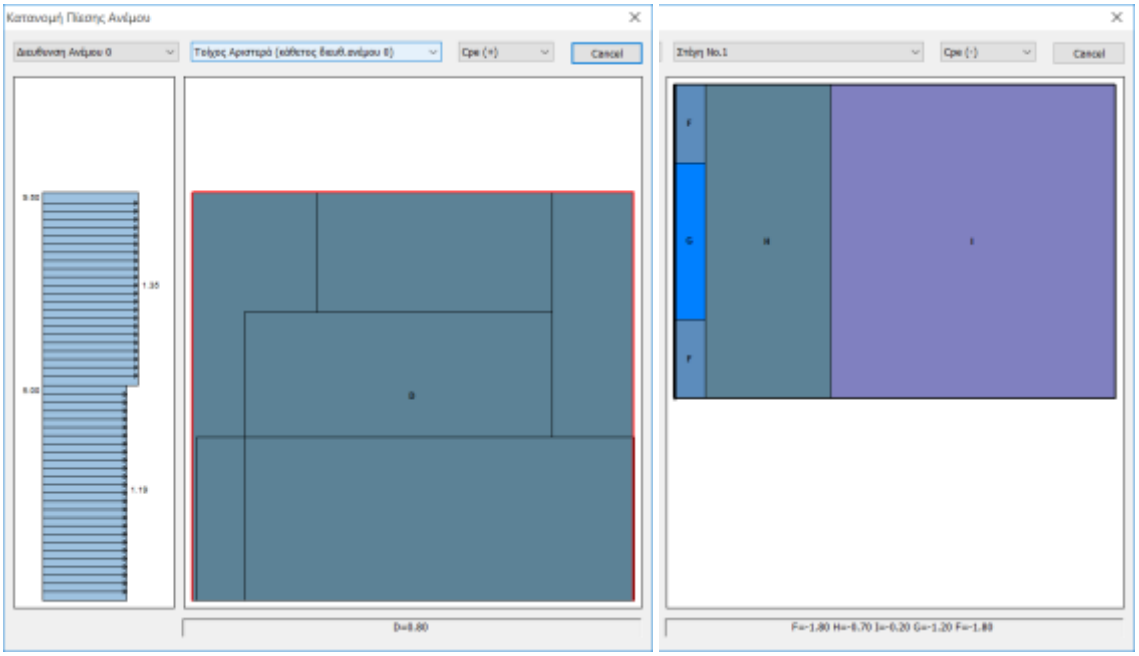
and from the list 

the "Earthing". The "Roof Coordination" field is modified accordingly enter the necessary geometric characteristics. "OK" to save the parameters.



3.1.4 Wind Show

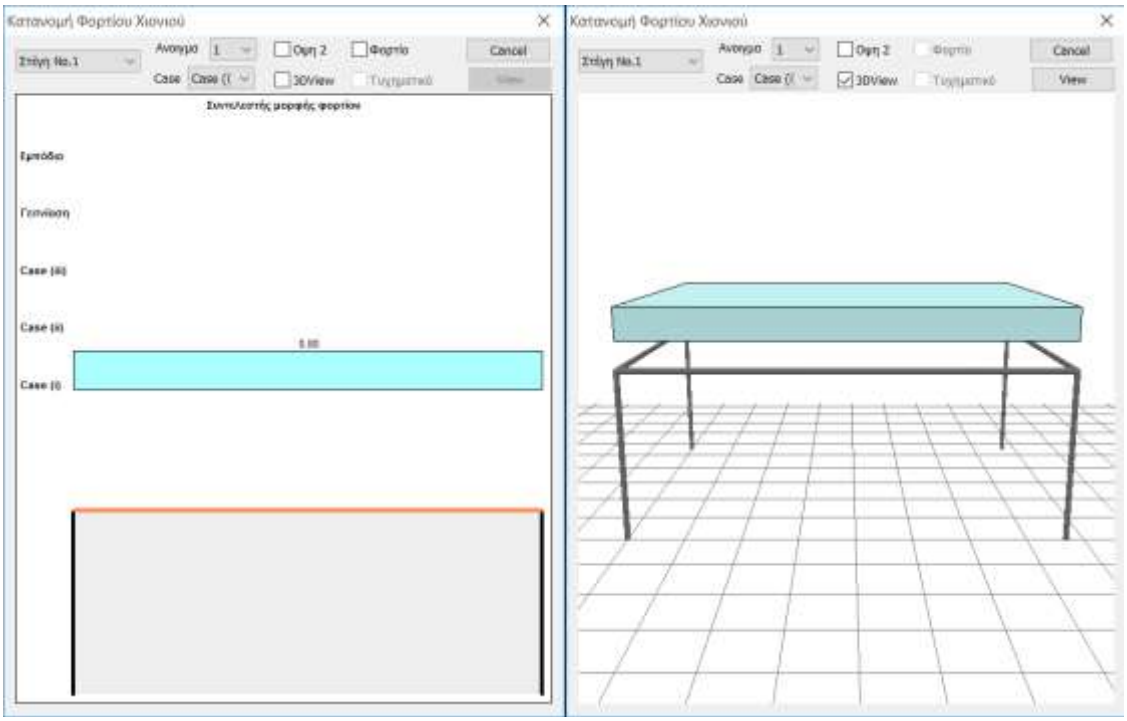
With the command "**Show**" > "**Wind**", you can see for each wind direction the distribution of pressure per height with the coefficients Cpe+ , Cpe- , Cpi , for each wall and for the roof.



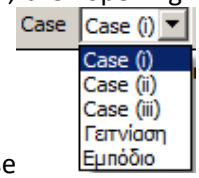
3.1.5 Snow Show

Similarly, from the next option you can see the snow load distributions on your roof during EC1.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



In the dialog box select from the lists on the top left the number of the "roof", the "opening"

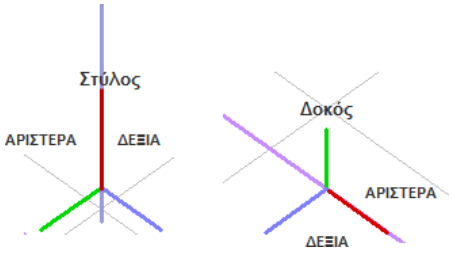


meaning the number of the frame, in case you have more than one, and the "Case for the distribution of the snow load.

Also check the checkbox next to "Load" to see the load values and next to "3DView" to get the snow distribution in the following illustration.

3.1.6 Matching members

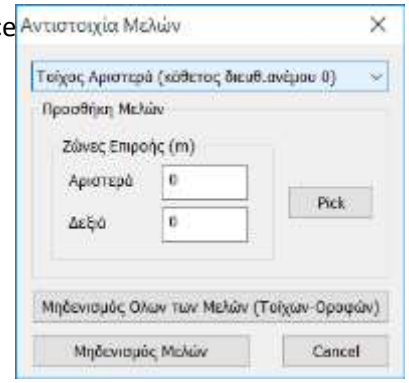
The next step is to define the zones. This is to determine the distribution of pressure (wind and snow) on the structural elements of each facade and roof.
Select for the wall and roof members the influence zones, i.e. the width of the pressure surface, Left and Right of the member and make Pick on the members of the member you want to assign.



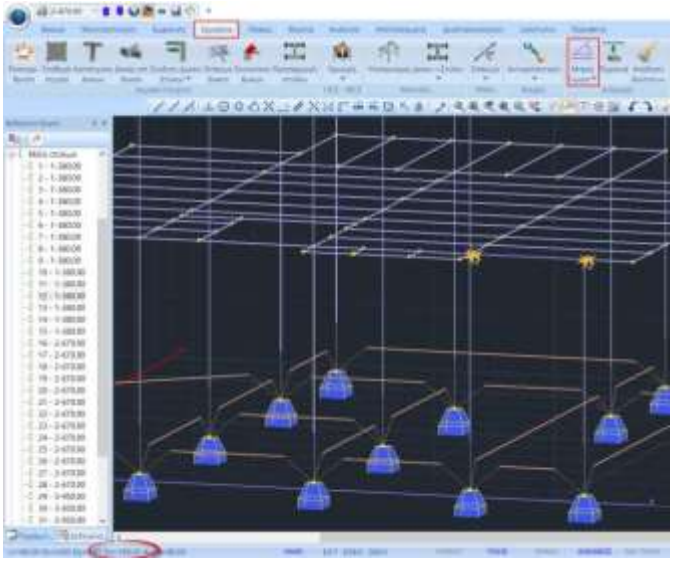
⚠ The Left and Right widths are referenced to the local x-axis (the longitudinal) of the member. We are assumed to be at the member's start node (i) and looking at the end node (j).

In detail:

For example, if you want to set the zone of influence of a Wall Left pole:

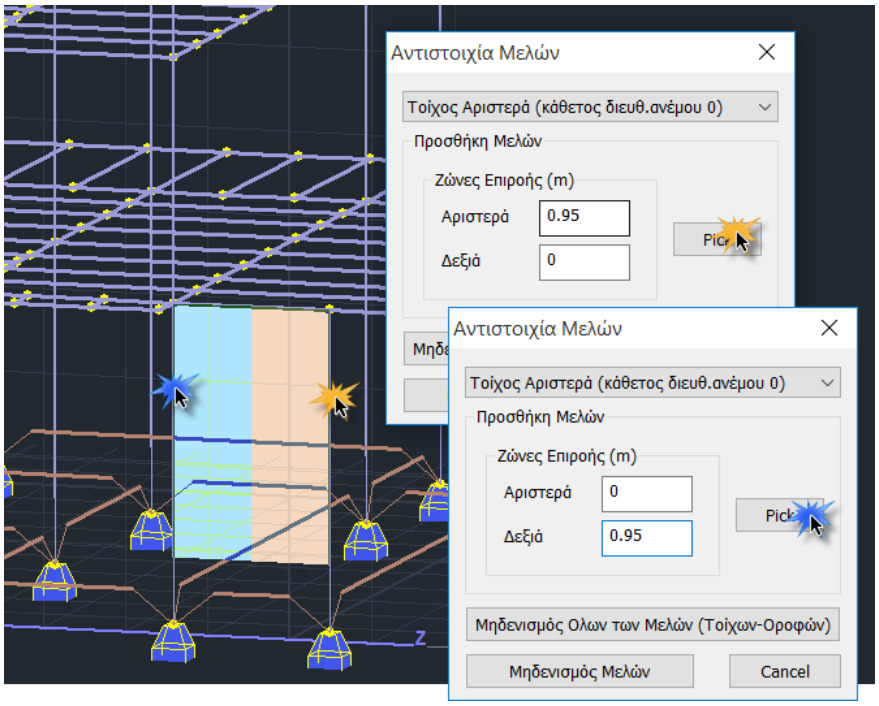


First you need to know the width of the zone that the two successive poles will share. With the help of the Tools and the "Length-Angle" command, measure the distance between the two poles (dZ=190cm):



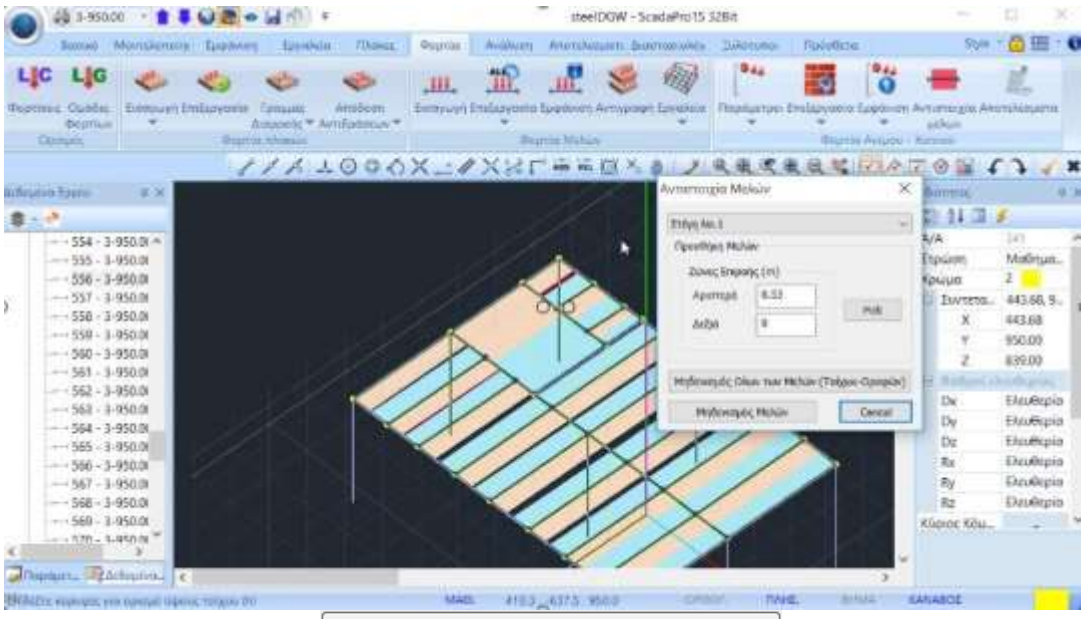
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

Each pole will receive half of the pressure in the zone with a width equal to the distance between them, i.e. 190/2 cm = 0.95 m on the left for the right pole and on the right for the left.



⚠ **NOTE**: In each Pick you can select more members and assign the same influence zone width.

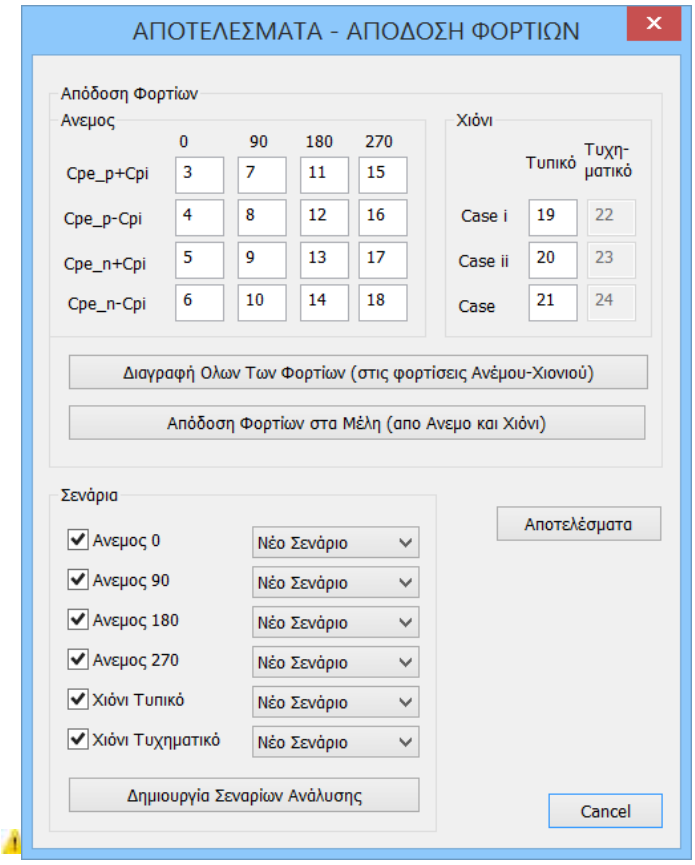
Similarly you define the zones of influence on all members of all walls and roofs.



⚠ **NOTE**: Pressing Μηδενισμός Ολων των Μελών (Τοίχων-Οροφών), reset the zones for all the members of all facades of the building, while with the Μηδενισμός Μελών, only the zones of the members of the selected wall or roof are reset.

3.1.7 Results

Last command, the "Results" command.



In the dialog box, in the "Load Performance" field there are two sections;

- the wind loads, 4 loads for each of the 4 directions, for a total of 16 loads, and
- snow loads, 3 loadings for typical snowfall (random is not applicable to the Greece).

The numbers shown in the fields are the numbers of the loadings.

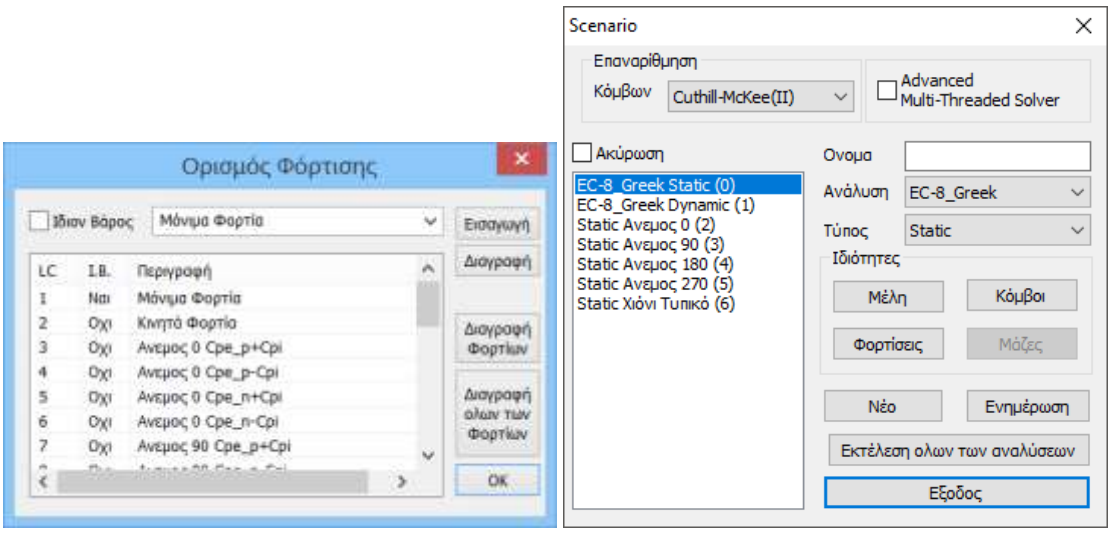
Permanently
Charge 2: Mobiles
and now another 16 loadings are added for wind (from 3 to 18) and 3 for snow (19, 20 and 21)

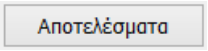
Select to command Απόδοση Φορτίων στα Μέλη (απο Ανεμο και Χιόνι) to to attribute loads of wind and of snow to members the construction , or members the construction, or Διαγραφή Ολων Των Φορτίων (στις φορτίσεις Ανέμου-Χιονιού) delete them all.

The "Scenarios" field contains a list of all possible analysis scenarios, which are automatically generated via the command ! Δημιουργία Σεναρίων Ανάλυσης

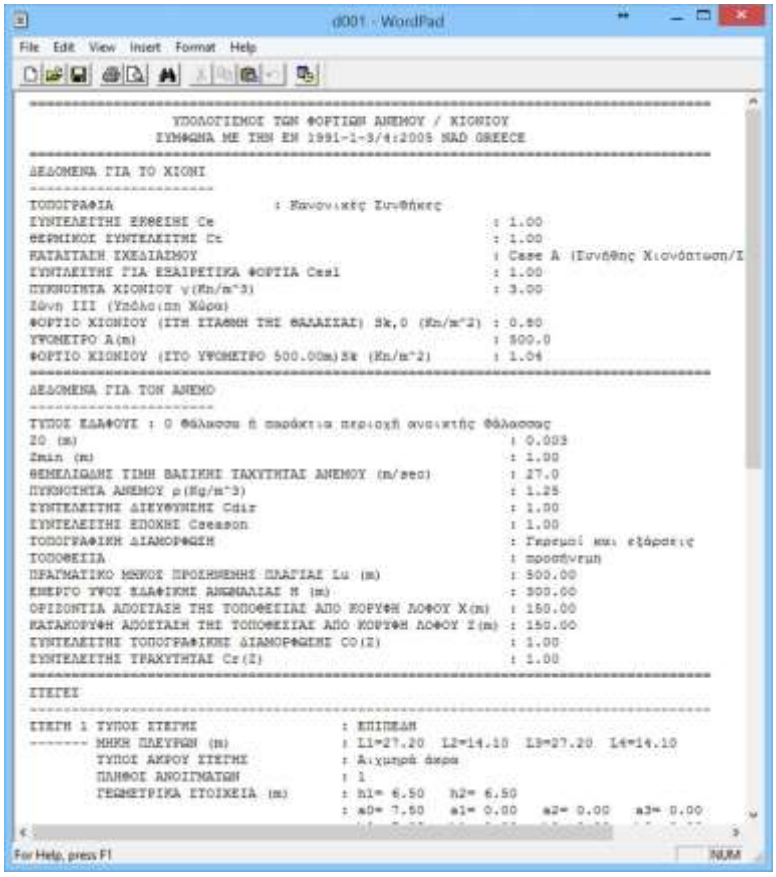
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

⚠ So SCADA Pro, in addition to automatically calculating distribution of wind and snow loads, automatically creates all the analysis scenarios.



The command 

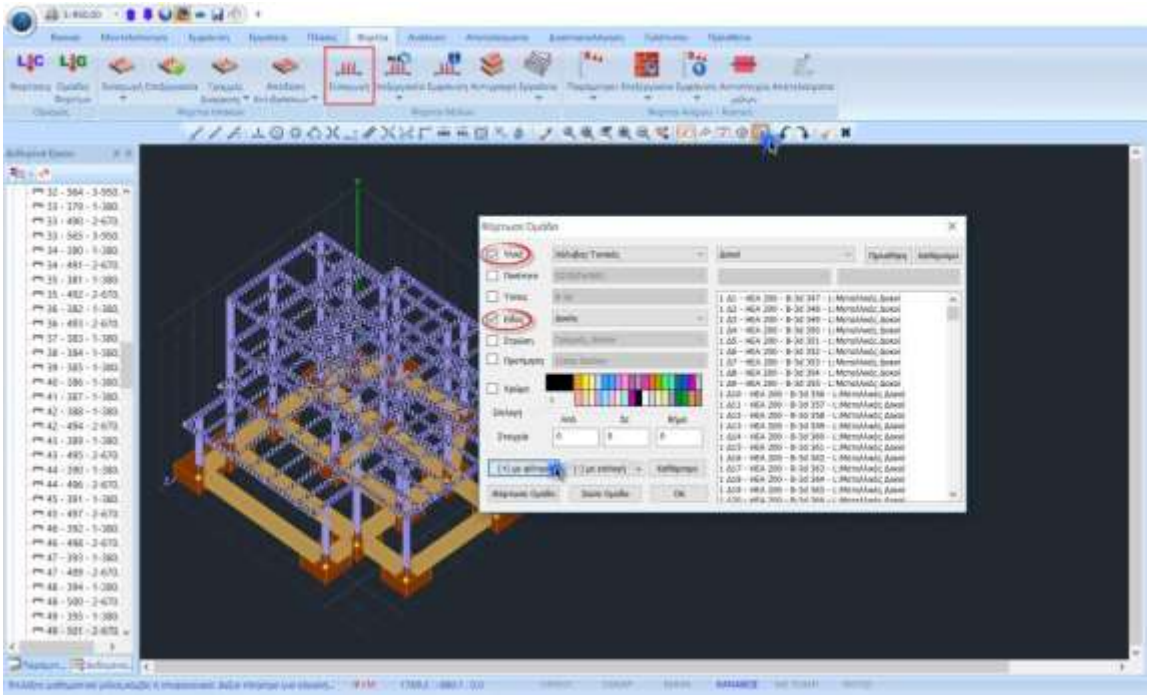
opens a txt file of the results, detailing all the data and calculations from each command in the "Wind - Snow Loads" group.



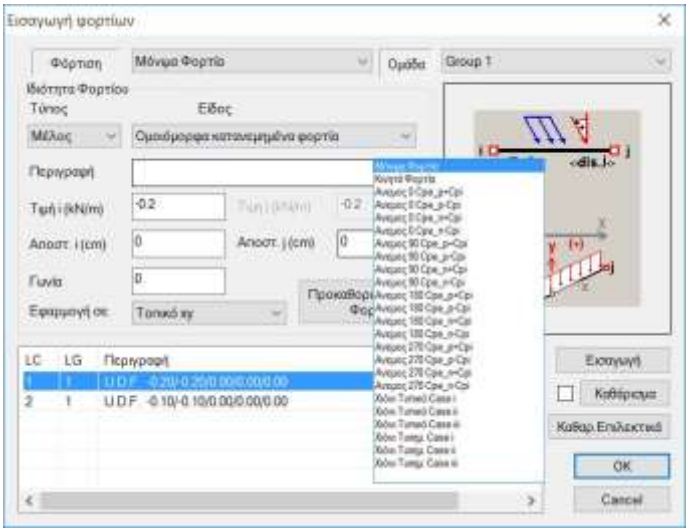
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

For the needs of this study you will provide additional:
- Permanent loads :0,20 KN/m in all beams
- Mobile loads :0,10 KN/m in all beams

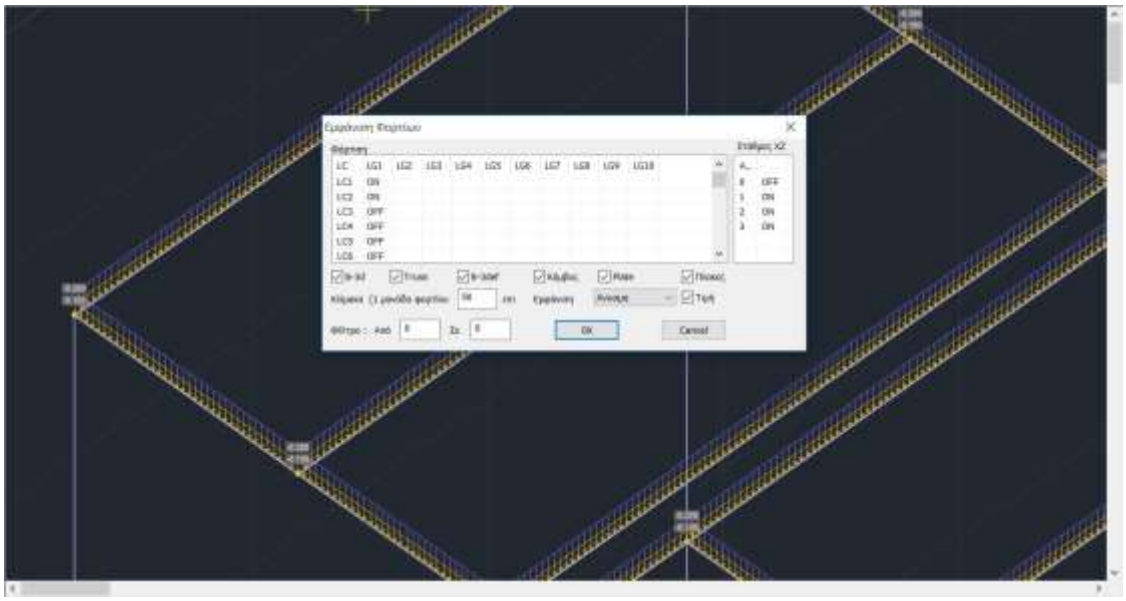
Select Import and the option with Group. Check Material: "Steel" and Type: "Beam" and press "(+) with filter" to select all metal beams.



OK and Right click to open the Load Import window to define the Permanent and Mobile loads on the beams:



EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

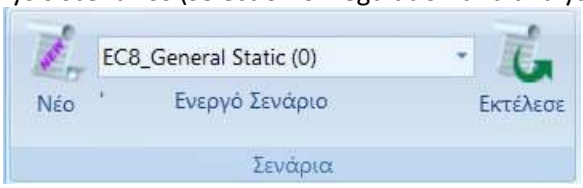


4. ANALYSIS

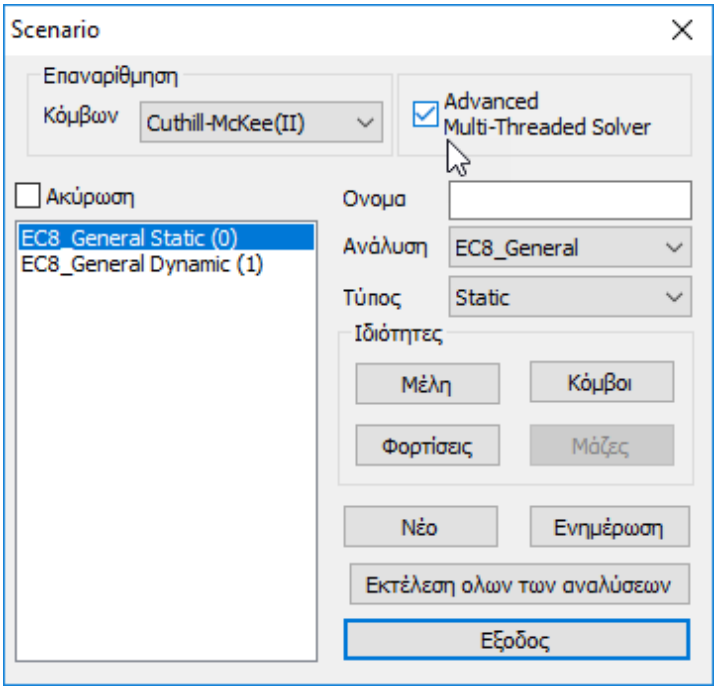
After the completion of the modeling of the structure and the input of the loads in the members, the analysis of the design based on the regulation you will define, the automatic creation of the load combinations and the results of the checks that will be obtained.

How to create an analysis script:

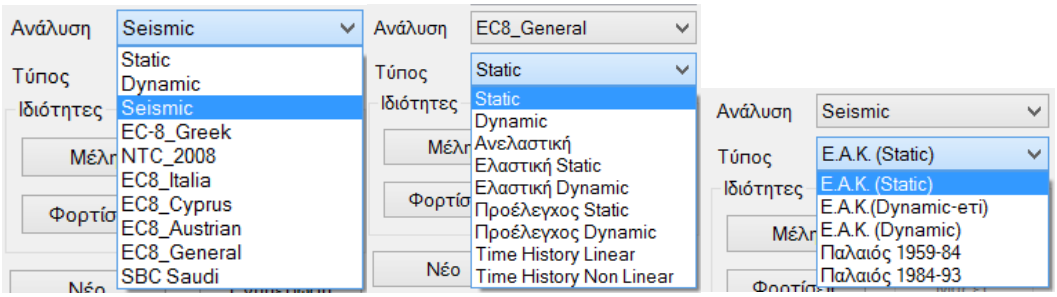
Within the "Analysis" Module, the commands of the "Scenarios" group allow the creation of the analysis scenarios (selection of regulation and analysis type) and their execution.



In the dialog box that accompanies the selection of the New command, you can create several analysis scenarios, in addition to the 2 predefined ones*



EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

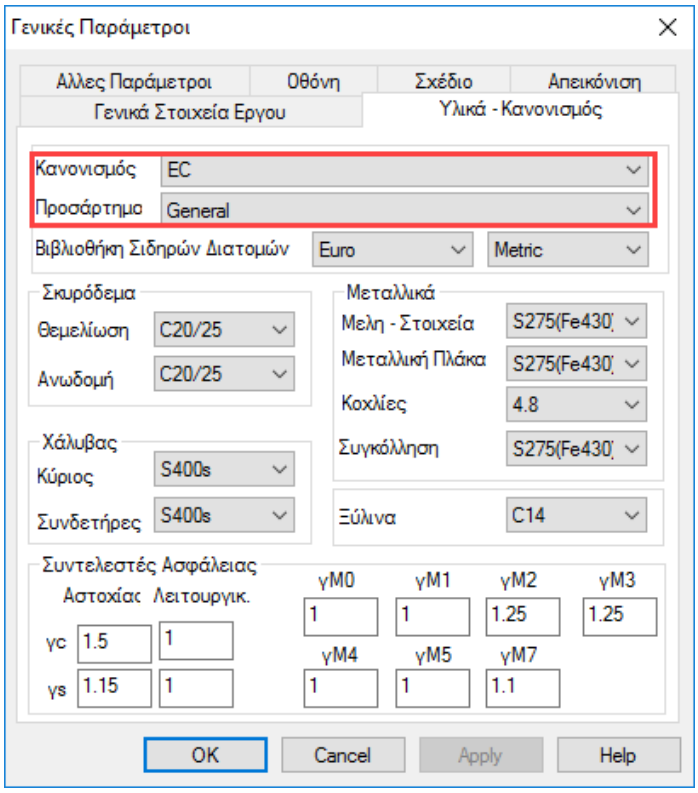


Select from the "Analysis" list and the corresponding "Type" list and click on 

to create a new script.

ATTENTION: The materials be in accordance 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.



NOTE: Materials are automatically adjusted according to the selected regulation, so that during data entry, all sections are given the correct grades and reinforced with the corresponding steel.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

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 homologues torsional pairs
- EAK Dynamic	Dynamic spectral analysis with shifting of the masses
- Old 1959-84	Seismic analysis based on the regulation of 1959
- Old 1984-93	Seismic analysis based on the regulation of 1984
- static	Analysis without the participation of seismic actions
- EC 8 Greek static	Structural analysis based on Eurocode 8 and the Greek Appendix
- EC8 Greek dynamic	Dynamic analysis based on 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 based on Erocode 8
- EC 8 Greek Time History Non Linear	Dynamic analysis based on the Eurocode 8
- EC 8 English Elasticity	Anelastic seismic analysis based on the 8 or the EDPC.

For overseas:

ELASTIC - UNELASTIC

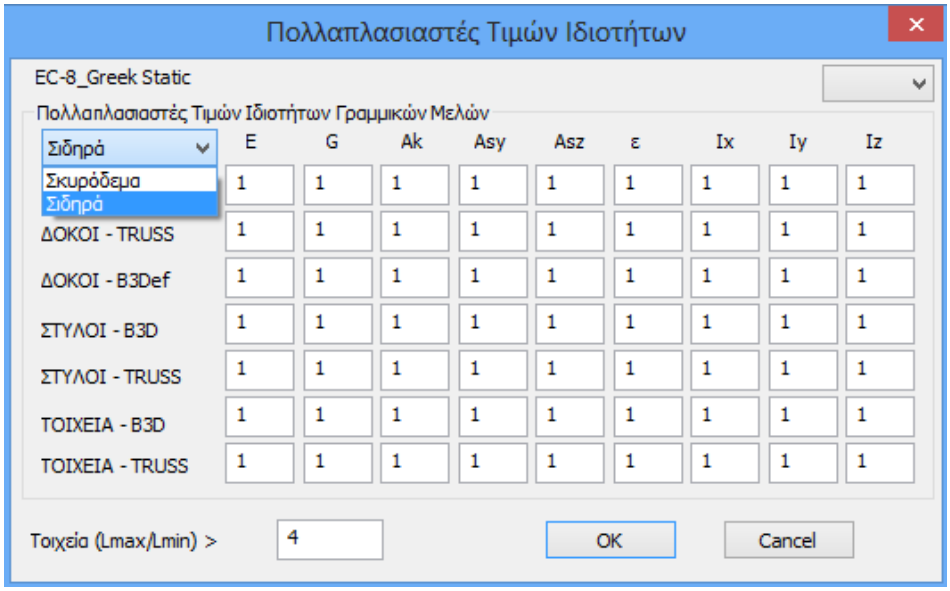
- NTC 2008	Seismic analysis based on the Italian regulation of 2008
- EC8 Italy	Seismic analysis based on Erocode 8 and the Italian Appendix
- EC8 Cyprus	Seismic analysis based on Erocode 8 and the Cyprus Appendix
- EC8 Austrian	Seismic analysis based on Eurocode 8 and the Austrian Appendix
- EC8 General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

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

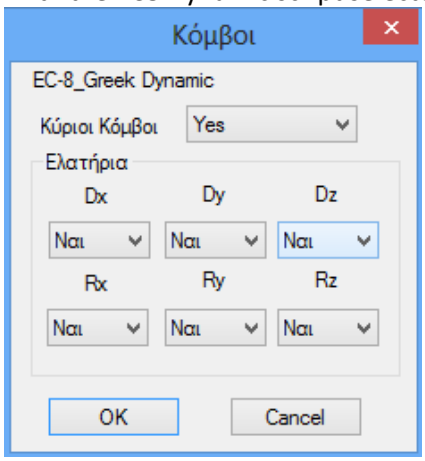
For this example you will select only the EC8 dynamic earthquake scenarios, as well as the Snow Typical, Wind 0 and Wind 90 scenarios, which were automatically created by the program in the previous step.

With the EC8 Dynamic script selected, the **Properties - Members** command includes the property value multipliers of the linear members.

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



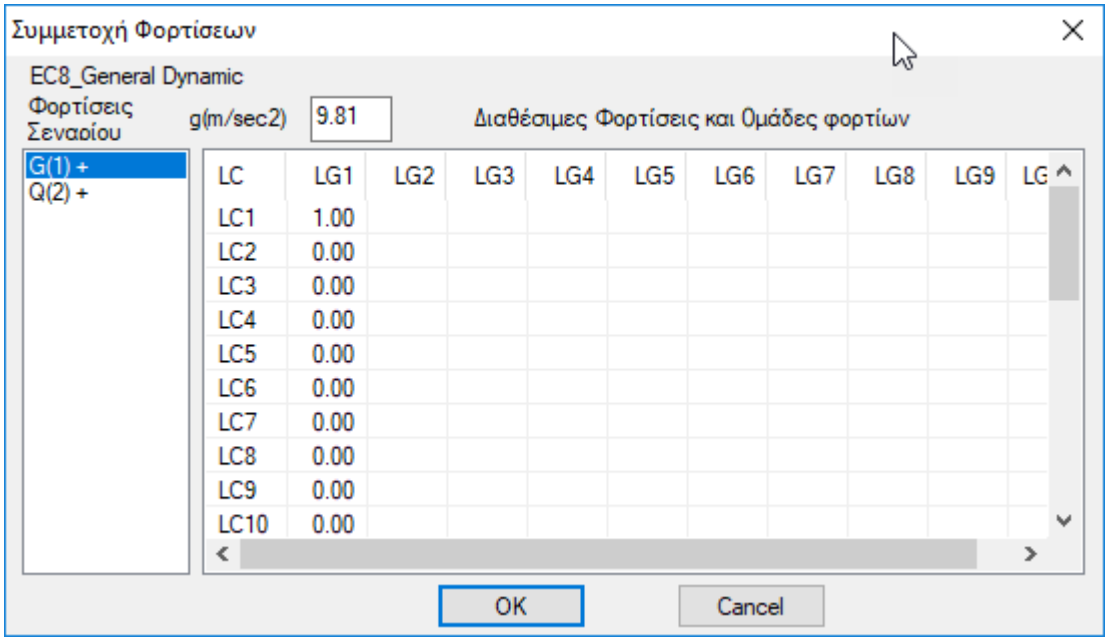
With the EC8 Dynamic script selected the "**Nodes**" command displays the following dialog box:



Here you can specify to resolve your girder without diaphragm mode altogether, even if there are diaphragm nodes, as well as to resolve it pressed (Spring No) even if elastic foundation is set.

In cases where Dynamic analysis is required, if you select "Nodes" for the corresponding dynamic scenario and "open" the springs ("Yes"), then you can use the dynamic combinations for the dimensioning of the foundation.

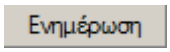
With the EC8 Dynamic script selected, the "**Load**" command opens the following dialog box:



where, for each charge of the scenario you created (left column), you assign one charge (LC) from the ones you created.

- select the value 1.00 for LC1 (after first selecting the category "Permanent Loads" - G(1), which is coloured blue) and 1.00 for LC2 (after first selecting the category "Mobile Loads" - Q(2), which is coloured blue).
- The "+" next to the charge category 

indicates that there is a charge participation for that particular charge.

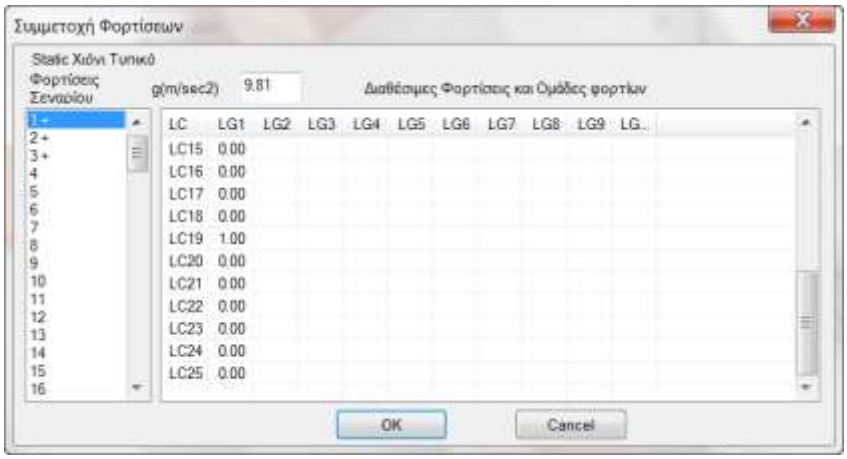


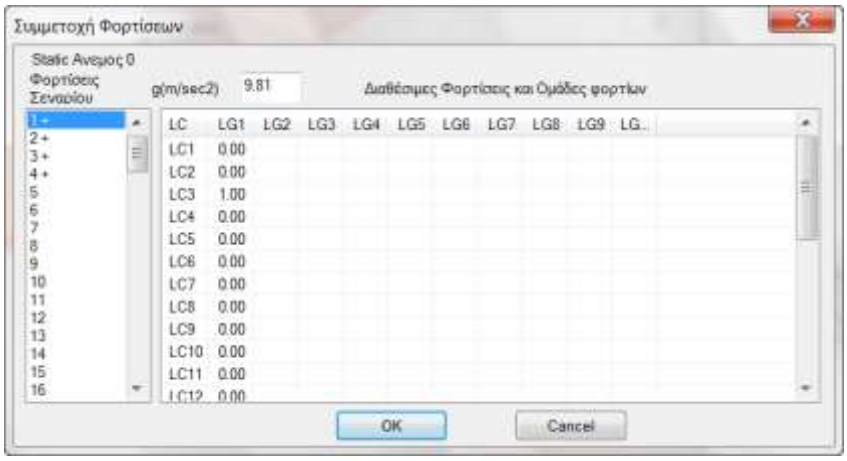
to update the script and register the changes.

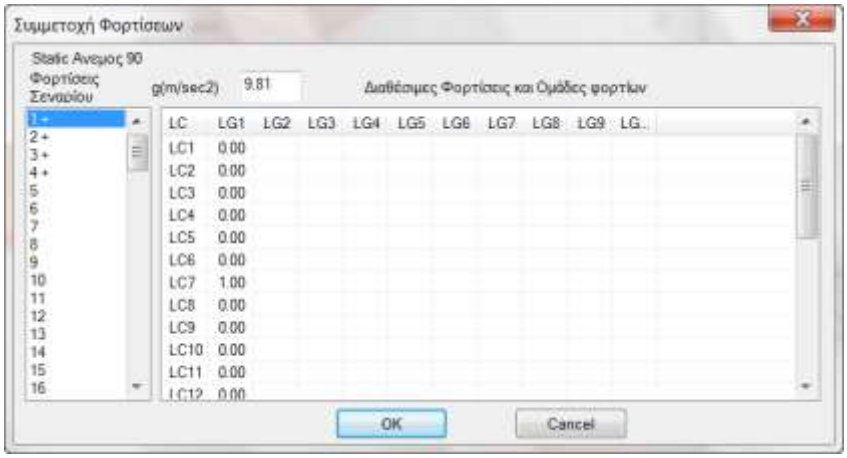
⚠ *The program automatically populates the unit in the corresponding charge, so any modification is, again, optional.*

In the Static snow and wind scenarios the respective loads are included in the static analysis without including the permanent LC1 and mobile LC2 , since they have been taken into account in the seismic analyses.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'





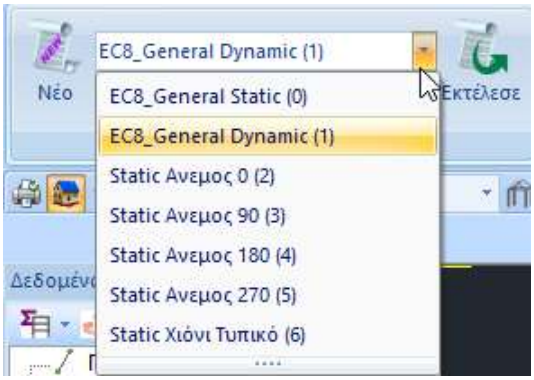


When each charge is activated, the+ symbol appears next to the charge number.

OBSERVATION

In each scenario you can set up to a maximum of 4 loads.

How to run an analysis script:



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

OBSERVATION:

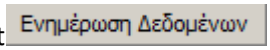
Alternatively, the new **Run all analyses** command 

allo

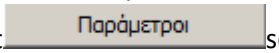
ws you to run all the scripts in the list with one click.

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

- √ the scenarios of the **Eurocodes** and
- √ the scenarios of **Statics**

First of all, you select 

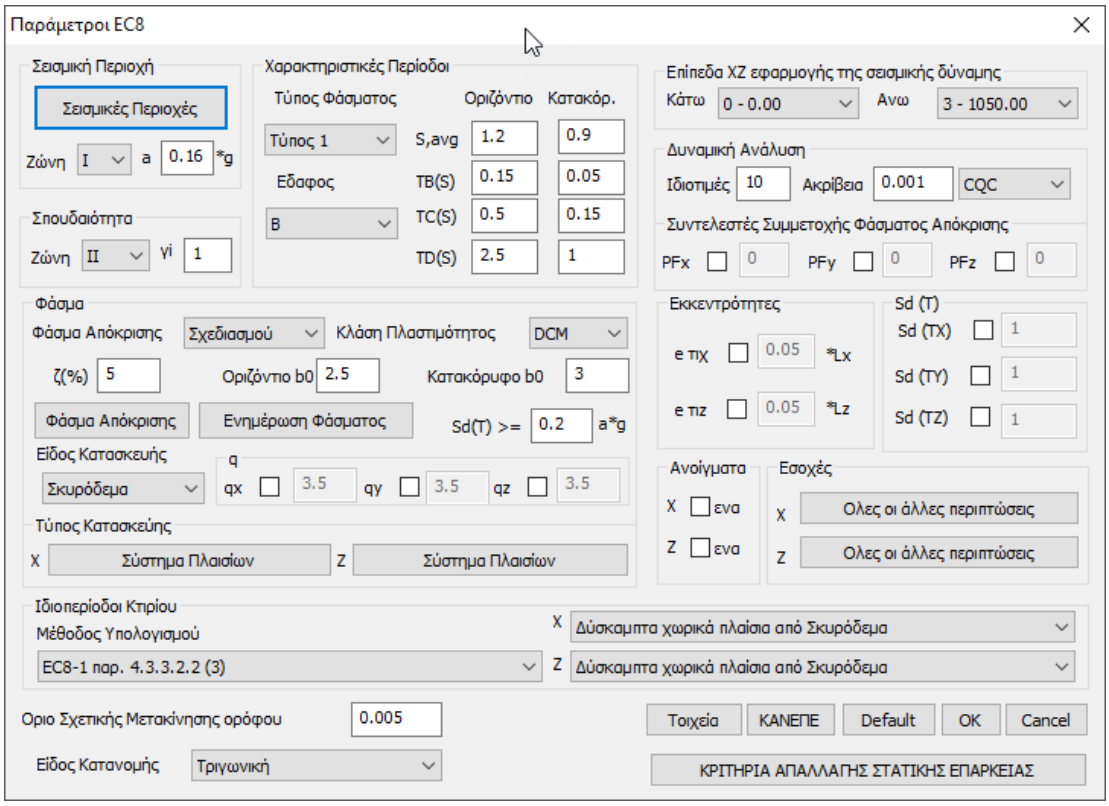
to update the parameters of the active script and delete the data of the previous execution process.

Then, select 

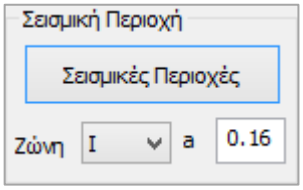
set the parameters of the specific study.

⚠ *Depending on the scenario you select, the configuration dialog box varies. In this example, having chosen the Eurocode 8 scenario, the dialog box will have the following format:*

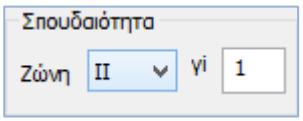
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



In this dialog box, enter the necessary information about the seismic area, the terrain and the importance of the building, as well as the earthquake application factors and levels.

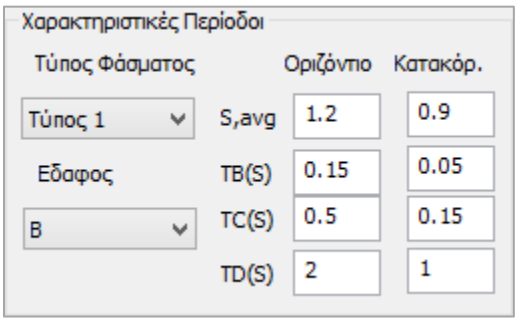


Select the seismic zone, after first checking the file that appears by clicking on "Seismic Areas" for the number of the zone corresponding to the municipality where your study belongs. Select the number from the "Zone" list and the factor "a" is automatically filled in.



Accordingly, you select the "importance category" automatically fill in the importance factor "y".

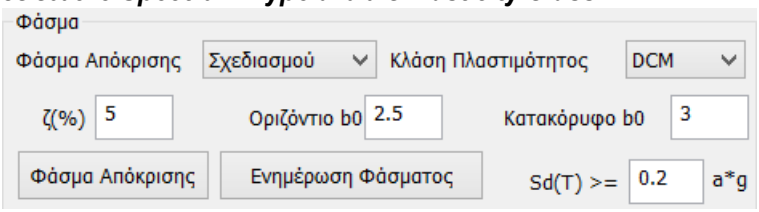
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



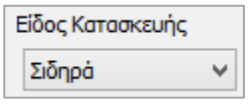
Then you define the type of spectrum (in Greece type 1 is used) and the soil category, so that the coefficients for the horizontal and vertical spectrum are automatically filled in.

⚠ *You can always modify the default values and set your own in all the parameter fields that are automatically filled in by the program.*

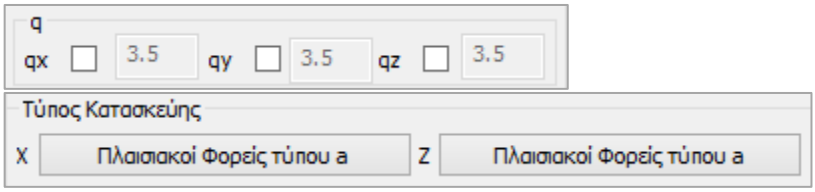
Select ***the Spectrum Type*** and the ***Plasticity Class***



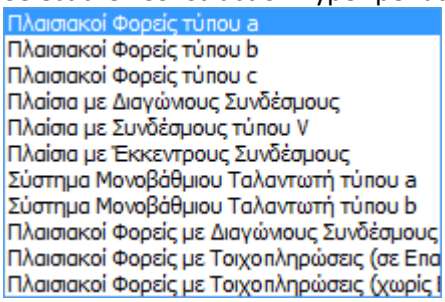
Select the ***Type of Construction***



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



Select the "Construction Type" per address by selecting from:

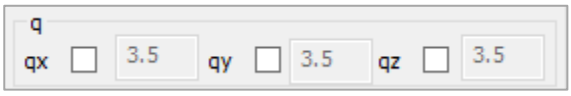


According to the Eurocode**, the "Seismic Coefficient q"** is derived from a calculation and the **"Type of Construction"** from specific criteria.

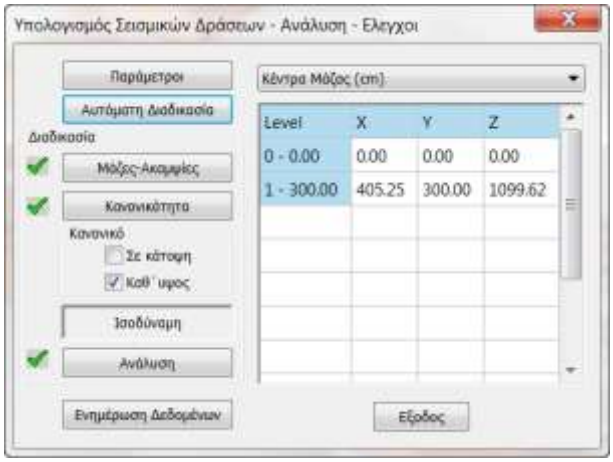
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

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

- After filling in all the previous fields, leave as is:

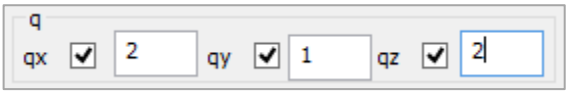


- Select "OK" and with "Automatic Process" perform a **first analysis**.



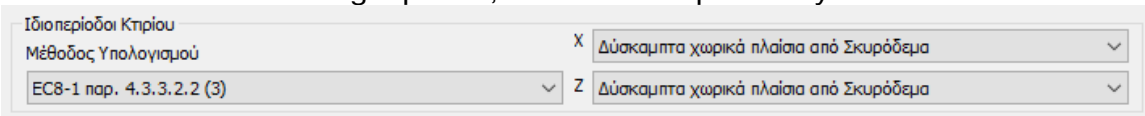
- Knowing all the previous parameters, the program calculates the "Seismic Coefficient q". Open the parameter dialog box one more time. In the "q" field read the values calculated by the program.

- You can proceed by keeping these values or modify them by checking the corresponding checkboxes and entering your own values (which you could have done from the beginning, but then the program would receive your values without calculating the EC8-based values).



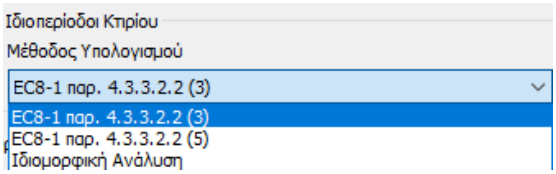
In the ***Building Properties*** field:

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 a choice of three ways to calculate the eigenperiod everywhere.

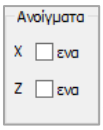
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



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



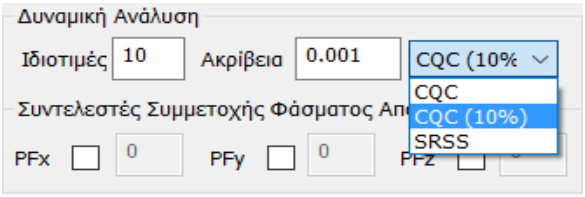
the corresponding checkbox in the "Openings" box is activated)

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).

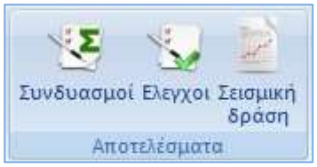
The user can increase or decrease the number of Idiosyncrasies, in case of dynamic analysis, and Static, in case the user chooses to calculate the eigenvalues from Idiomorphic Analysis, and the accuracy rate.



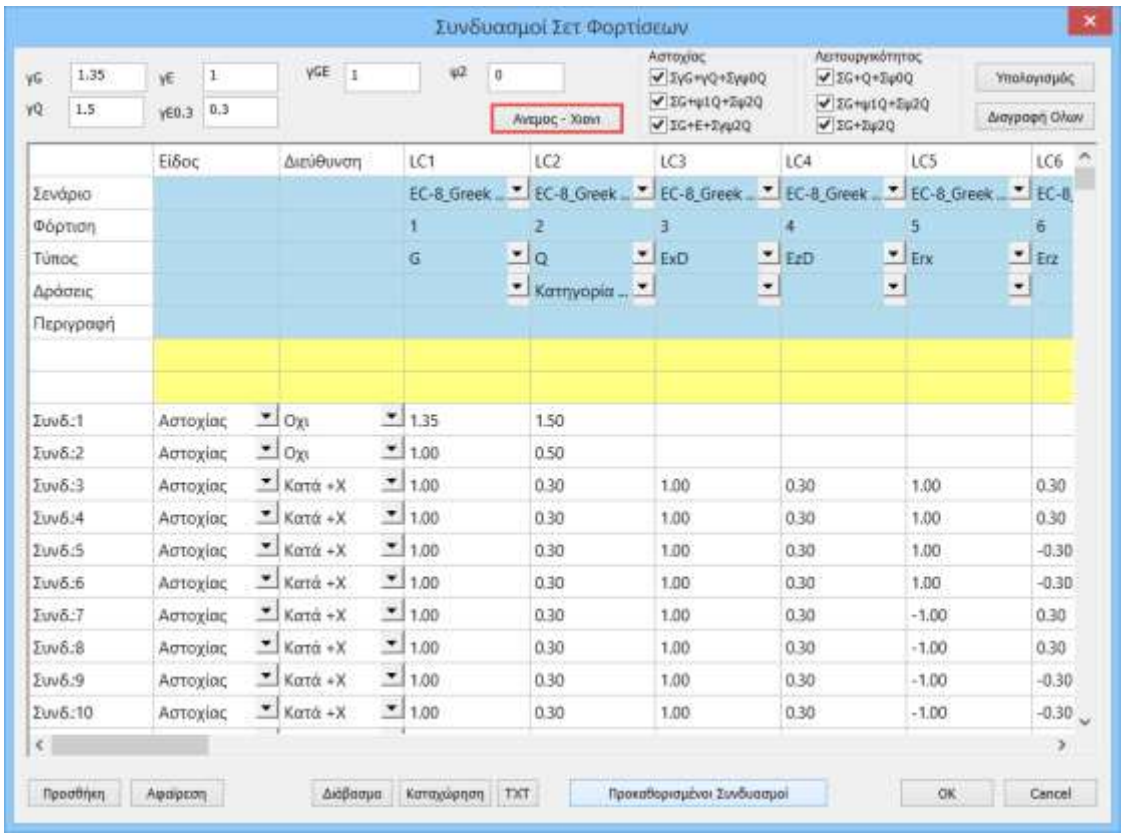
It is also possible to choose the mode of overlap of the eigenmodal responses either according to the Full Quadratic Parallelism CQC and CQC(10%) rule (3.6 EAK), or the Simple Quadratic Parallelism SRSS rule.
Also, the results of the seismic action now include the results of the eigenmode analysis for the static scenarios.

How to create combinations of charges:

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:

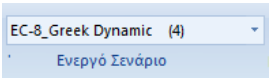


Selecting the "Combinations" command opens the "Load Set Combinations" dialog box where you can create your own combinations or call the predefined ones included in the program.

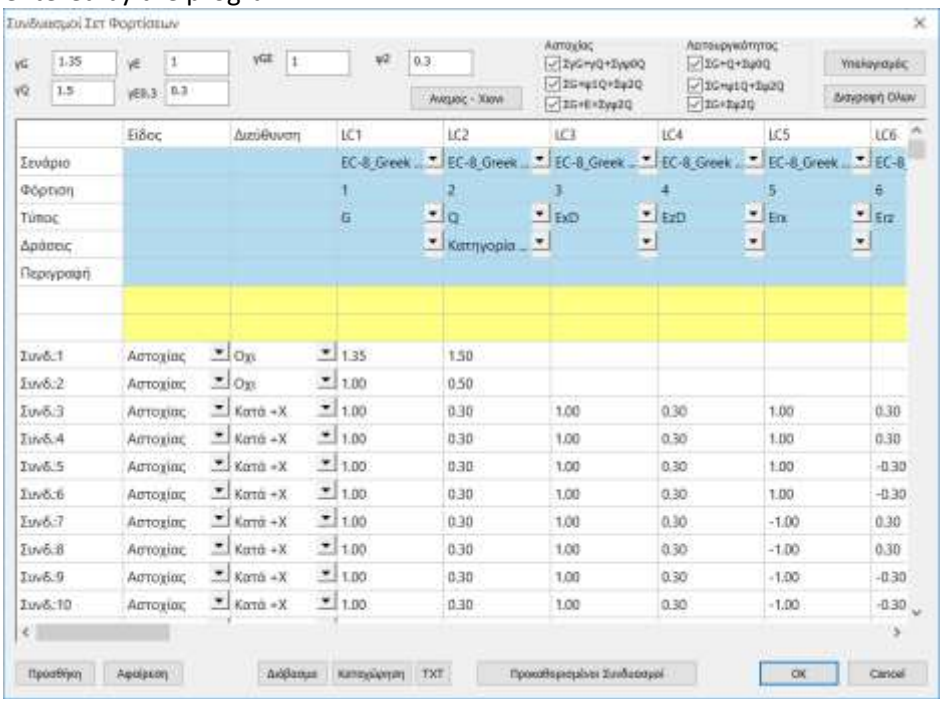


After running an analysis scenario, its combinations are automatically generated by the program. Calling the command "Combinations" opens the table with the combinations of the active scenario.

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



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

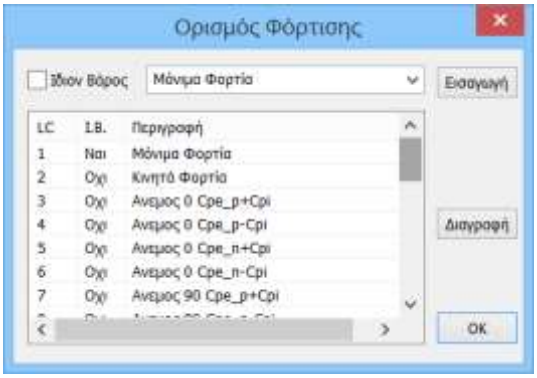


In addition to the predefined combinations, the designer has the possibility to create his own combination files, either by modifying the predefined ones, or by deleting all of them "Delete All" and entering his own values. The "Load Set Combinations" tool works like an Excel page offering copy, total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

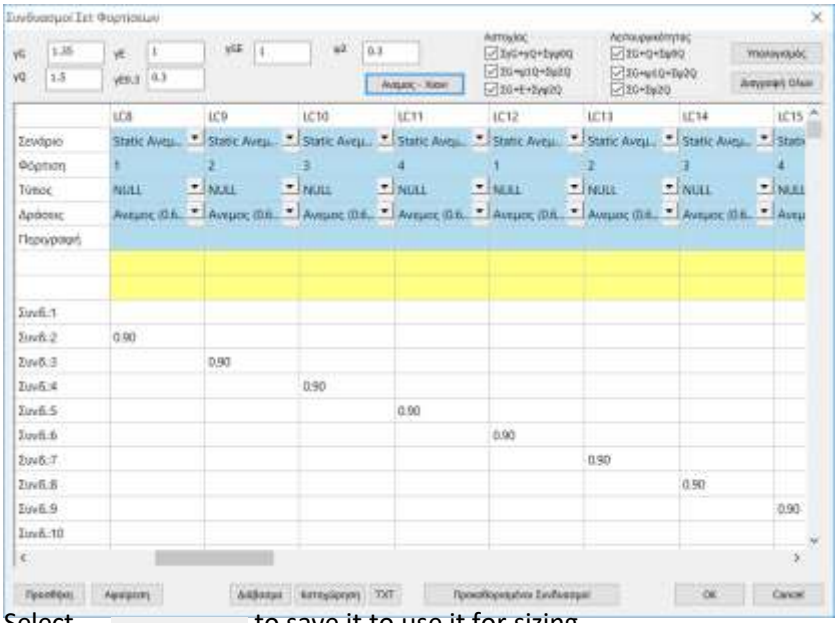
The predefined combinations refer to seismic scenarios. To create combinations of scenarios that do not contain an earthquake, both **automatic** and **manual** modes are available.

The **automatic** mode assumes that the automatic procedure for the calculation and distribution of wind and snow loads, the automatic creation of the loads and combinations, as in the example above, has been carried out beforehand.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



Subject to the above conditions, it is possible to create the wind and snow combinations automatically using the Ανεμος - Χιονι command.
So, after first running the earthquake scenario and all static wind and snow scenarios, with the earthquake scenario active, select the "Combinations" command. The combinations of the active scenario are automatically filled in. To automatically create the other combinations (wind and snow) press the button Ανεμος - Χιονι . The coefficients of the wind and snow scenarios are automatically filled in, providing a complete file of combinations of all the study loads.

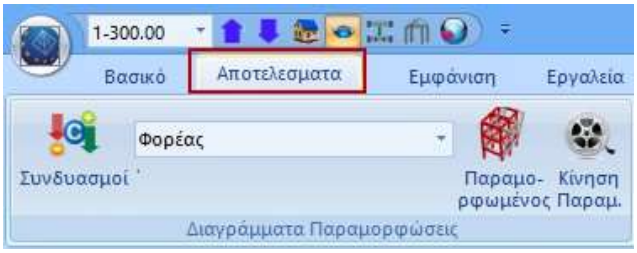


Select Καταχώρηση to save it to use it for sizing.

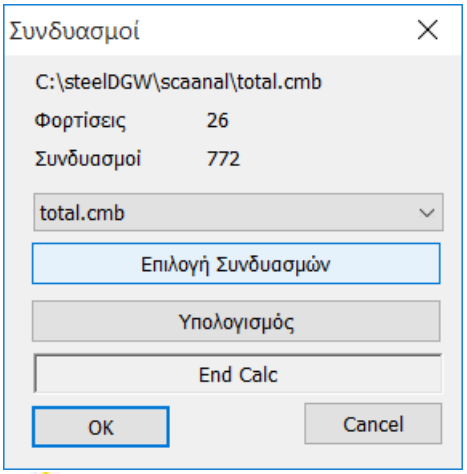
5. RESULTS

5.1 How to view diagrams and deformations:

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.



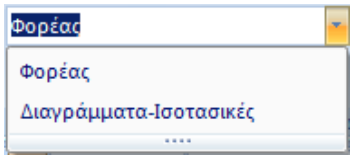
Depending on the results you want to see, from the "Combinations" command and within the dialog box:



- Select a combination from the list containing the combinations of all "running" analyses, and let the calculation be completed 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.



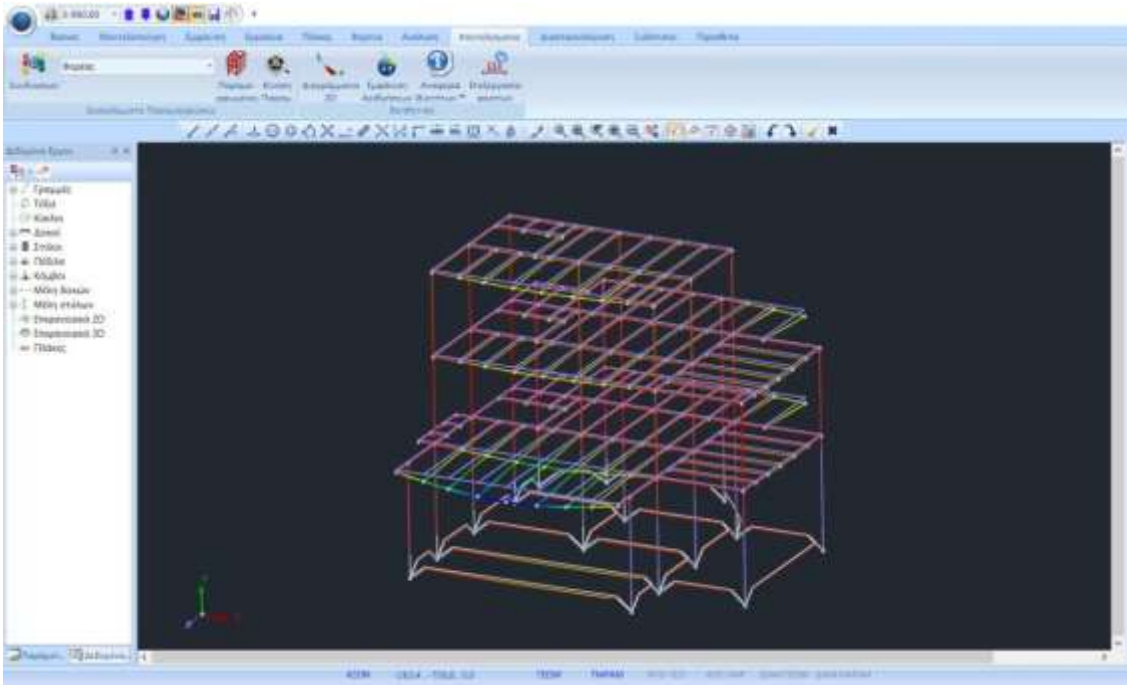
From the list on the right, depending on the results you want to see, select:

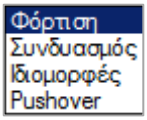
√ Institution or

√ Charts-Important

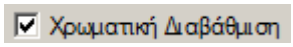
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

Operator + "Deformed Operator"

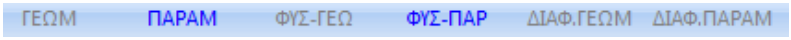


Select from the list 

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

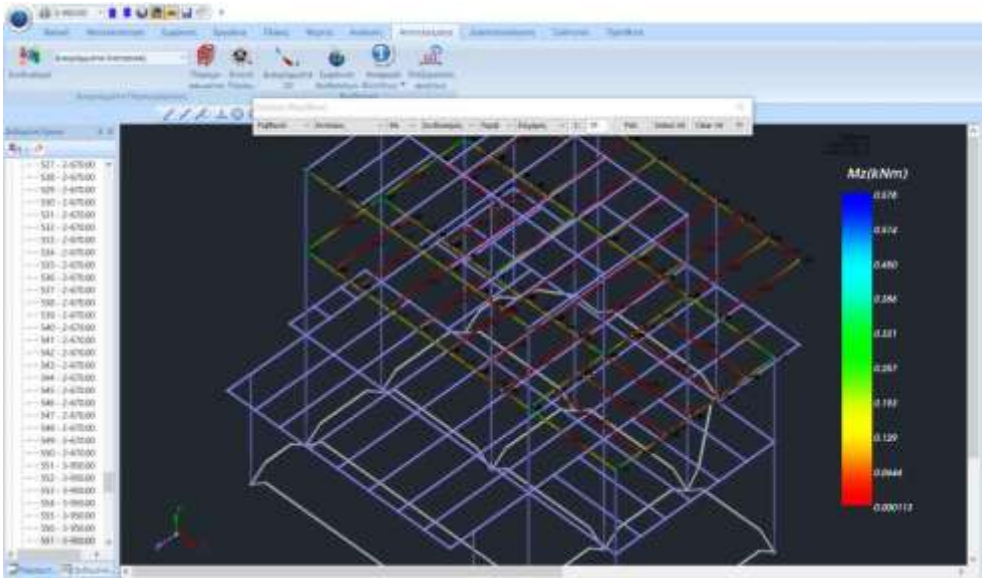
Activate 

, modify the "Scale" and the "Motion Step" to see the best and most intuitive visualization.
At "Status bar" select (double click, blue=active, grey=inactive) the way to display the deformed vector.

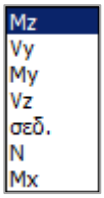


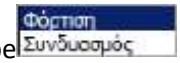
The "Motion" command is the switch that turns on and off the motion of the deformed vector, according to the choices you made in the dialog box of the previous command.

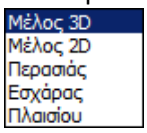
Charts - Equalisation :



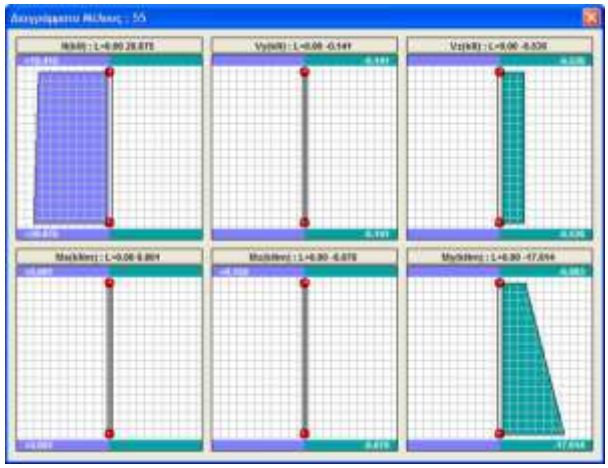
In this section you see on the members of the vector the diagrams of the intensive magnitudes for linear members, and the isometric stress, strain and reinforcement curves for finite surface elements. In particular, to view the stress diagrams for the Rigid elements, select the stress diagram for the Rigid elements from the list.

Size 

, then select the type of charge or the combination or combination or envelope 

and finally select the way to display the diagram 

To view all charts in a member. Select 2D Member and left-click, for example, on the bottom right sub-pillar of the 1st frame .

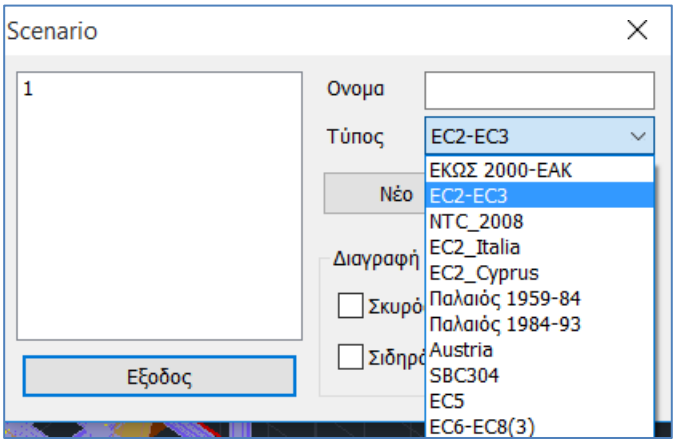


6. DIMENSIONING OF METALLIC SECTIONS

After you have completed the analysis of the structure, check the results and the deformations, the next step to complete the design is the dimensioning of the structural elements.

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).

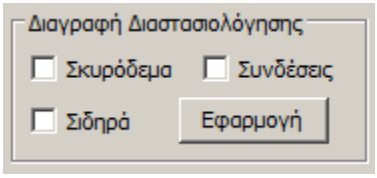


Type a name, select a type and New to populate the list of scenarios.

In this example, a Eurocode scenario was used.

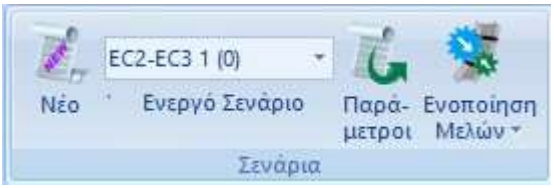
Comment: *For metals, EC3 is applied through the program and is included in all scenarios regardless, since there is no corresponding Greek regulation. The EC2 designation refers to the method of analysis as well as the method of dimensioning concrete sections.*

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



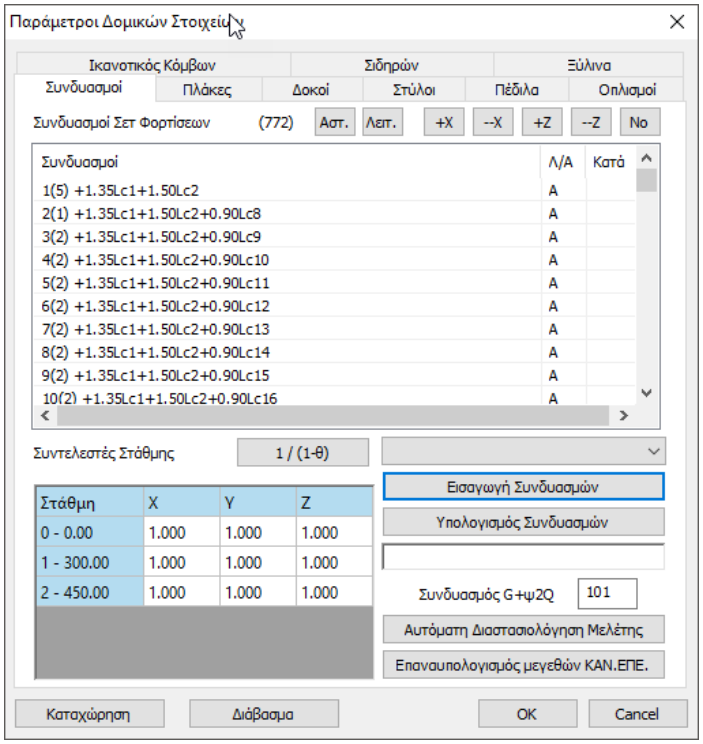
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

How to determine the parameters of the dimensioning of metal sections:

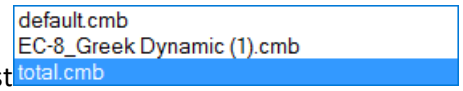


From the list of scenarios you have created, select the scenario you will use for sizing.

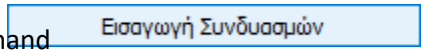
With the selected scenario active, you display the Parameters

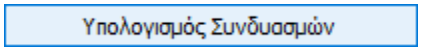


A prerequisite for sizing is the calculation of combinations.
The selection of the .cmb file of the combinations registered by the analysis is either:

- from the list 

with automatic calculation

- through the command 

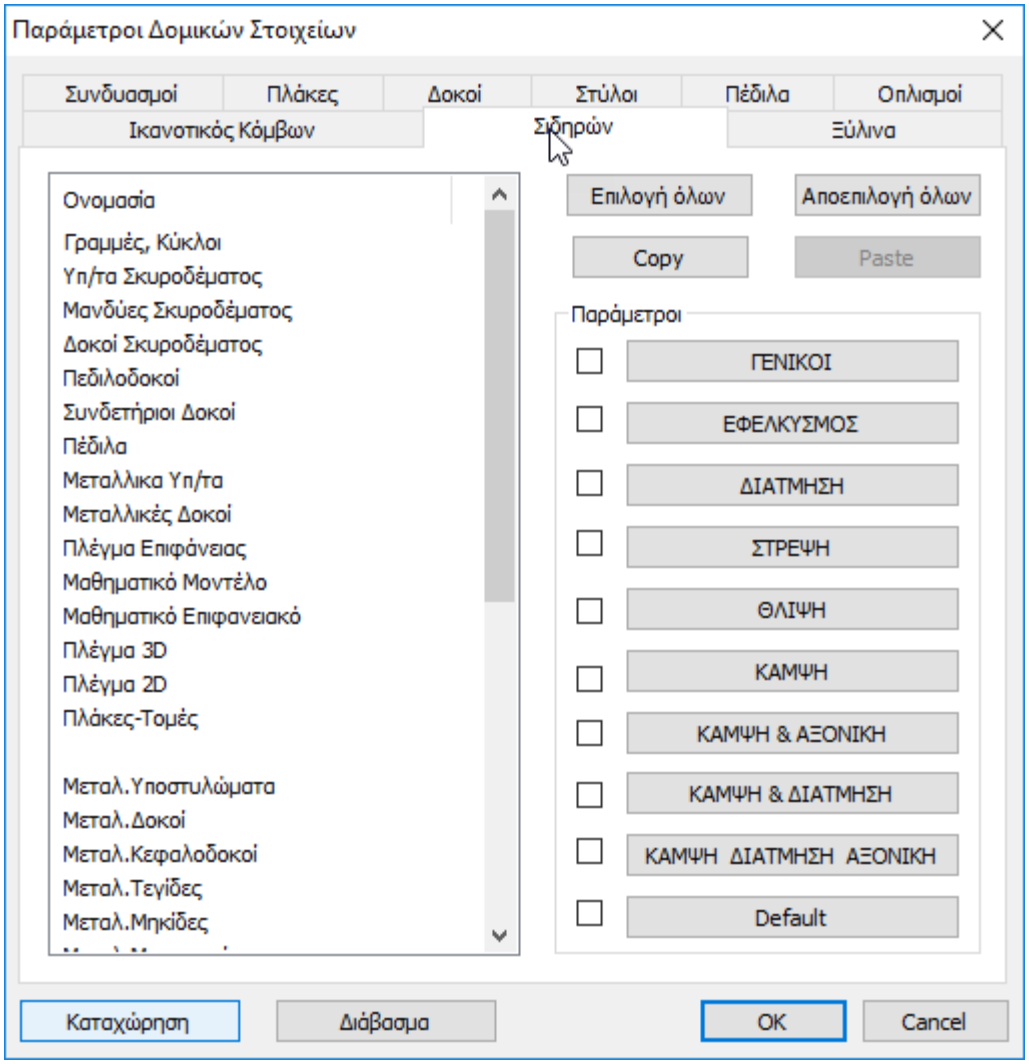
where, within the study folder, you select from the registered combinations the file of combinations with which you will dimension and then via the button 

button to make the calculation.

For the example, the file of combinations of dynamics with snow and wind, which was previously entered, was used.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

In the fields **slabs, beams, columns, slabs, reinforcements** you can specify various parameters for concrete sections.
For metal structures, to set the parameters related to the sizing of metal elements, select the "Iron" field.
The box that appears is divided into two parts: on the left is a list of all layers and on the right is a list of controls, each containing the corresponding parameters of that control.
First you select one or more layers with the help of "ctrl" , or all of them with the "Select all" button. Then you activate the checkbox of a control and select the corresponding key to enter the parameters.
The "Deselect all" button cancels the previous selection of layers.
Once you have set the parameters of one layer you can copy them to other layers using the "Copy" command. Select a layer and "Copy", then select another layer and "Paste" and the parameters of the first one are copied to the second one.



The definition of the sizing parameters of the metallic sections is done layer by layer.
Select the layer whose parameters you want to set (e.g. Metallic Lodges) and select the layer you want to set the parameters for.

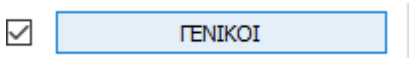
control category (General, Tensile, Shear, etc.), set the corresponding parameters. Once you have set the parameters for a layer, the program allows you to copy these parameters to another layer using the Copy and Paste logic.

For example, let's say you have set all the parameters for the Metallic Lumber layer and you want to pass these parameters to the Metallic Beams layer. You select the check box next to the "Default" option and all the parameter categories are automatically selected.
Then select the "Copy" button and select the Metal Beams layer and press the "Paste" button that has already been activated. Now all the parameters of the Metallic Lumber layer have been passed to the Metallic Beams layer as well.
An alternative method to set the same parameters for all layers that include metallic cross-sections is to select all layers with the "Select all" button and set the parameters for each control category once.

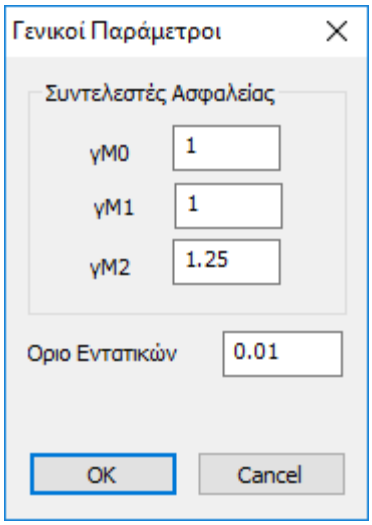
It should also be noted that to set parameters at least one (or more) layer must be selected.

The parameters for each control category are explained in detail below.

- By selecting the "**GENERAL**" section, the following dialog box appears:



to set the safety factors γM:



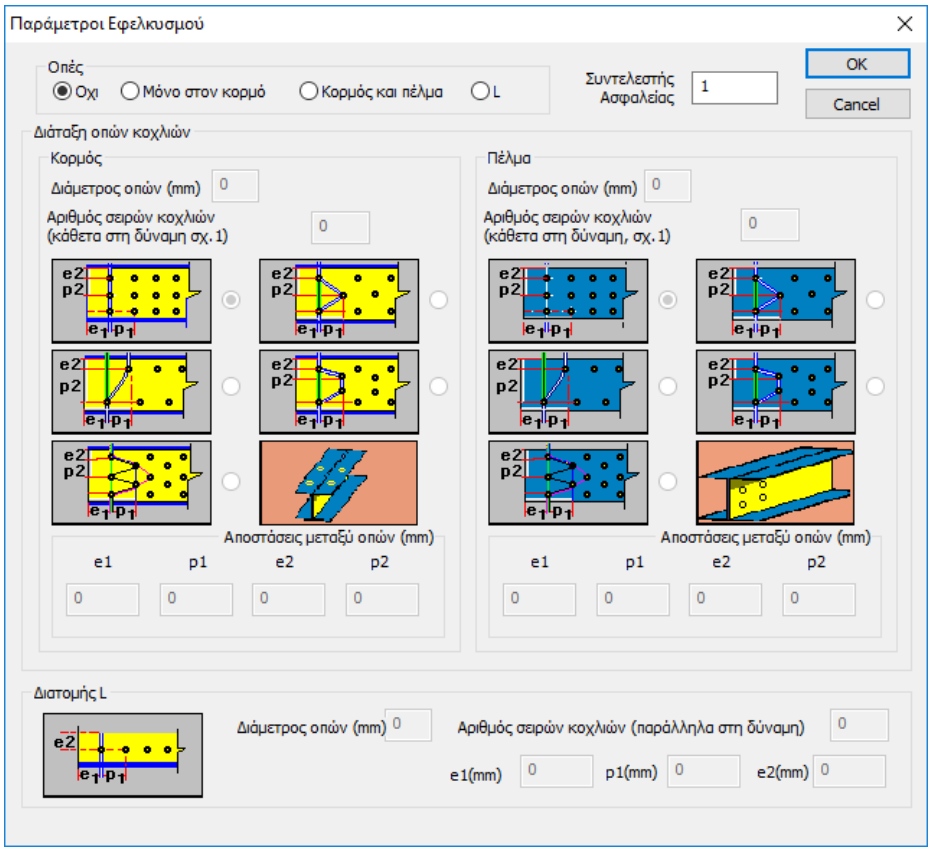
γM0= transverse stress resistance for each category of members γM1 = resistance to buckling based on tests cM2= tensile fracture strength of cross sections

Here you can set the individual safety factors and a minimum threshold for the intensive sizes below which the intensive sizes are not taken into account. The above values are those proposed by the Eurocode.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

- **ΕΛΛΚΙΣΜΣ**"

☑ ΕΦΕΛΚΥΣΜΟΣ set the "Tensile" parameters and check the position of the holes (EC3 chapter 1.8 §3.5):



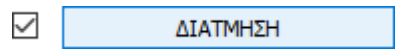
For the holes define the distances from the ends, the diameter and the number of rows on the torso and tread.
In the case of an L section, set the parameters at the bottom of the box.

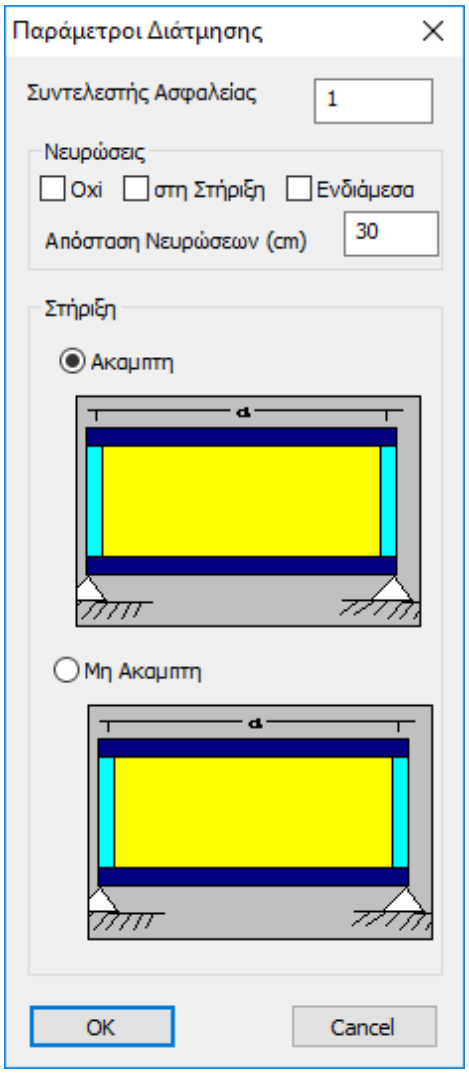
The rationale here is to specify whether the application during the tensile test take into account the bolt holes of the connections in order to account for reduced tensile strength of the cross-section. If you decide to provide data you will derive it, for the specific layer (e.g. Metallic Supports) from the corresponding connection checks you should have already performed. So the connection check must have been done before, before you can give data here.

The factor of safety for all checks is predefined and equal to unity, which means that the program calculates the ratio of the corresponding stress to strength and if this ratio is greater than unity, failure occurs.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

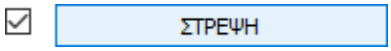
- "**DISTRICT**"

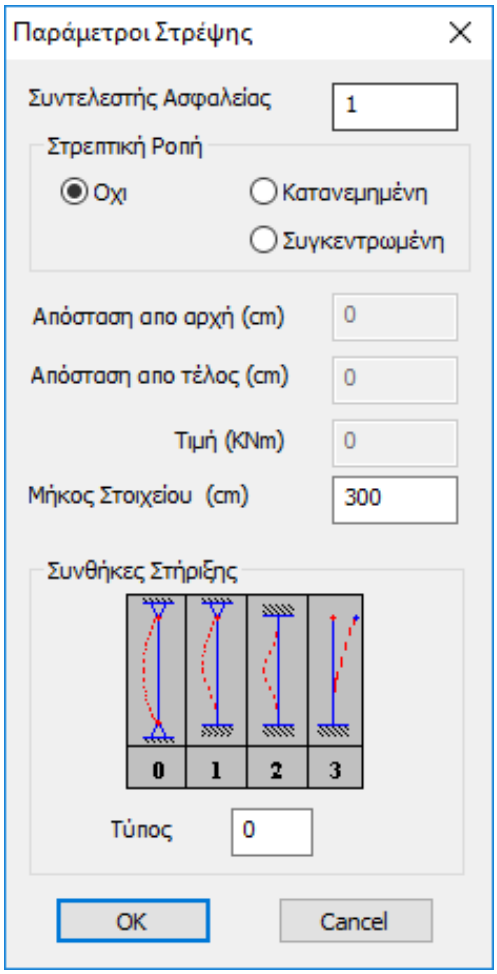




Here you define whether the elements of this Layer have ribs or not and if they do, where they are present (on the support and/or the trunk). You also define the spacing of the ribs as well as whether the support of an element is rigid or not.

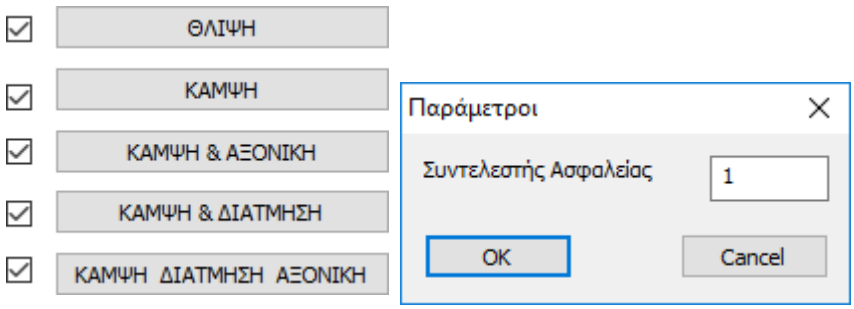
- "**STRIP**"



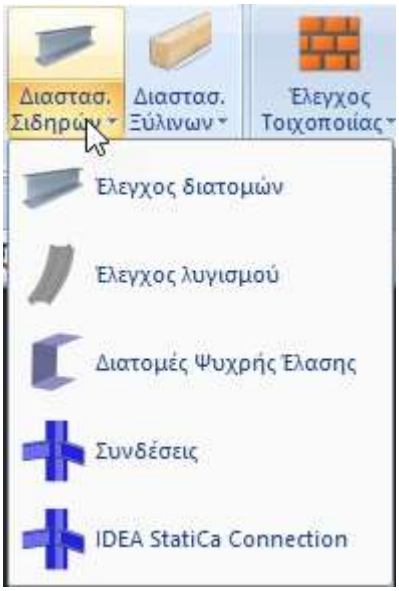


Here you specify whether the members of the layer are loaded by torsional moment (distributed or concentrated). If they are loaded, you define the elements of the loading. You also specify the support conditions of the members based on the support type shown in the graph.

For all checks set the "**Safety Factor**", i.e. the ratio between the design value and the corresponding resistance value. The default value is 1.



Dimensioning of steel sections:



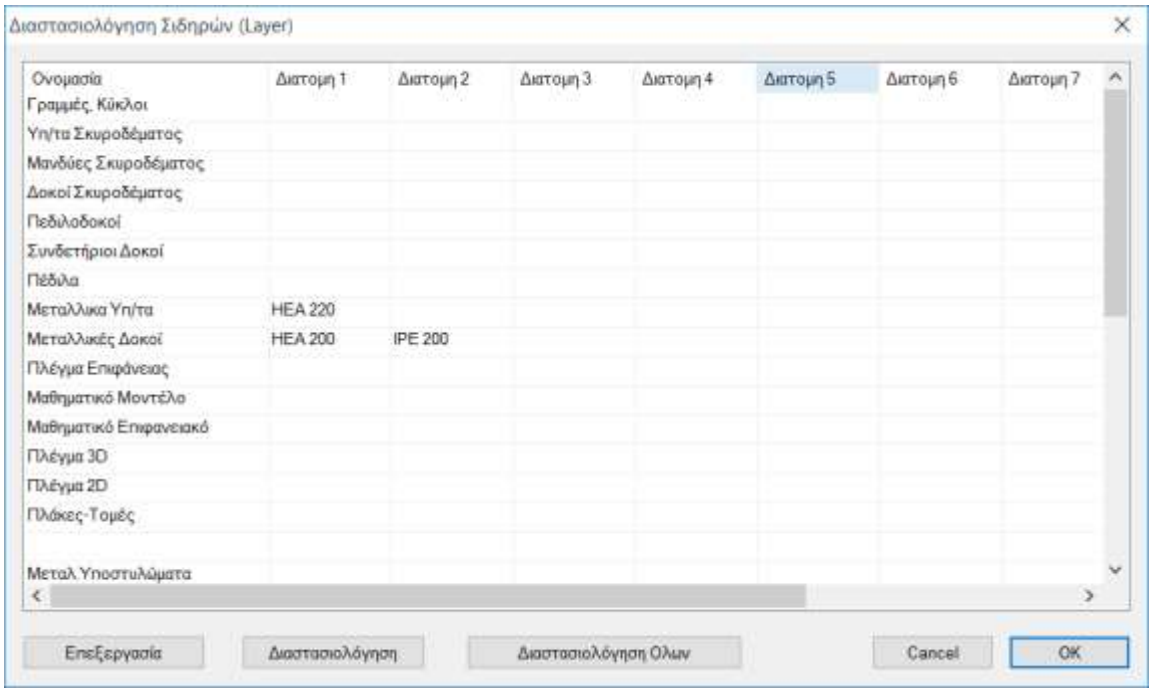
In the "Dimensioning" module, the "Iron" field includes the commands related to the solution of the metallic sections with adequacy check, buckling check and the check of the connections.

6.3.1 Control of metallic sections:

The Cross-section check option is used to check the adequacy of metal cross-sections.

Using the command, the following dialog box appears.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



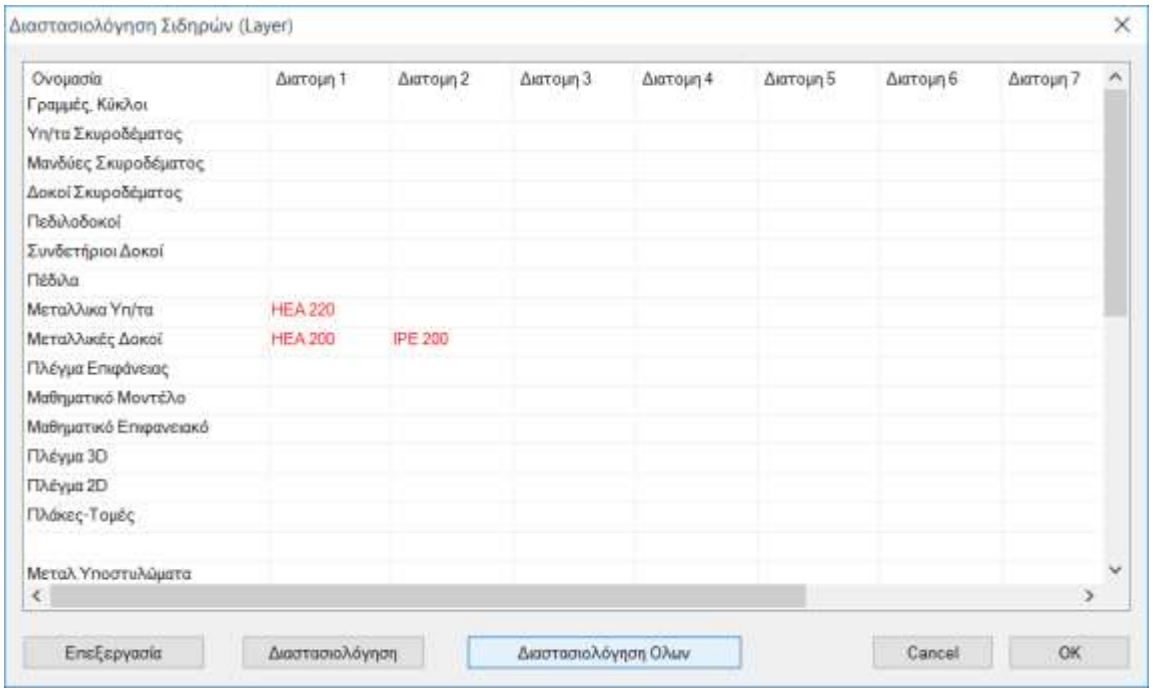
The cross-section check is done globally for all elements in a layer.
For each intensive quantity, the program identifies the element with the worst value for that quantity.
The first column is the layers that exist in this study and in the following columns are the types of metallic cross-sections that exist in these layers. In this particular example, the metal supports of the structure with cross-section HEA220 have been placed in the layer "Metallic Supports". Correspondingly, the metal beams have been placed in the "Metal Beams" layer and the corresponding members in the other layers.

⚠ *In SCADA Pro you are given the opportunity to create your own layer thus grouping the types of cross-sections.*

For example, you could create a layer called "Metal columns Prominent" and place all the columns on the left side of the building on it. The logic is like that AUTOCAD : similar objects in one layer. In this way you will notice that you can more easily and massively dimension steel sections and members. You could use the same technique to create a layer and include only one element in it. So you can dimension only that.

By selecting the command "**Dimension all**" the cross-sections for all combinations will be automatically checked and the groups - layers in which no cross-section fails will be displayed in green and the groups in which even one cross-section has exceeded the unit, i.e. has failed, will be displayed in red.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



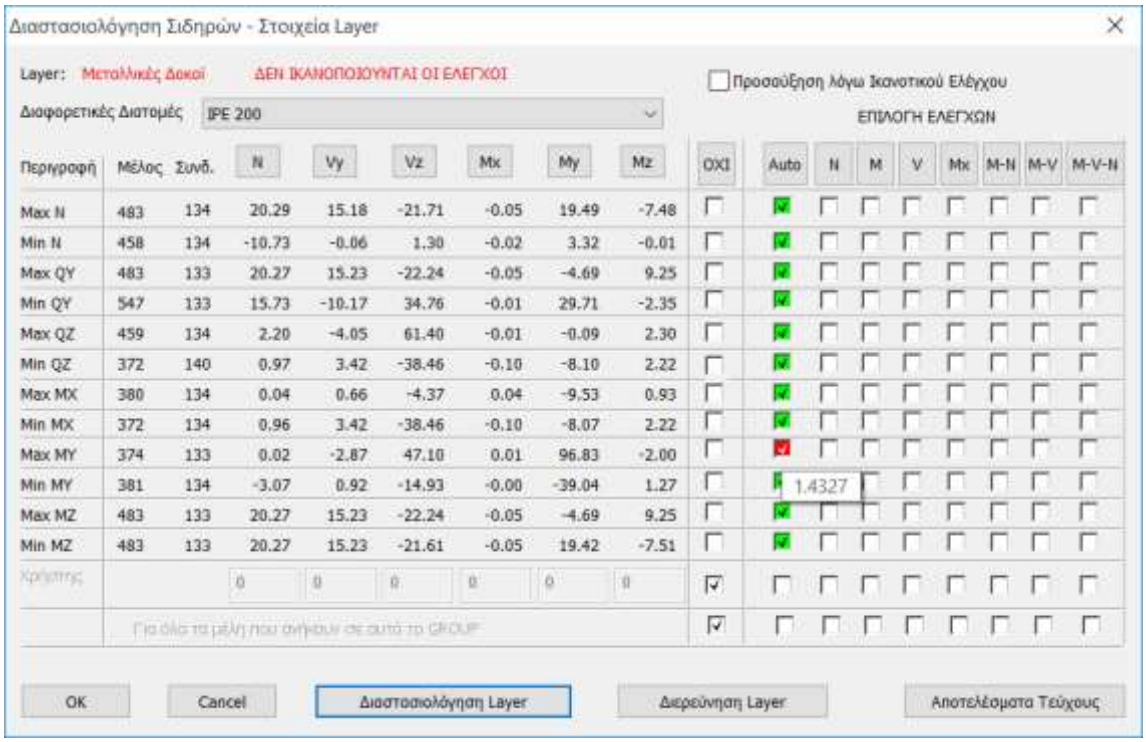
The fact that a layer is shown in red does not mean that all members of the layer have failed. To see for example which columns have failed you should select the "Steel Beams" layer and then select the "**Edit**" command.

In the window that appears, you can see in tabular form the results of the cross-sections of the selected layer with colours and values.

In the automatic procedure, the program finds the 12 worst combinations of all the members of the vector (Max N with the corresponding 6 group of intensities, Min N and so on) and performs the check. (see Manual Chapter Sizing).

When you touch your mouse cursor over a red cell then the value displayed will be above unity (miss).

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

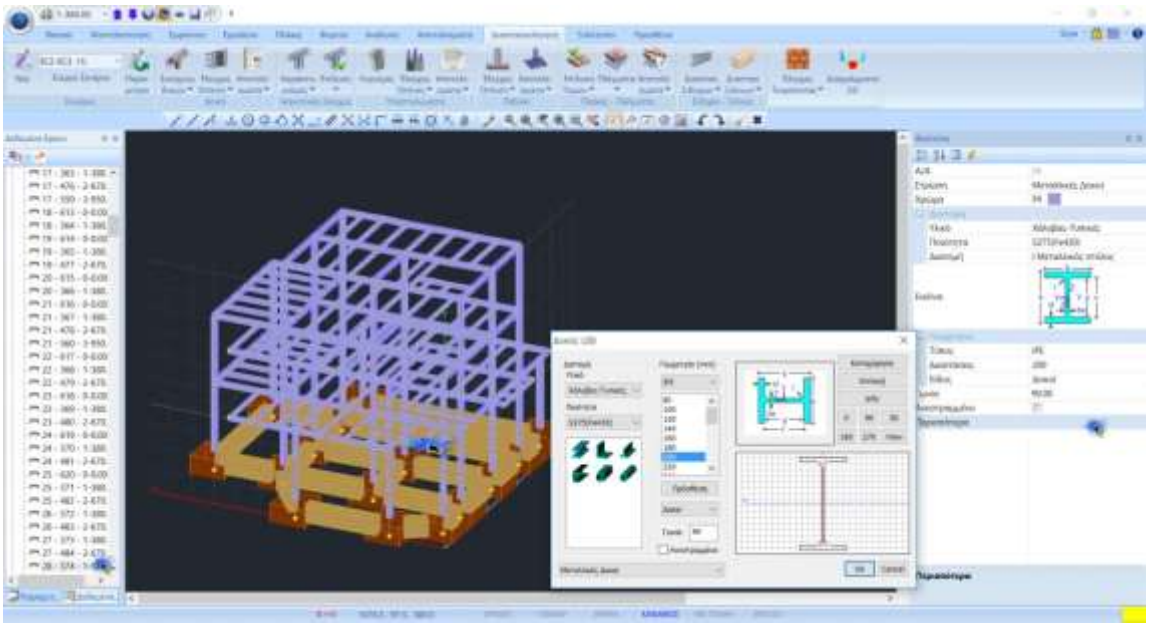


In general, in addition to auto you can choose for which individual intensities or combinations of intensities you want to perform the respective checks.

You also have the possibility to do sizing by entering your own intensive sizes through the "user" option.

The results of the dimensioning in summary, (either from the automatic process or from the selection of the individual intensities, or by the user) can be viewed by clicking on the command "Issue results" or in detail by clicking on the command "Layer Investigation". The files displayed are also the ones produced by the program for the generation of the calculation issue.

In cases of failure you intervene in the design, either by changing the cross-section of the failing members, or by adding additional elements, e.g. windbreaks to "relieve" the failing members.
In each case you will have to calculate the new intensities and therefore run the analyses again.



If, for example, you want to change the cross-section of a particular element, you numerically locate the physical member that misses within the tree and select it to display it graphically in red on the screen and open the Properties list on the right, where the "more" command displays the window with its physical properties. To change the cross-section of the beam it is sufficient to define a new one.

⚠ **NOTE**:
The check of the cross-section will be done based on the new cross-section but with the same intensive sizes if you do not run the analysis scenario again and simply recalculate the combinations in the Dimensioning parameters field.

6.3.2 Buckling control of metallic sections

This mandate is used to control the members. That is, checks are performed for each member belonging to the specified layer:

Limit State of Failure

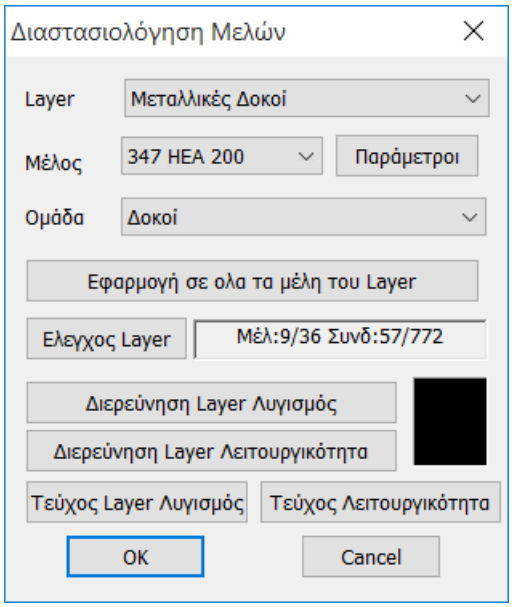
- Check for bending (lateral) buckling due to axial compressive force
- Torsional buckling check due to bending moment.
- Check for torsional bending due to the simultaneous presence of axial compressive force and bending moment.

Limit State of Functionality

- Member deformation control
- Edge (node) movement control

Using the command, the following dialog box appears:

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



The check is done per layer.

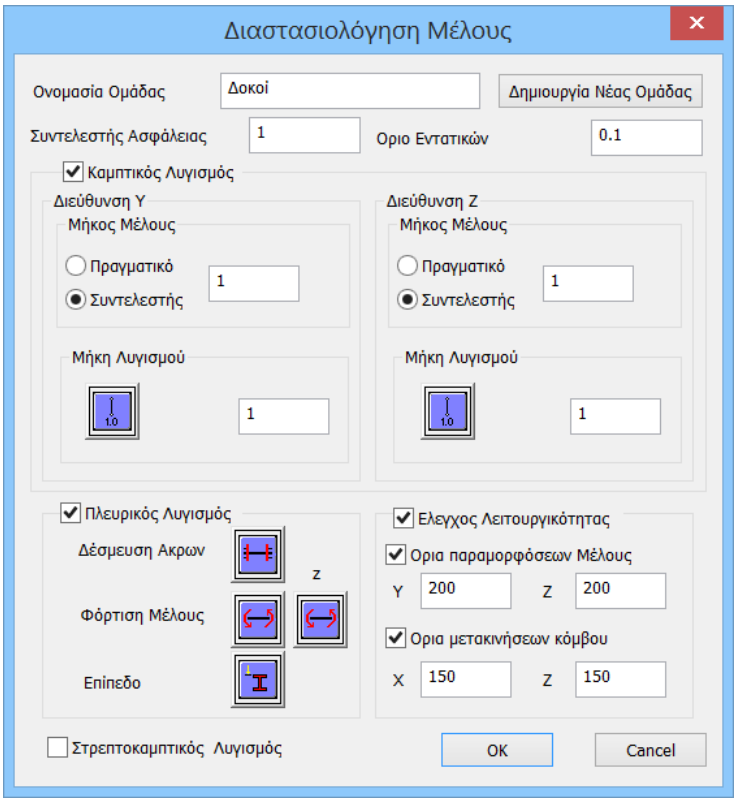
By selecting the layer, all the members of that layer appear in the "Member" list and their cross-section.

The first step in dimensioning the layer is to define the dimensioning **parameters.** Because it is possible that for some of the layer members you may want to define different parameters, it is possible that within the same layer you can define different groups of parameters to which the layer members belong. The program has two default parameter groups.

In Configuration window, in the "Group Name" field is the name of the configuration group. If you want to create your own group, enter a new name and press the "Create New Group" button.

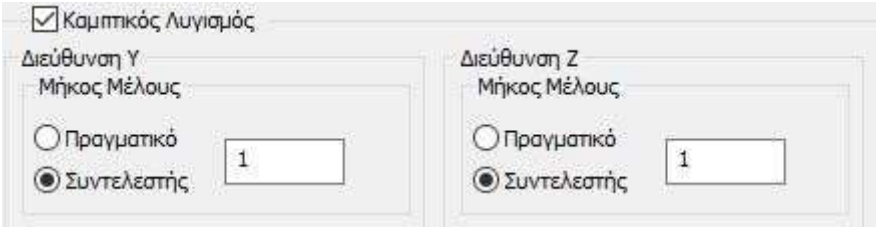
For this example, select the parameters shown below:

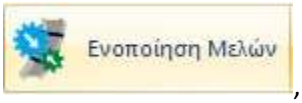
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



In this way the program will calculate as the buckling length (**bending buckling** field - direction Y and Z) the length that has been given to the member geometrically (i.e. the length of the member from start node to end node).

⚠ If instead you chose to set the bend length to an actual value you could enter a value in the "actual" field in meters. Then the program would take as length what you would arbitrarily set -manual.



⚠ In earlier versions of SCADA Pro and before the creation of the command 

, the user was asked to set the length of the member and the bending length in both the Y and Z directions respectively, following the procedure below:

In "**Length of Membership**":

- if you select "*Actual*" you must enter in the field the actual length of the member in m.

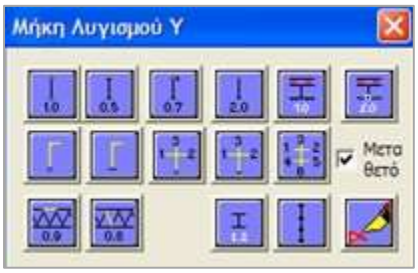
- if you select "*Coefficient*" you must enter a coefficient by the different lengths of the members belonging to this parameter group will be multiplied.

If you want the program to take the actual lengths of the members into account when checking the bending bend, select "Coefficient" with a value of 1.
If you have some members with different or equal lengths that are laterally secured at the same distances (e.g. at 1/3), then you give the value 0.33 and of course create a separate parameter group to which these members belong.

The next parameter is the member's **buckling length** which depends on the support conditions of the nodes of the member ends always within the buckling plane.

⚠ **OBSERVATION**

If there has been a Consolidation, then the Bending Length refers to the Consolidated Member.



From the "**Bend lengths**" field you can select the support conditions of the member via the icons. Depending on these, the program calculates a coefficient, e.g. for a double-jointed member = 1.0. Here you select 1,0 in both directions.

Then click on the side bend field to have the program perform the corresponding check. Here you need to describe the "**Edge Binding**" , the "**member loading**" form by y and z, and the "**loading level**" .

⚠ *For a detailed explanation of the icons please refer to the corresponding paragraph of the Manual Chapter 9 Sizing.*

Also, to perform the functionality check and torsional bending, click on the respective fields.

⚠ The parameters for torsional are the same as those you gave flexural flexure.

After you have entered all the parameters, you are returned to the previous window. Here if you select the command "**Apply to all members of the layer**" then the parameters you set before for member 61 will be applied to all members of the corresponding layer, i.e. the metal beams.

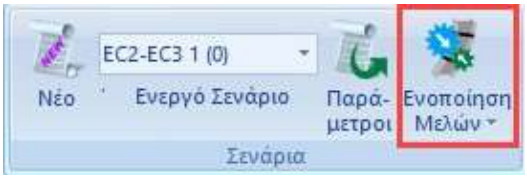
⚠ You could create layer groups by typing the name DOKOI 2, selecting different parameters and finally clicking on "create new group". This way the members belonging to the "metal beams" layer would have parameters either from the "beams" group or from the "DOKOI 2" group depending on the assignment you would make to each member.

Then you select the "**Check layer**" command and calculation of all the members for the combinations you have already defined starts and at the end a green or

red square which if you click on you will see the ratios resulting from the bending checks of each member.

- The results of the buckling tests can be summarized by clicking on "Buckling Layer Investigation" or in detail by clicking on "Buckling Layer Investigation". The files displayed are also the ones generated by the program to create the calculation issue. The same applies to the functionality checks.

6.3.2.1 Consolidation of Members



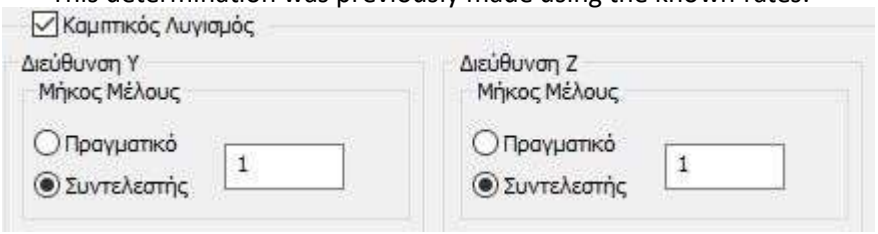
In the new version of the program a new group of commands has been added which concerns the consolidation of metallic members for the calculation and display of bending and deformation checks based on EC3.

(See User Manual § 10.c Sizing).

IMPORTANT NOTES:

- With the use of this tool, it is now possible for the designer to correctly define the initial length of the member per direction to be taken into account in the checks of bending. 

- This determination was previously made using the known rates:



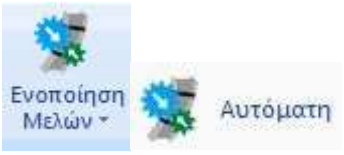
- Now with the use of the consolidation by direction, the process of the rates will not be needed, but the consolidation will be done, in most cases automatically.
- It should also be noted that with the consolidation process the bending length is correctly calculated and in the printing of the results a consolidated member is now printed once with an indication of the individual members it includes.
- Basic concepts of buckling about strong and weak axes and what the corresponding buckling lengths ly and lz mean can be found in chap. Sizing of the user manual.

NOTE

As a general rule, we could say that, we obtain **unified length Ly** in the direction where the local y-y axis is parallel to the supporting elements -

ensure the member, while in the other direction, if no data are available, **the individual lengths** are taken as **Lz**.

We select the consolidation command group and select the Automatic command:



The logic of the consolidation methodology is that, either automatically or manually, the individual members of an element are consolidated by bending direction.

The buckling length taken for calculation purposes is not the actual length of the member, but the unified length from the beginning to the end of the column or beam respectively.

In addition, in the presentation of the results, for these consolidated members the worst controls are shown only once and not for each one as was the case until now.

Finally, in automatic consolidation, there is the definition of stop levels.

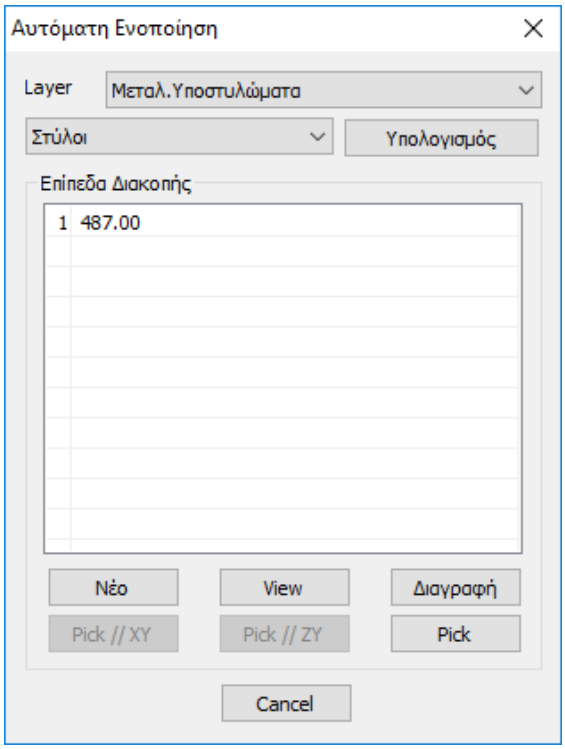
Break planes are horizontal or vertical planes that are used as breakpoints in the consolidation of a continuous element.
Thus, for vertical elements (Pillars) the stopping levels are horizontal levels which are defined, like the levels, with an altitude.

⚠ **OBSERVATION**
It is good to work on the 3d mathematical model and have the local axes displayed.

6.3.2.2 Automatic Consolidation

Using this command displays the following dialog box

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



In the upper field you select the layer of the elements you want to consolidate. The

Consolidated members are displayed in colors on the screen.

- The y-y consolidated local data are shown in yellow.
- In cyan colour the z-z consolidated local
- In pink are the consolidated along both axes

Right below you specify the type of element contained in the selected layer. The program automatically understands the type of element if it is vertical (Pillars) and all other elements are Beams.

With the "**Calculate**" command the program consolidates the data of the specific layer based on the above mentioned.

Interruption levels are levels that are boundaries of the beams and poles where the integration for either one or the other direction is to be interrupted.

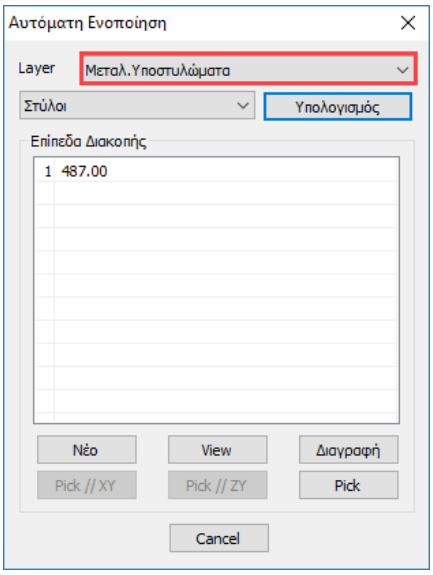
- For the poles, the stop levels are horizontal planes defined by the altitude.
- For beams, the stop planes are always vertical planes defined by two points.

Predefined limits:

- for the horizontal planes the horizontal planes are the foundation level and the upper last level (the last level).
- and for the beams are the vertical limits of the girder.

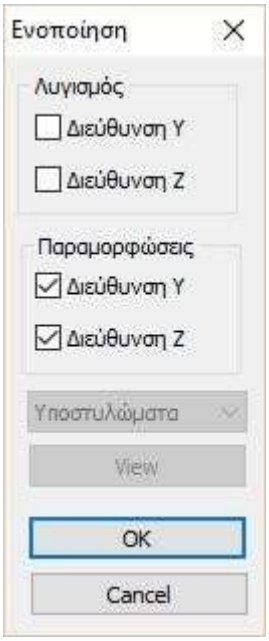
⚠ The default limits are not shown in the table of cut-off levels.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



6.1.1.1 User Consolidation

Select the command and then point to the start and end points of the members you want to consolidate.
Selecting the second point (end point) displays the following dialog box:



where you set the direction of unification for Bending and Deformation.

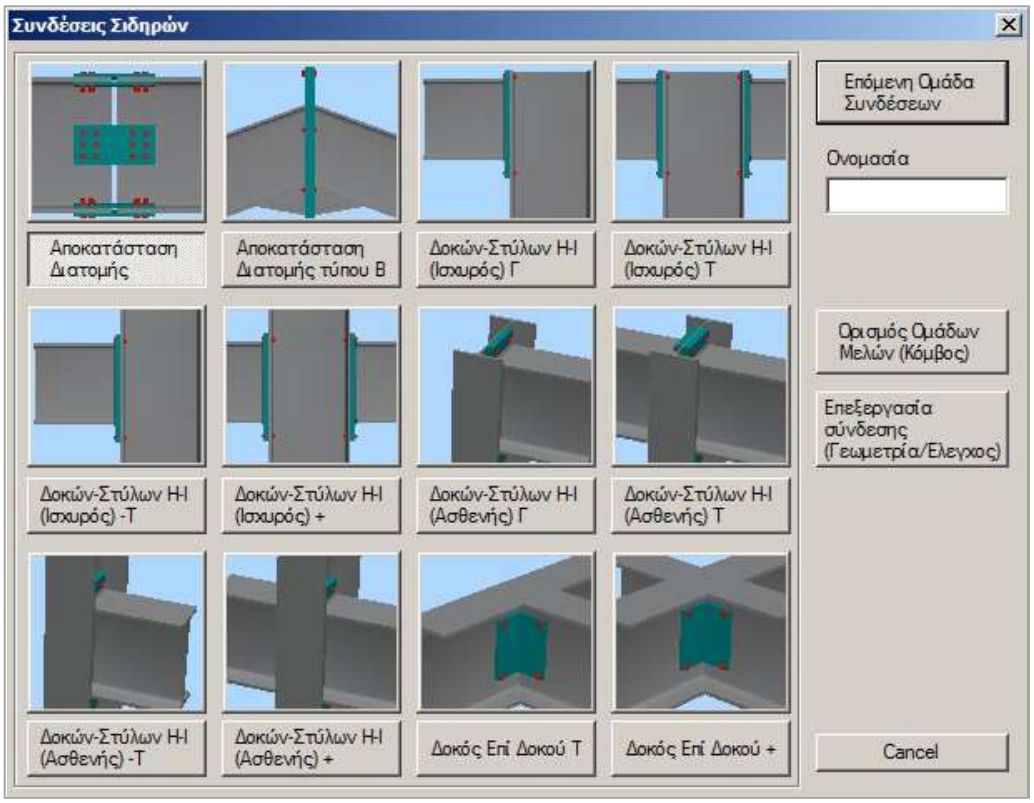
(See User Manual § 10.c Sizing).

7. Dimensioning of connections

How to dimension the connections of the metal members:

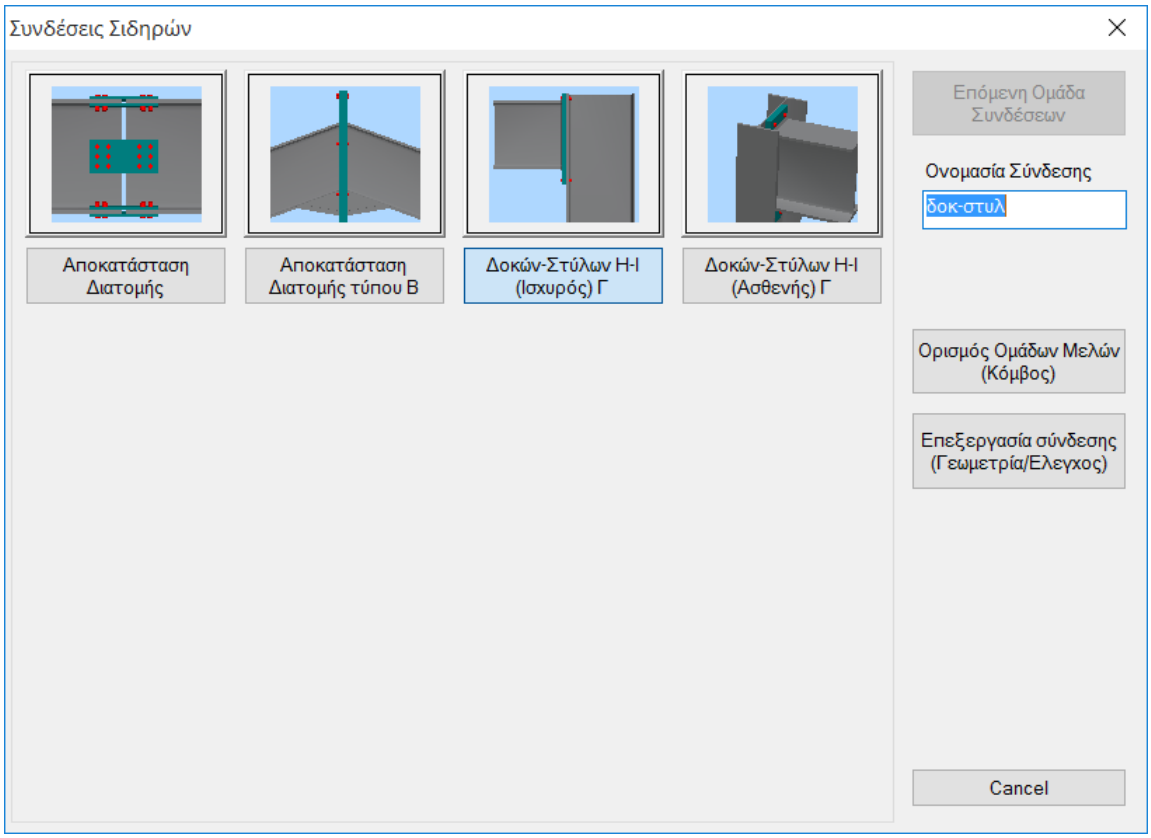
The last chapter of dimensioning for steel structures is the dimensioning of the connections of the structure. Select the command and you have two options proceed with the sizing of the connections:

A) Click on the "Connections" command and then right-click on the space (desktop) to display the library with all the available connections
from where you can choose the one you want.



B) Alternatively, you can click on the "Links" command and then left-click on the members you wish to link. Right-clicking then displays a window in which only possible connections consisting of only two members are included.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

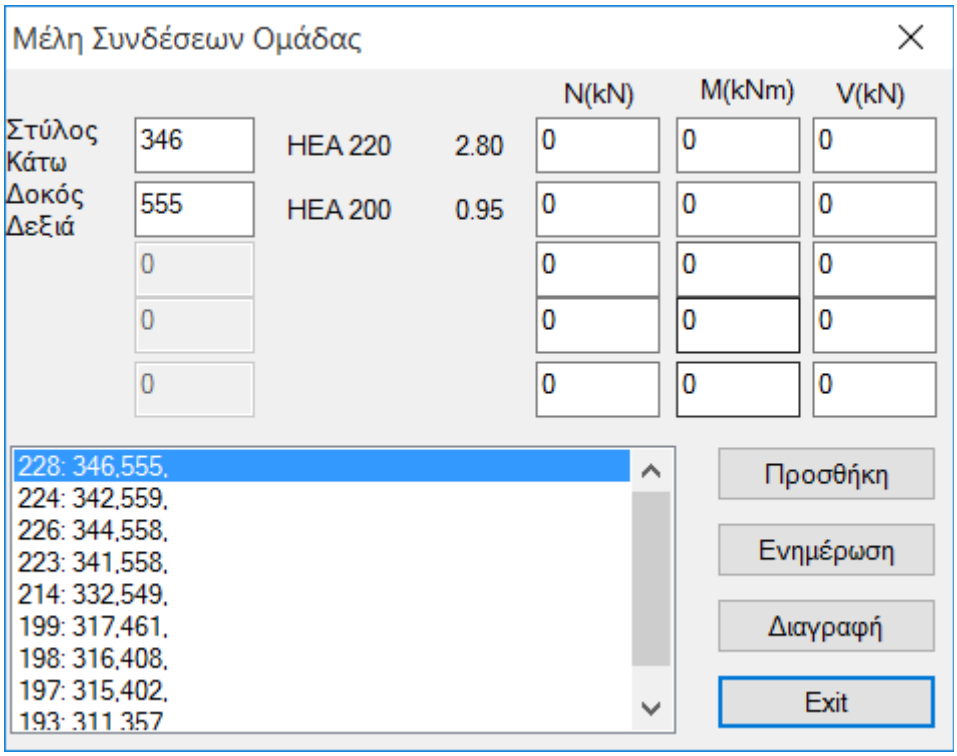


For example, select member 30 (column) and member 154 (beam) in sequence. Right-click to display the window with the 4 possible connection types. Select the last (to the right) connection which corresponds to a Beam - Column connection of cross-section type H or I on the strong axis. You will then enter a name for this connection.

⚠ The name must be in roman characters and there must be no spaces between the words.

Then select the "Define member groups" command and in the dialog box you can add other similar cross-section pairs (column - beam) or add your own values for the N,M,V intensive sizes to the existing pair. To add other similar pairs, click on the "Column Bottom" field and then select Subcolumn 24 on the desktop. Similarly then click on the "Beam Right" field and select beam 153 (or just type in the corresponding member numbers in the fields if and when you know them). To add your selections click on add.

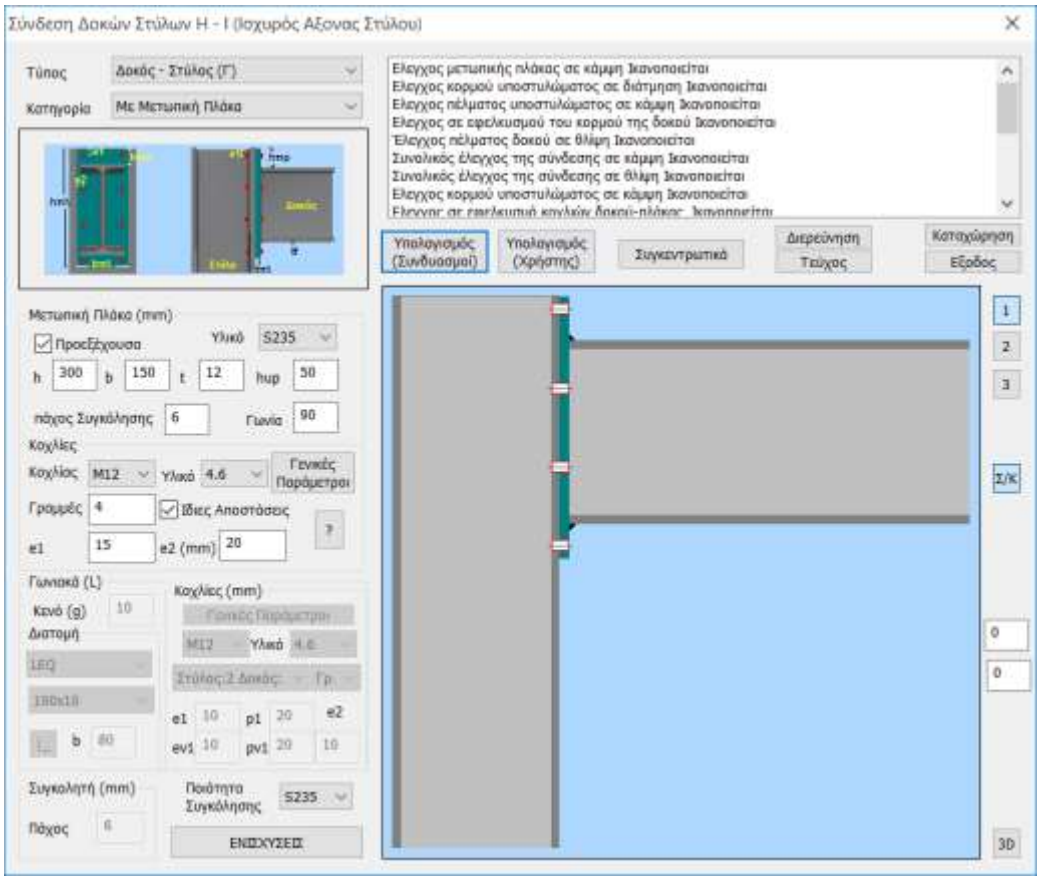
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



Essentially way, you can massively size all of the
connections of the beam members of the girder which are connected to the weak axis in the same way (bolts or welds, plate geometry, etc.) and which have common
cross sections. The program will automatically calculate the intensive sizes of each pair and proceed to dimension the connection based on the most unfavourable combination. This way you will not have to guess where in your structure the most unfavourable beam-post connection in the strong axis will be developed, while at the same time if one connection is satisfied, all other connections of the same type will automatically be satisfied.

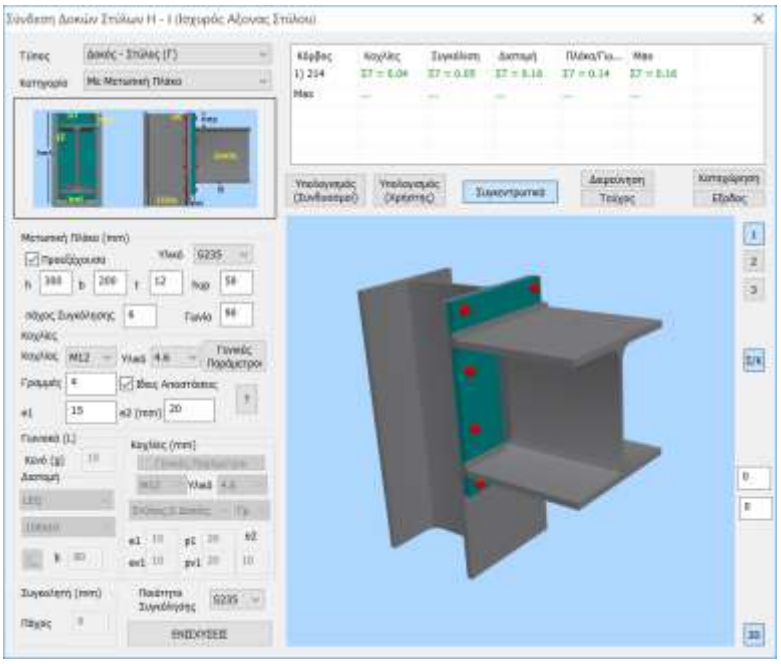
Then select "exit" and then "Edit Link-Geometry Check".
The window appears automatically, through which you can precisely define the type
and geometry of the specific connection.
Give the typical values shown in the figure or try to create your own connection.
To then check the adequacy of the link with the combinations in the analysis, select the "Calculate (Combinations)" command. Initially the program will perform a geometric check of the connection (e.g. if the bolts are too close to the edge of the plates). If there is a problem an error message is displayed accordingly in the top right field . In this particular connection change the distance e1 from 10 to 15 cm and click again on "Calculation (Combinations)".

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



If you click on the 3D command (bottom right) you will see a three-dimensional representation of the connection which is dynamically updated as you make changes to the parameters. Buttons 1, 2, 3 correspond to side view -1, side view -2 and plan view -3 and via the S/C command you can display the welds and bolts in the 3D view.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



When the checks of the geometry and topology of the connection are satisfied, the program will make all the required calculations and display all the checks performed according to Eurocode 3 for the specific connection.
You can see the aggregated results in the corresponding field. There, sufficient reasons will be displayed in green font and the failures of the link in red.

If all the checks are satisfied, the program will be able to proceed with the registration of the connection and the automatic generation of the drawings. Otherwise the process is interrupted and then you will have to change some values of the connection to continue. In the investigation as well as in the issue you can see in text format the results of the tests in detail or in summary. Finally, click on the entry and exit to return to the connection types window.

8. FIELD SIZING

How to size the sandals:

Once you have completed the dimensioning of the connections, you can proceed to dimensioning the pedestals.

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

Select the command "**Check Arming>Total**" to do 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.

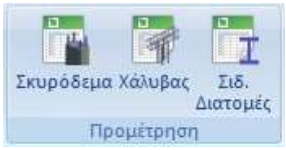
 The skirt missed. The type of failure is also indicated as a symbol above failure indication. The failure indications are respectively the letter "Z" which means failure at limit load, the letter "e" which means failure due to load eccentricity and the letter "s" which means exceeding of the developed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

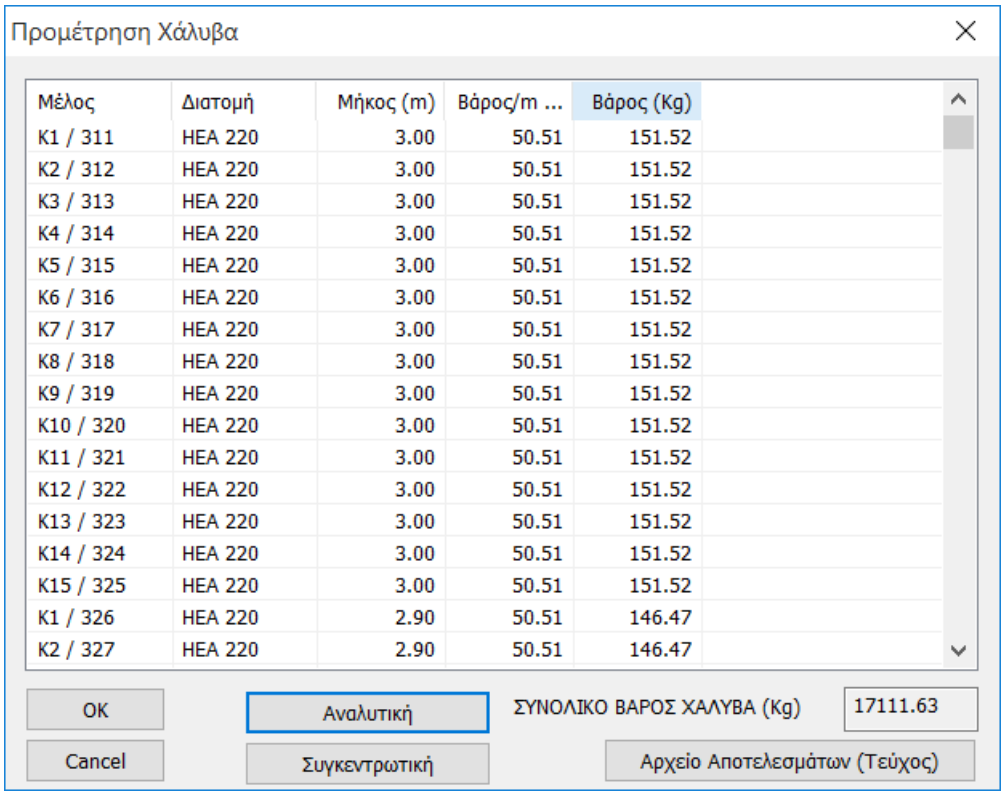
As is well known, seismic intensities derived from dynamic analysis are unlabeled because they result from the superposition of the eigenmodal responses. In the diagrams and wherever there is a necessity to superimpose them, they are always used with positive values. And the dimensioning of the elements there 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.

9. PROMOTION



In the "Extras" section the "Premeasurement" group contains the commands for the premeasurement of the design materials.

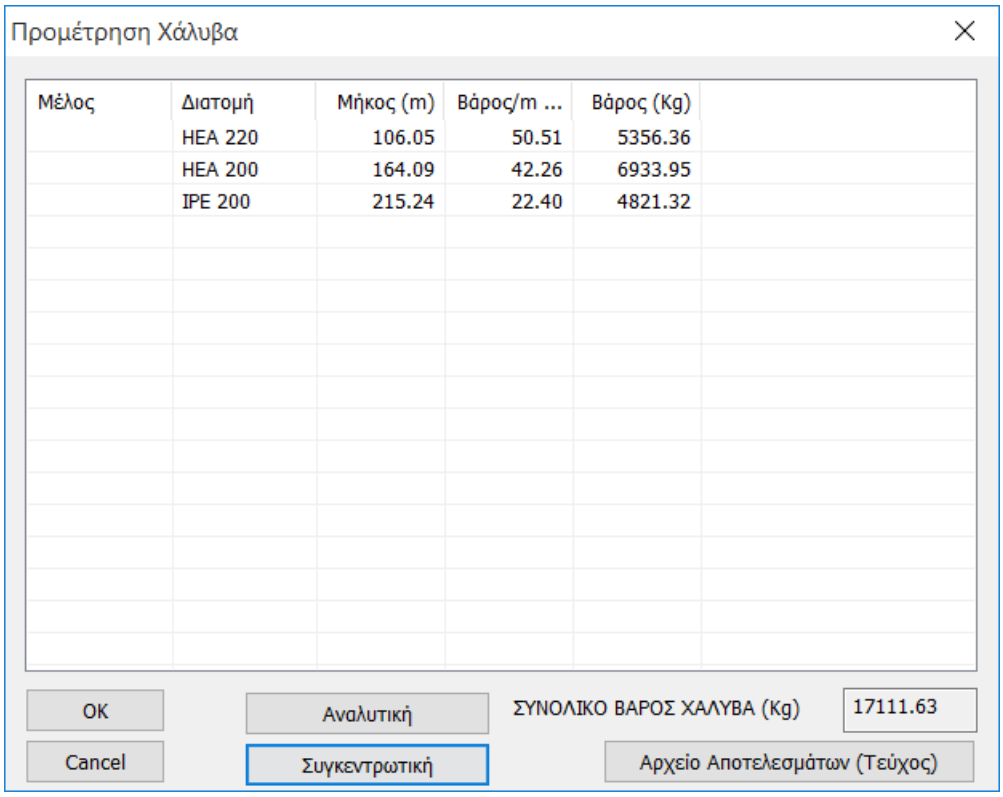
Iron Sections Select to display the dialogue box of the premeasurement of metallic **sections** either in detail: per member and section with reference to length, weight/m and weight in Kg, or in aggregate: per section and in total.



SCADA Pro enables you to have a detailed premeasurement of each steel section by member or an aggregated premeasurement by section category.

The results of the premeasurement either analytical or aggregate (steel or concrete) can be attached to the calculation book of the structural design by selecting the corresponding command, as mentioned in the Manual chapter "Add-ons".

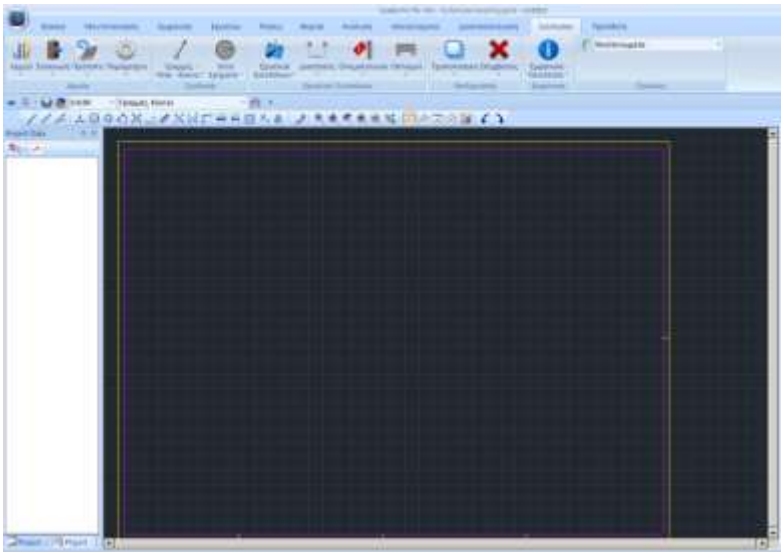
EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



10. DESIGN

After completing the dimensioning of the carrier and the creation of the connections for the metallic ones, in the Timber Formwork Module you enter, modify and finally create the drawings of the formwork and its details.

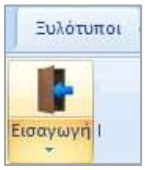
By selecting the "Wooden Forms" section, the drawing paper frame displayed on the desktop.



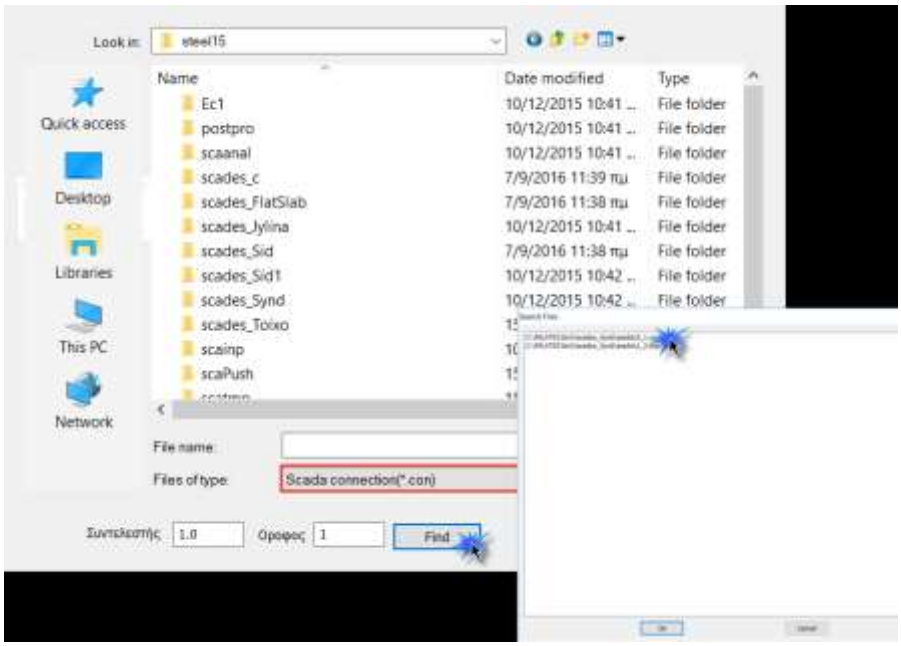
10.1 How to enter the plans of the links:

Plans of the registered connections are in the study file, specifically on the route:
C:\scadapro\ "Study" \scades_Synd\sxedia

And you open them within SCADA Pro design environment with the command:



EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

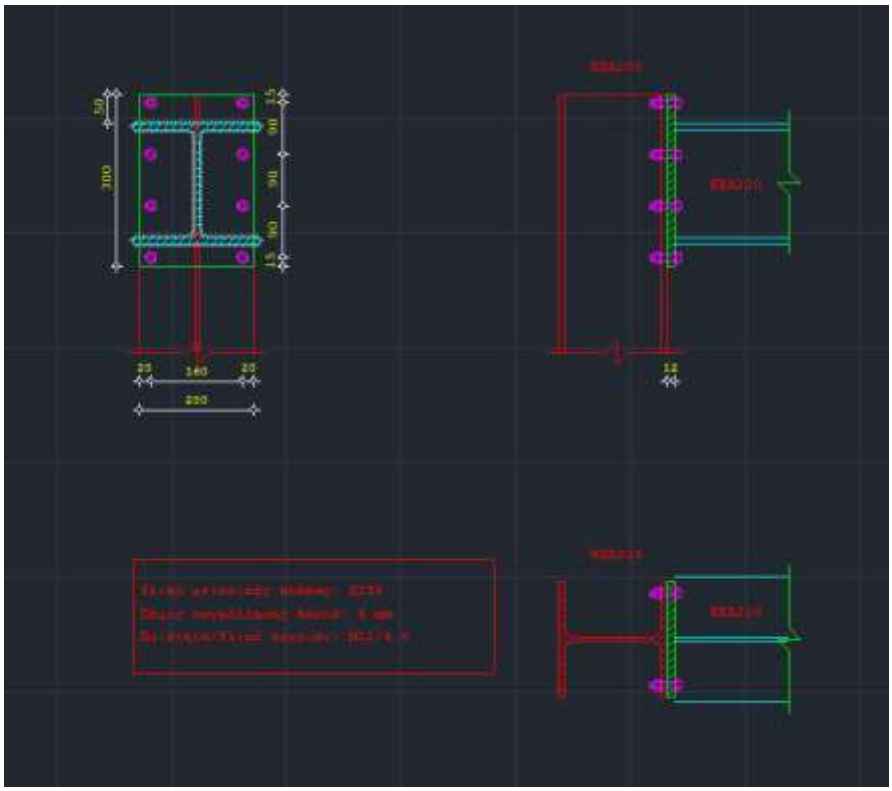


In the dialog box:

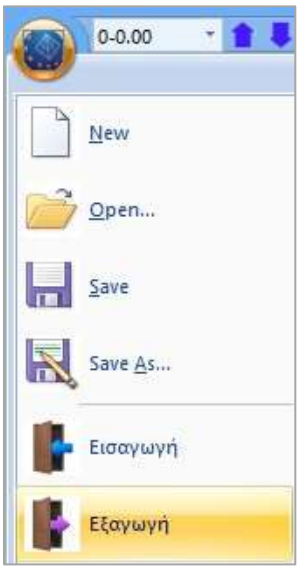
- in Files of Type select **Scada Connection(*.con)**

Then select the name of the link (so that it turns blue), then "ok" and finally click on the desktop where you want to insert the drawing. This will automatically create a floor plan and two views of the link detail you selected

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

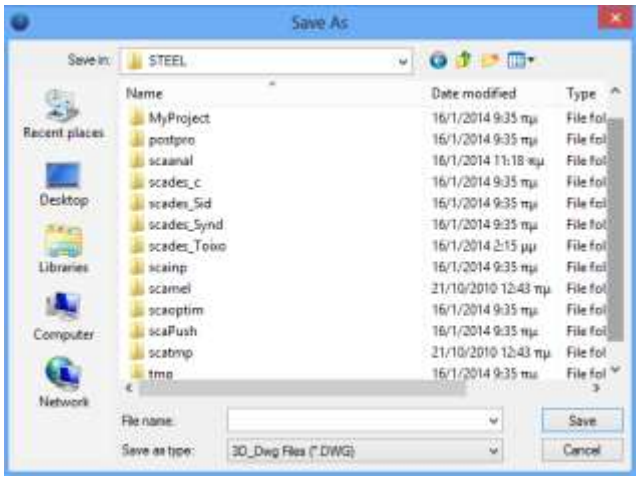


By following the above procedure you can produce over 120 different types of connections covered by the program.
To create corresponding faces, plan views and sections of the overall vector you will to follow a different way.



So, you should click on the command "Export" which opens a new window through which you can export the SCADA Pro file to an autocad *.dwg file. In the "Save As" field you select your design folder to export a 3D version of your structure. To do this you type a name in the File name and then in the "save as type" field you select the 3D_dwg Files (*.DWG) format.

EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'



Then, if you open the generated *.dwg file from autocad you will notice that the whole construction has been exported as a 3D spatial model by SCADA Pro automatically and even the nomenclature of each cross-section is displayed. So, working now in autocad environment you can create any design of your metal structure, visualize your carrier in 3D and with photorealism.

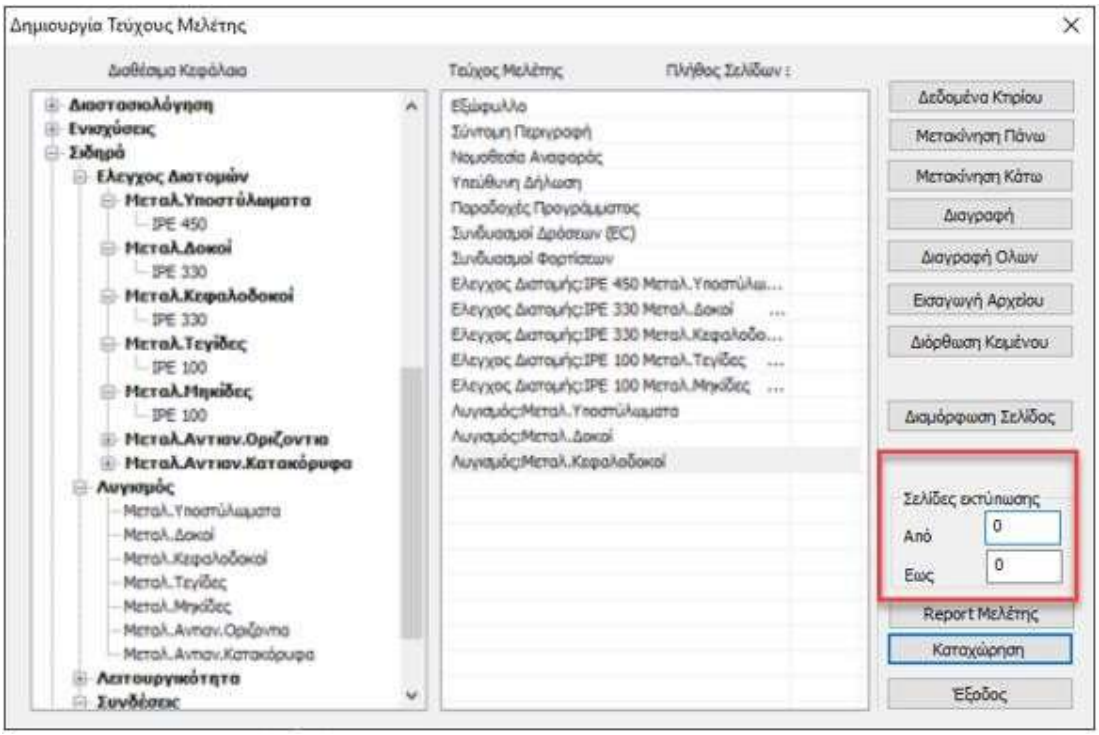
11. COPY

11.1 How to create the study issue:

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

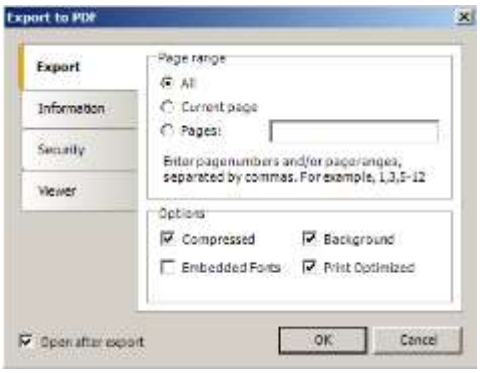
For this example select the chapters you want to include and press the "Study Report" button.
The preview interface of your issue is automatically displayed.
In the new version of SCADA Pro all the printouts of the study results booklet have been redesigned and implemented with modern tools in order 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.

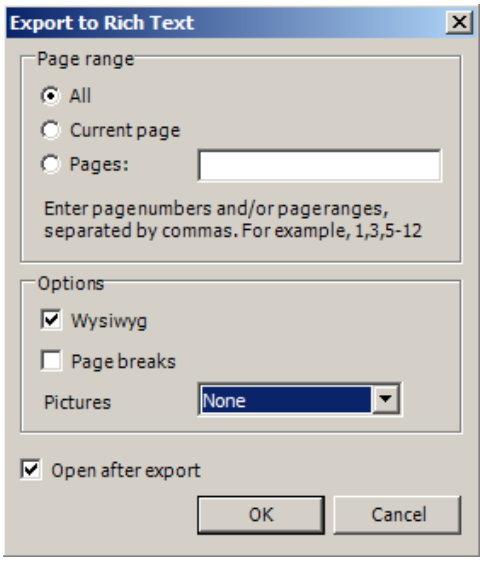
In addition, the ability to "break" the study book into individual sections has been added, a useful and practical feature especially for the easy management of multi-page studies.



EXAMPLE 3: 'DESIGN OF A METAL STRUCTURE'

Through this interface you can save your issue as a file
.pdf, or .doc, .excel, .xml and edit it further in the respective application.





Through this simple example, you were able to experience just a few of features of the new SCADA Pro. Working with the program you will discover that it has unlimited possibilities for simulation, design and analysis of even the most complex metal structure.