SCADA Pro™
Structural Analysis & Design

Example 4

Metal Study

Construction with 3D dwg

ACE HELLAS
INTEGRATED SOLUTIONS

CONTENTS

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 which we can create 3D constructions, process them in the field and build the final structure in simple steps and complete even the most complex studies.
SCADA is a program that is constantly upgraded, evolving and adapting. The technical department of ACE-Hellas in permanent cooperation with Metsovio Polytechnic University is engaged in its continuous development and its adaptation based on new data, applications and needs. A "living organism" that matures!

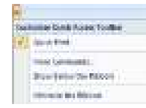
INTRODUCTION

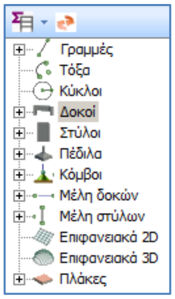
This manual was created to guide the designer in his first steps in the new SCADA Pro environment. It is divided into chapters and based on a simple example guide.
Each chapter contains information useful for understanding both the program commands and the procedure to be followed in order to insert,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

THE NEW ENVIRONMENT

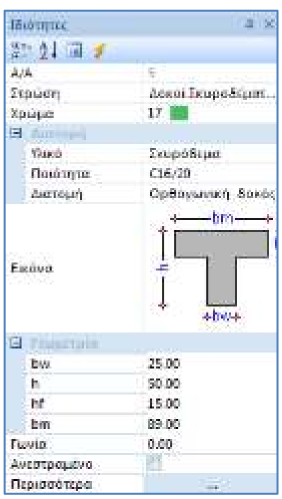
In the new interface SCADA Pro uses the technology RIBBONS for even easier access to the commands and tools of the program. The main idea of the Ribbons design is to centralize and group similar commands in the program, so that you can avoid navigating through multiple levels of menus, toolbars and tables,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

EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'


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".




The new SCADA Pro environment displays on the left side of the screen all the entities of the construction categorized in a tree format either per level or for the whole building as a whole. This categorization allows easy identification of any element and by selecting it it is displayed in a different color in the entity. At the same time, the level to which it belongs is isolated , while its properties are displayed on the right side of the screen, with the possibility of modifying them directly. This function can be performed bidirectionally, i.e. the selection can be made graphically on the vector and the element will automatically appear in the tree with its properties on the right of the screen. Also
it is 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.

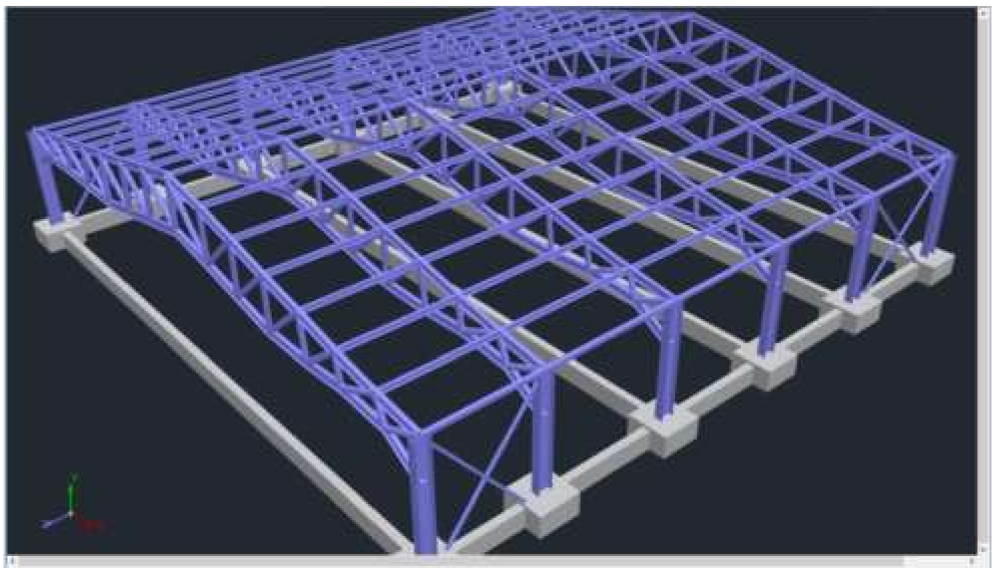


The "Properties" list on the right is automatically populated by selecting an element of the entity. It informs the user of its attributes, as well as allowing changes to them.

1. GENERAL DESCRIPTION

1.1 Geometry

The metal structure under study is a mesh, designed in 3D cad.
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 1 (EC1, EN1991), for wind and snow loads.
Eurocode 2 (EC2, EN1992), for the dimensioning of the foundation.

1.4 Cross sections

Pillars:	HEB500
Headhunters:	SHS150X8-SHS100X8
Crunch:	IPE300
Tread:	IPE300
Network Members:	CHS193,7X10
Tigers:	IPE200
Windbreakers:	SHS100X5

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

1.6 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.

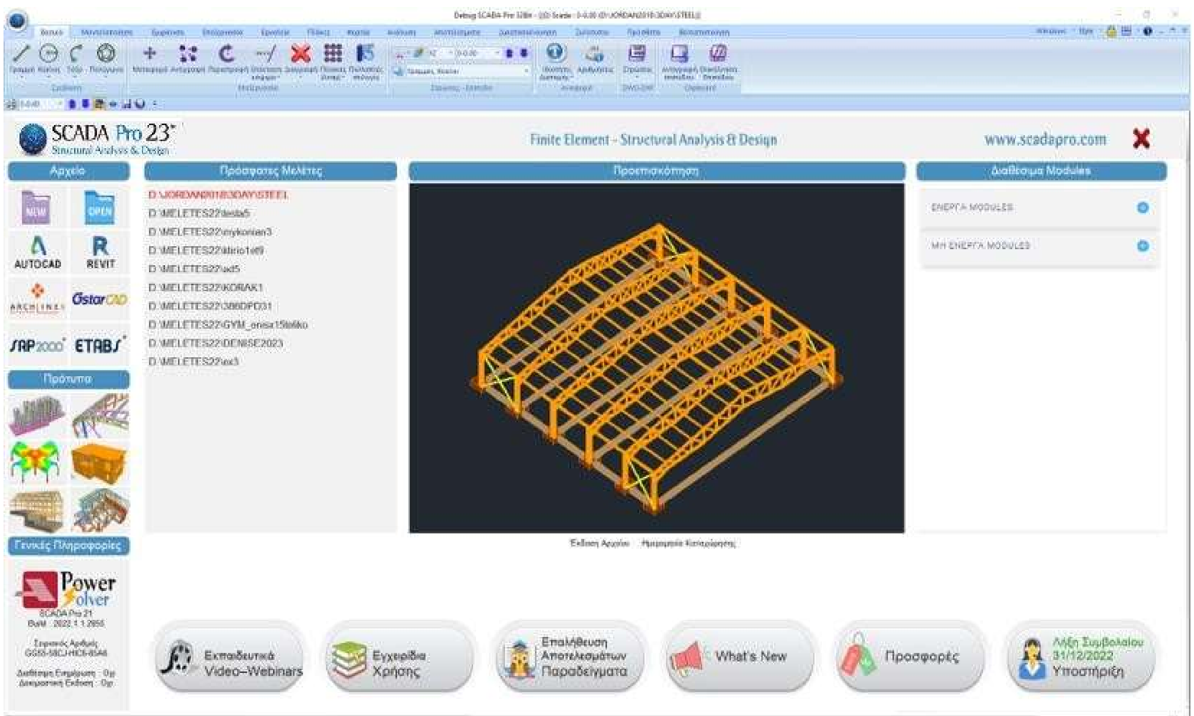
2. DATA IMPORT - MODELLING

2.1 How to start a new study

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

> In this example we will analyse in detail how to use **a utility 3D dwg file** for modeling a metal carrier.

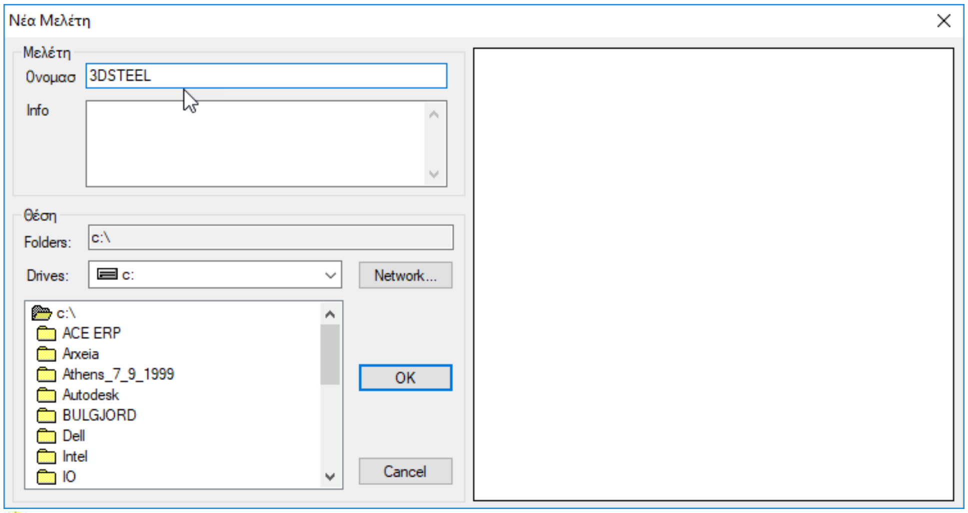
Upon opening the program, the startup window appears on the screen, which includes a set of commands to start the program:



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

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

EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'

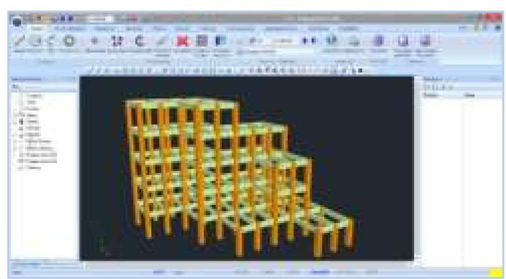


The file name must consist of ***<u>a maximum of 8 Latin characters and/or numbers, without spaces and without the use of special characters (/, -, _)</u>*** *(e.g. ARXEIO1). The program automatically creates a folder where it enters all your study data. The "Location" of the folder, i.e. the place where this folder will be created, should be on the <u>hard disk.</u> 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. The designer starts by defining the stations and inserting the cross-sections, using the modeling commands and with the help of the canvas 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 ready for analysis.

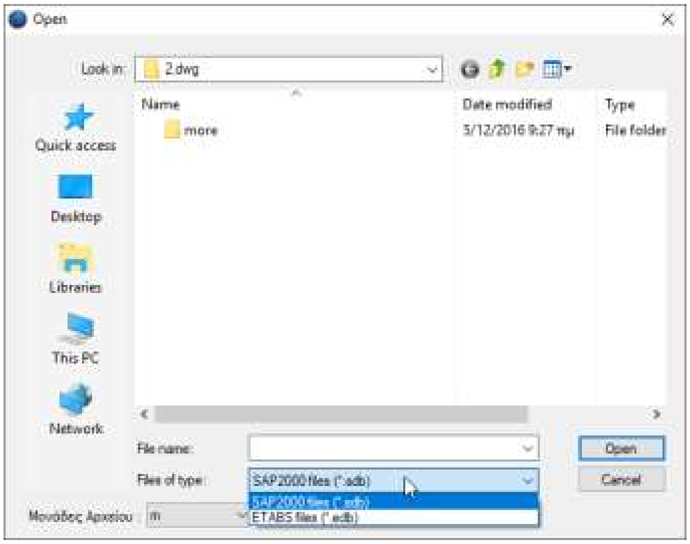


"**ArchlineXP**": read xml files from ArchlineXP.

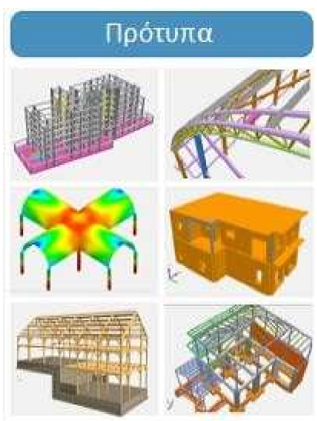
"**ETABS, SAP2000**": reading .edb & .edb files .sdb from the static programs ETABS & SAP2000 .

EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'

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

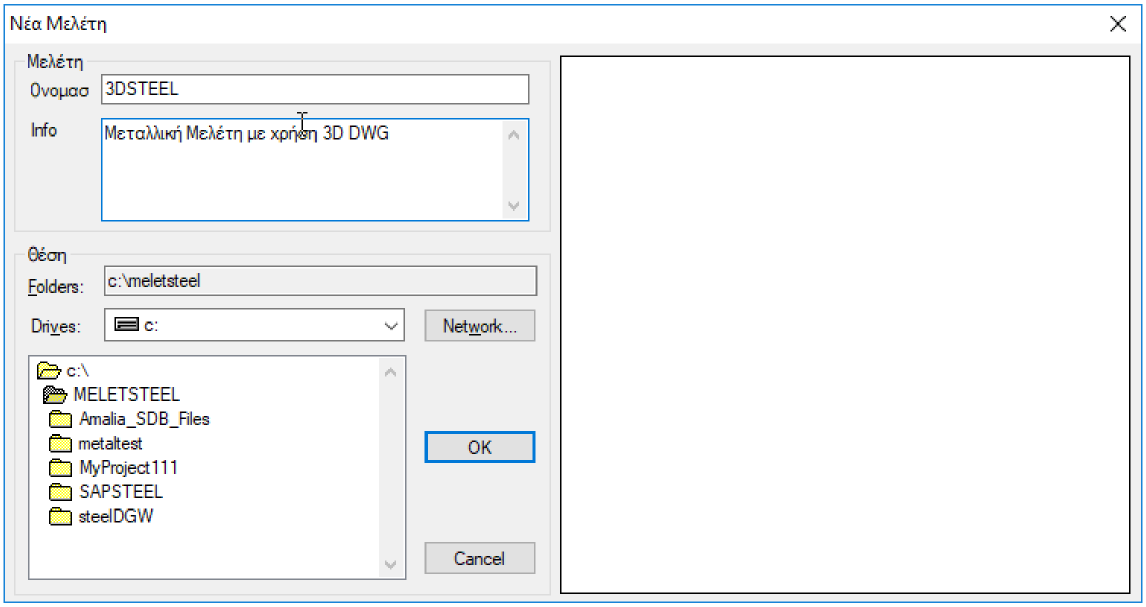
OBSERVATION:

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 standard construction you manage to model the carrier with a single movement! But also in in the case of more complex carriers, the use of standard structures can provide the foundation on which to build a complex carrier.

EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'

2.2 New Study

Select the relevant icon  and in the dialog box



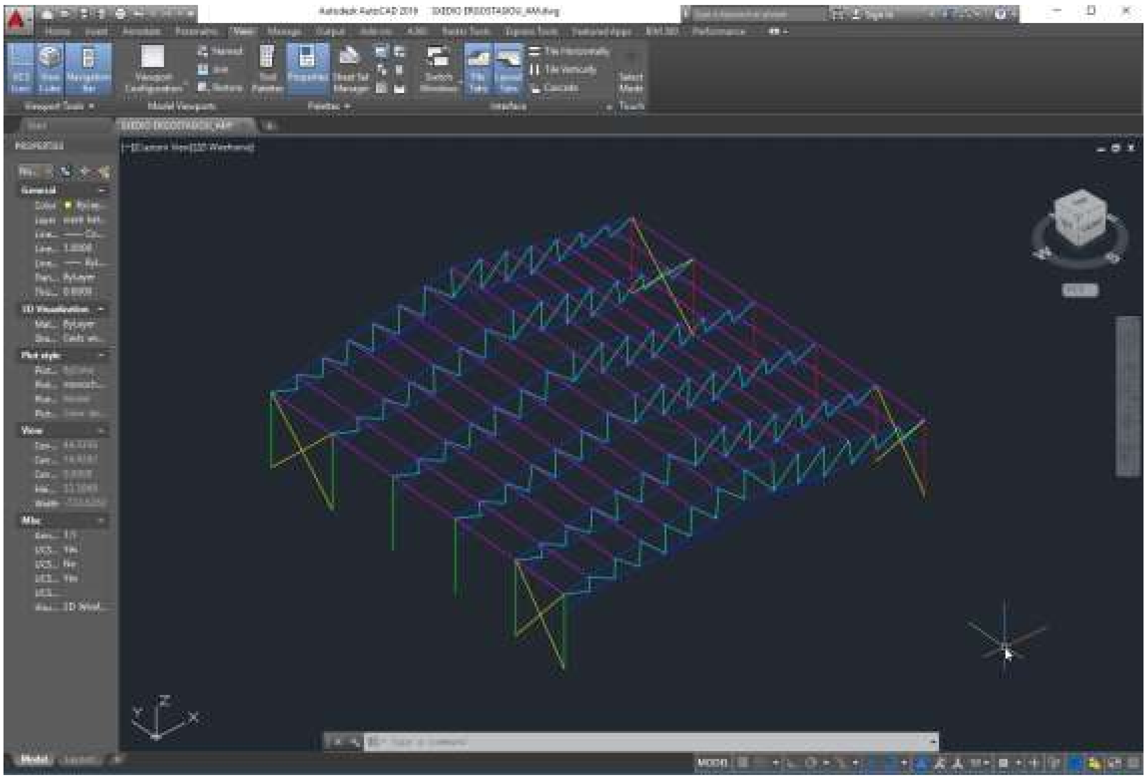
give a "**Name**" to the study. If desired, write some information about the study in the "**Info**" field and specify the location of the registration within the local disk.

2.3 Create the Model of the construction from 3D dwg file

This example is intended to train the user to model a metal structure from a **3D dwg** file. With the new version of Scada Pro it is to automatically identify the metal sections from 3D drawing.

In detail, for the automatic recognition of the metal sections using a 3D dwg file, the following steps are followed:

2.3.1 Preparation of 3D dwg file



For this example, the design of the network in the above picture is used. This is a single opening grid with 6 meshes. In the 1st and 5th frames there are chiasm windbraces on both sides, and purlins on the roof.

EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'

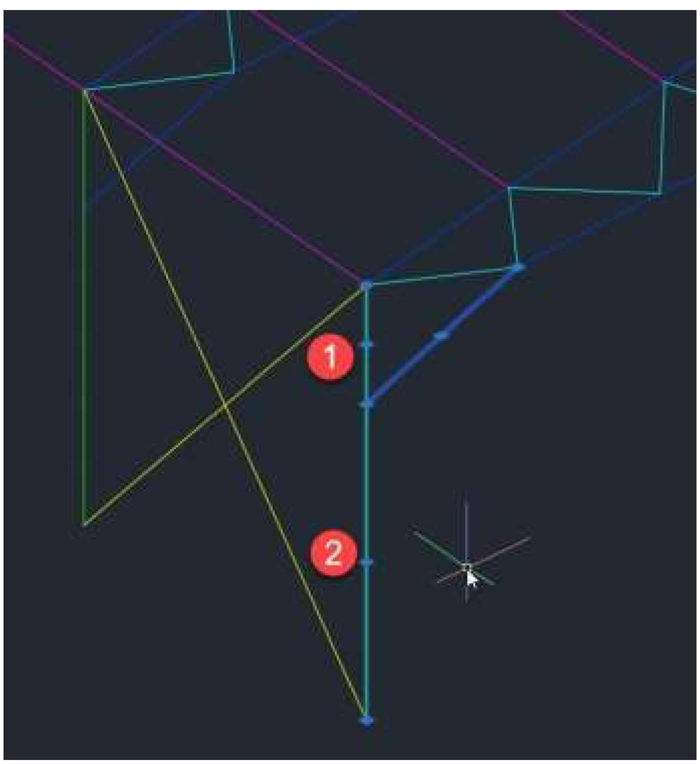
⚠ BASIC DESIGN REQUIREMENTS:

1. During the design different layers were defined for each group of cross-sections:

S...	Name	O...	Fre...	L...	Color	Linetype	Lineweig...	Trans...
	0				wh...	Continu...	—— Defa...	0
	col1				red	Continu...	—— Defa...	0
	col2				gr...	Continu...	—— Defa...	0
	kefalodokos				200	Continu...	—— Defa...	0
	pelma				150	Continu...	—— Defa...	0
	tegides				ma...	Continu...	—— Defa...	0
	TEXTM				yel...	Continu...	—— Defa...	0
	TEXTN				yel...	Continu...	—— Defa...	0
	truss				cyan	Continu...	—— Defa...	0
	xiasti katak...				yel...	Continu...	—— Defa...	0
✓	zygoma				160	Continu...	—— Defa...	0

2. In addition, at the intersection points of the lines, where a node is required at the tread during the identification of the cross-sections, the drawing was made with line segments.

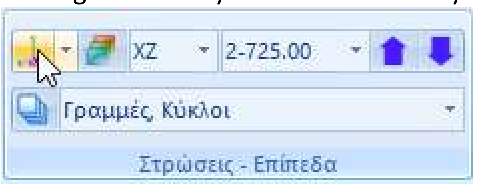
More specifically, the picture below shows that at the point where the curve meets the pole and therefore a node will be created, the line of the pole is not continuous, but consists of two consecutive segments.



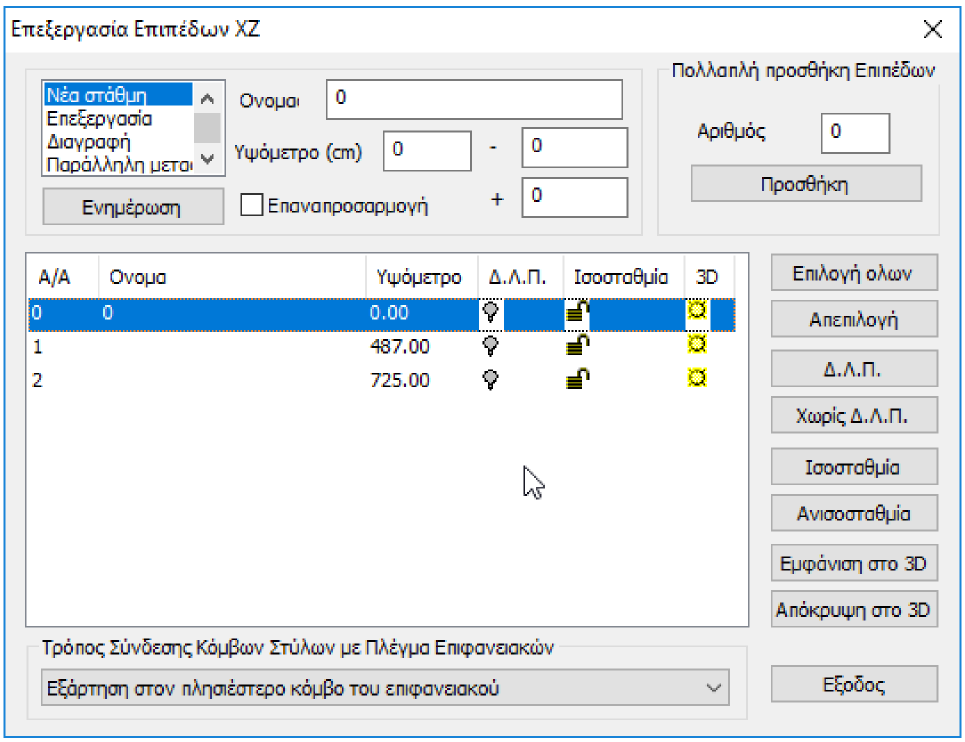
2.3.2 Insertion of an auxiliary drawing and identification of cross-sections

After you have set the name of the study, and before importing the auxiliary file, set the levels of the vector.

Through the Utility section and the Layers-Levels command group,



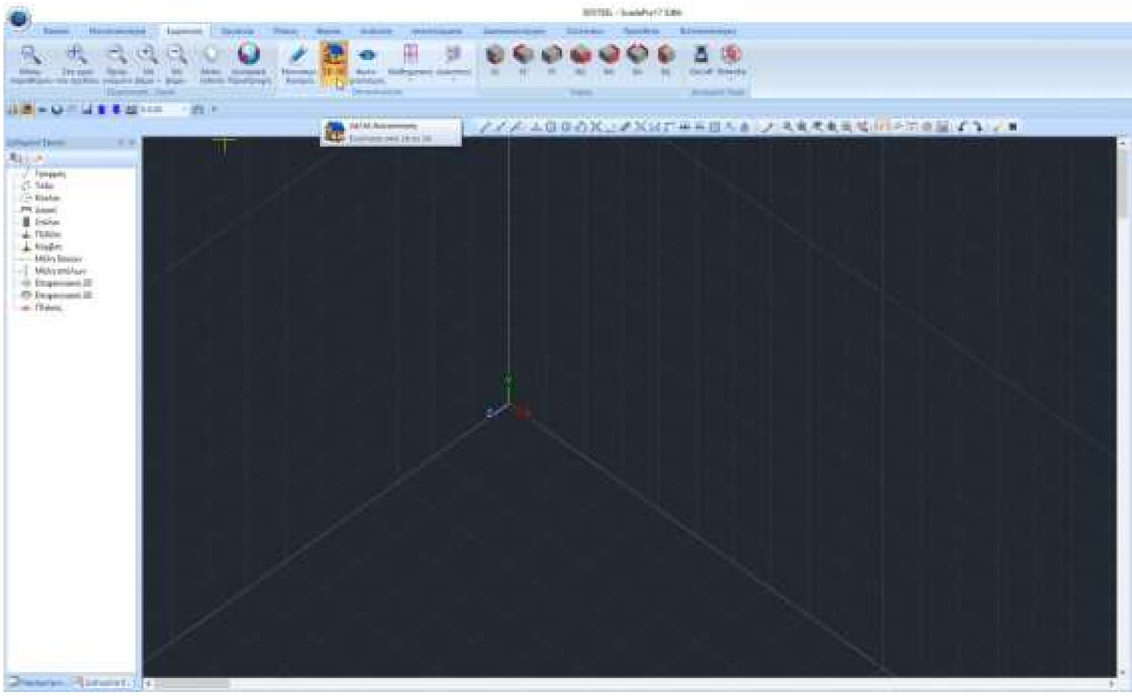
you select Edit HZ Levels and set the levels, while disabling the aperture mode:



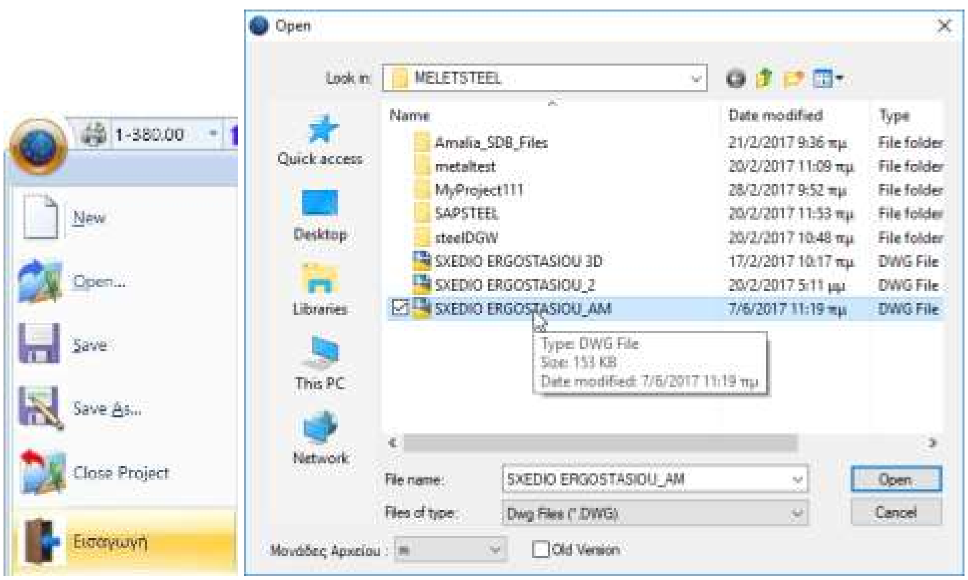
EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'

BASIC CONDITION FOR THE INTRODUCTION OF THE 3D PROJECT:

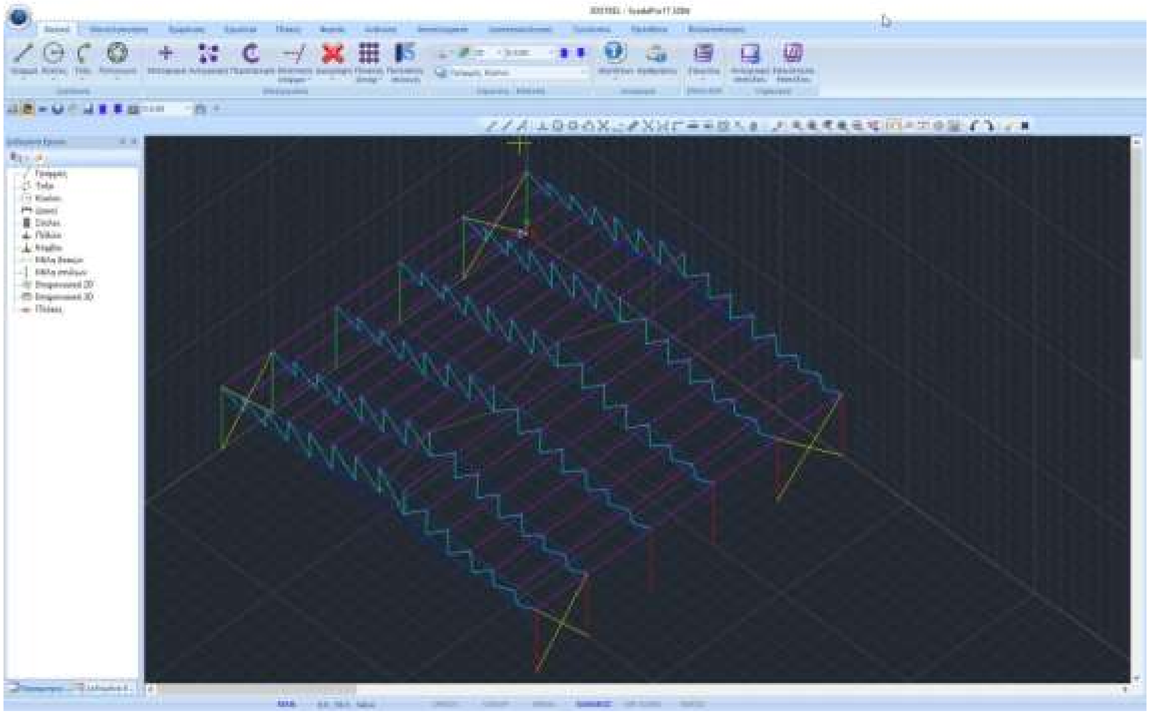
You select the 3D view of the blank Scada Pro desktop.



The import of the utility is done via the "Import" command



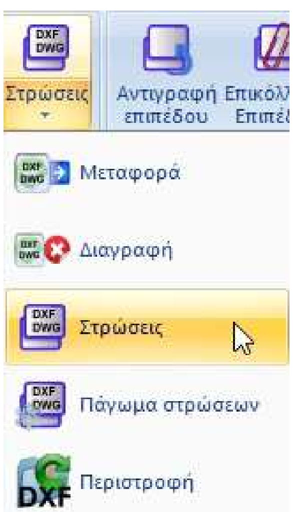
EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'

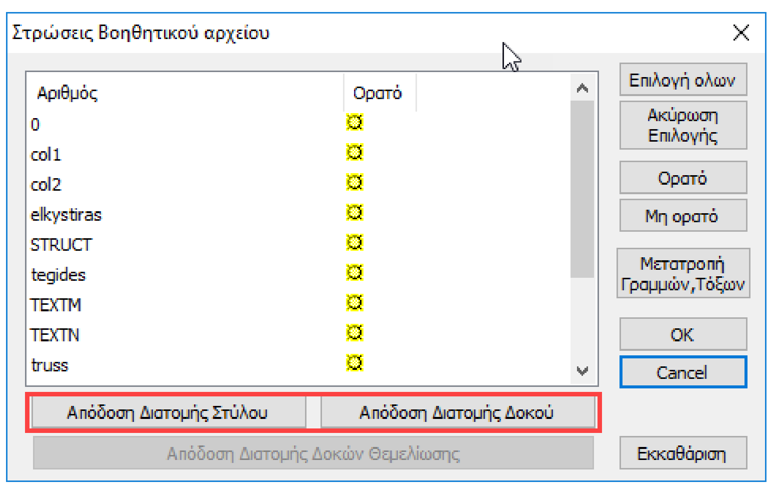


The drawing is displayed on the 3D desktop.

Within the "Basic" field and the DXF-DWG command group you start the automatic process of importing the metal cross-sections:

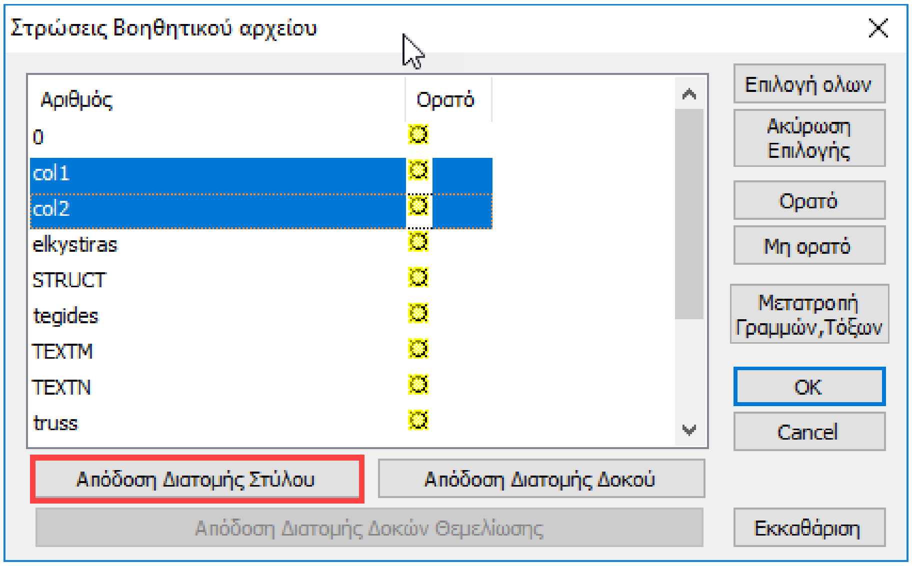
Select the "Layers" command to open the "Layers Utility File Layers" dialog box, which includes the list of layers in the design and two new commands "Column Cross-section Performance" and "Beam Cross-section Performance".



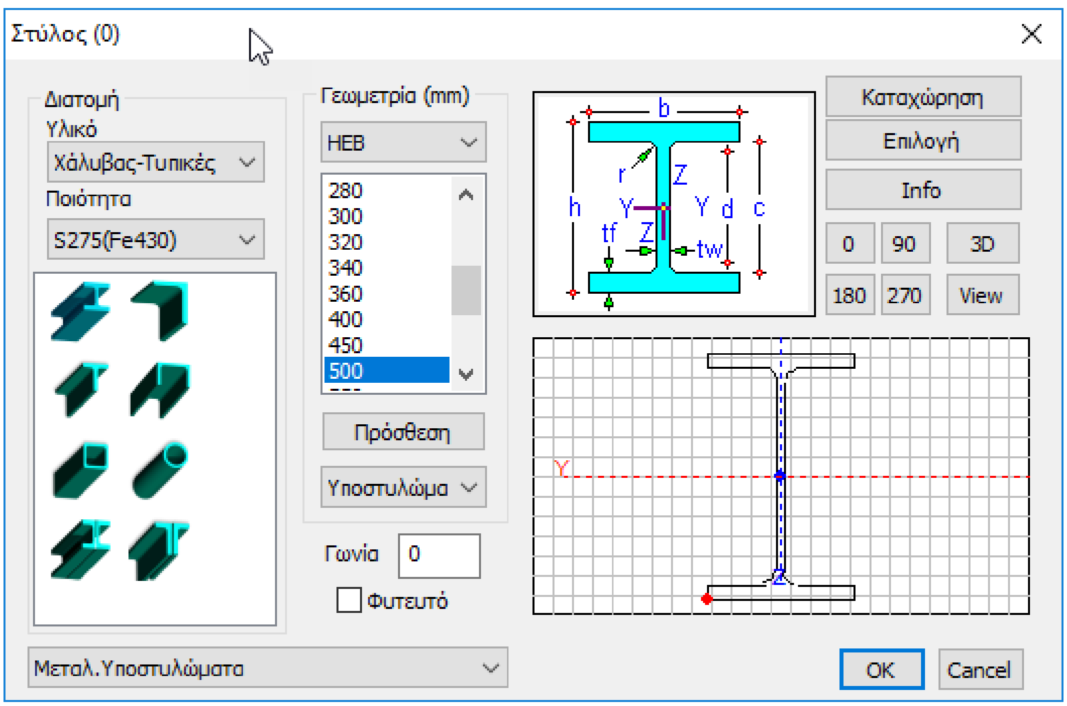


Select the Layers col1 and col2 and press the Column Cross-section Performance button

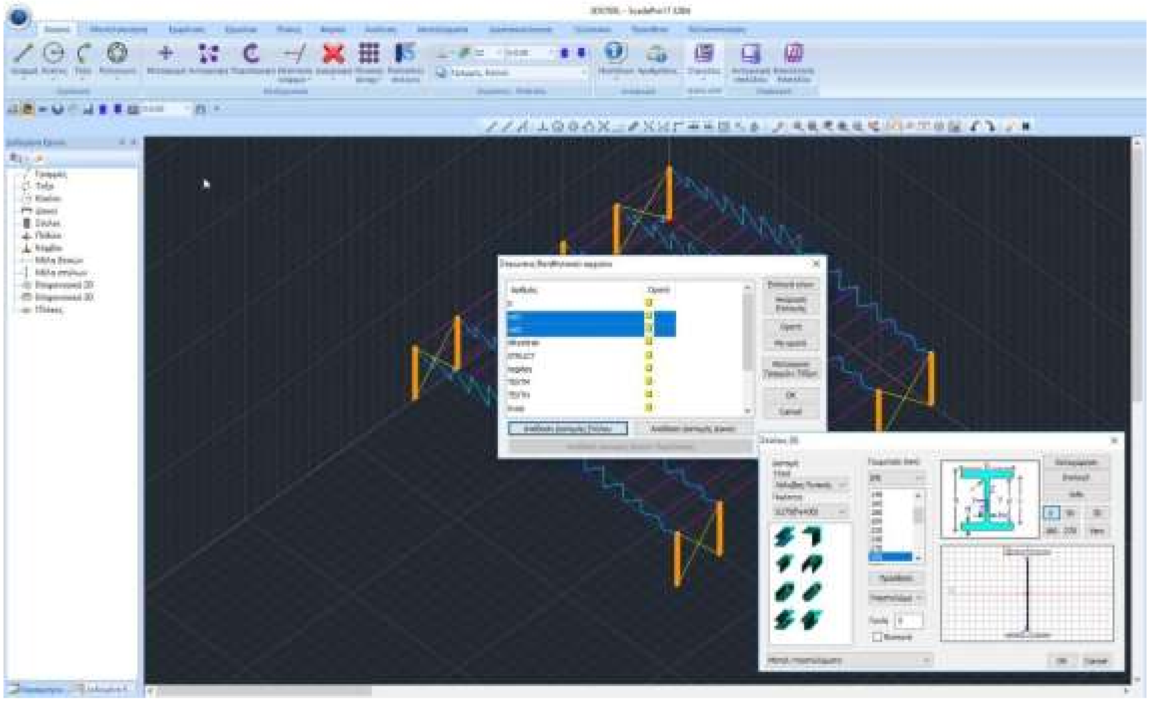
EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'



Set cross-section HEB500 with angle 0 quality S275 quality and layer Metal. Support columns.

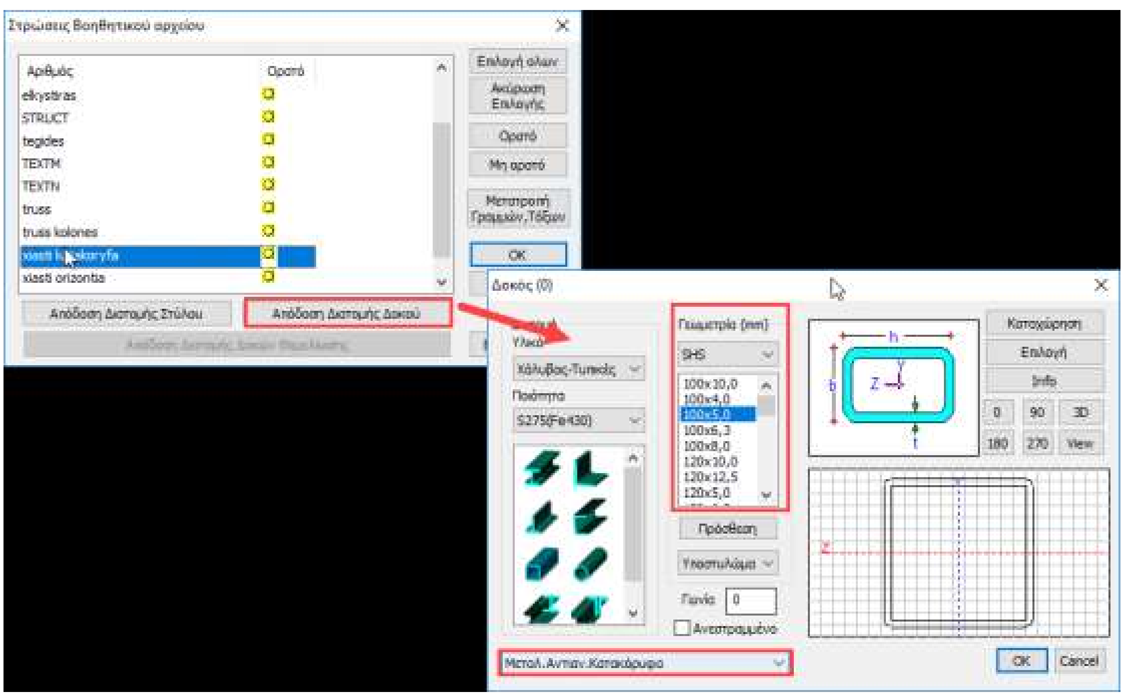


EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'

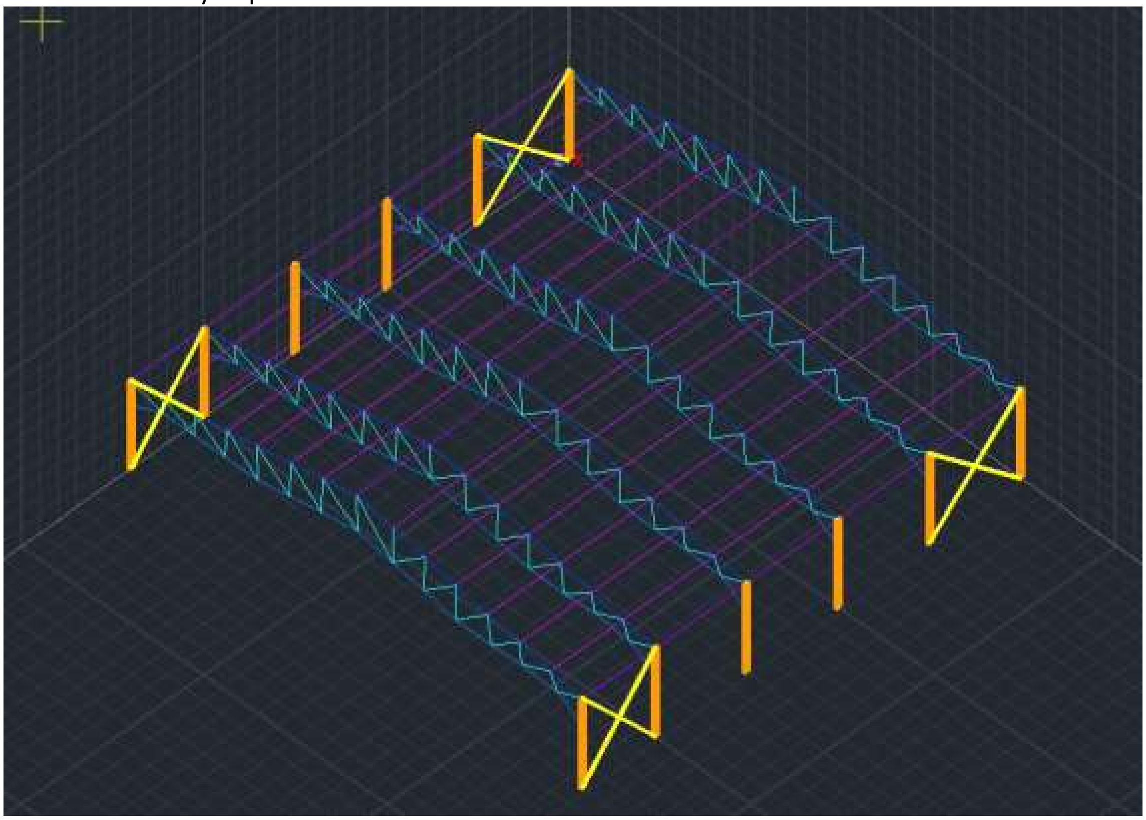


The cross-sections of the columns are automatically inserted in the position defined by the lines of col1
and col2 layers of the auxiliary file.

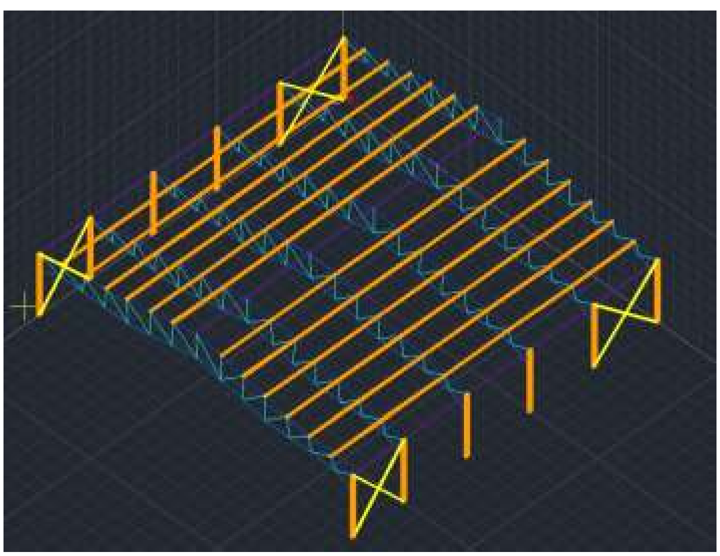
With the same logic select the xiasti katakoryfa layer and the Beam Performance:

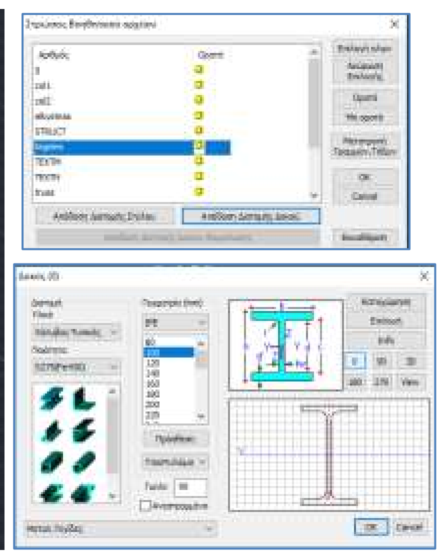


Set SHS100X5 and assign it to the layer Metal.Vertical. Ok, and the cross-sections of the abutments are automatically imported.



Likewise for the Tigers:

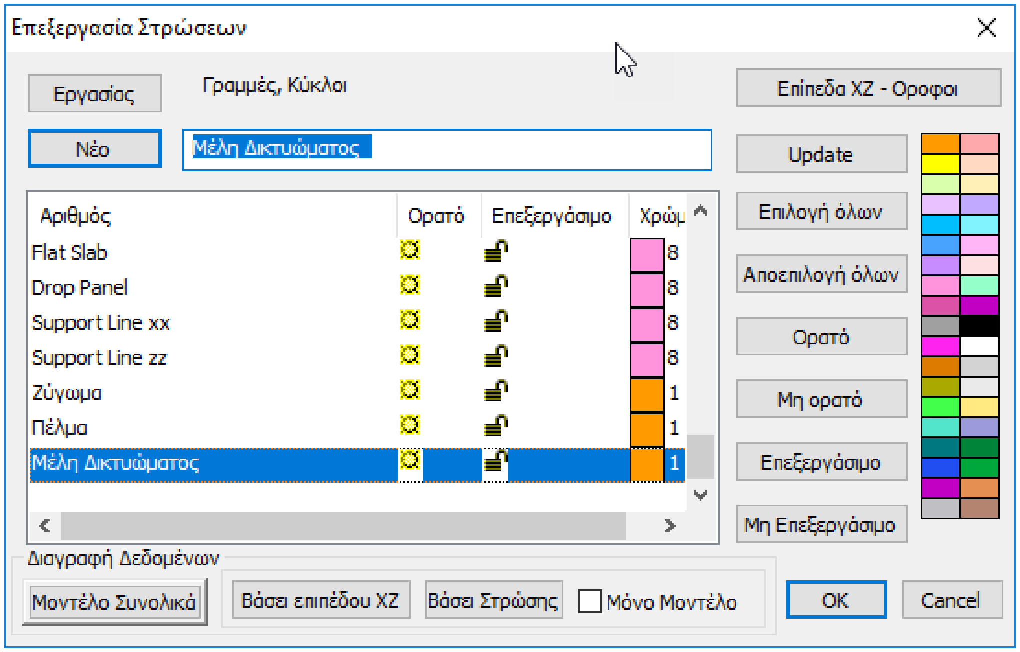




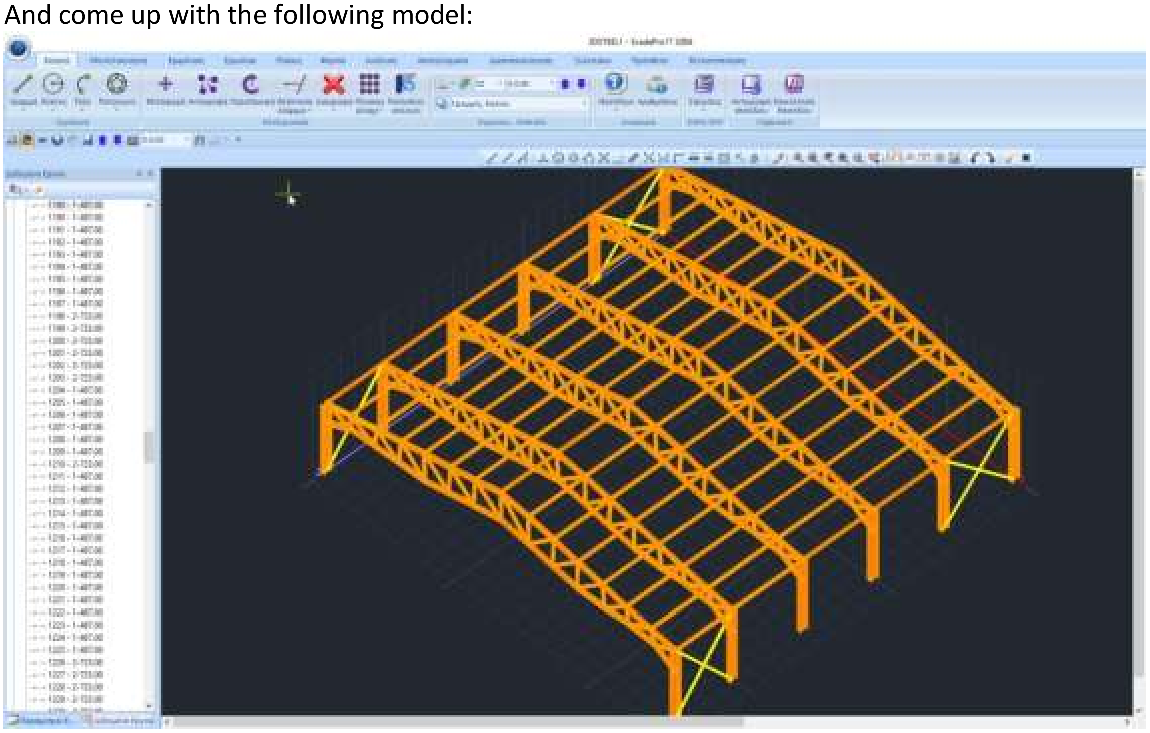
For the mesh elements, define three New Layers within the "Edit Layers" window:
-Egymia,
-Pela,,

EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'

-Network members,
in order to import all the corresponding cross-sections of the (zygwma), (pelma), (elkistiras) - (truss) layers of the design.

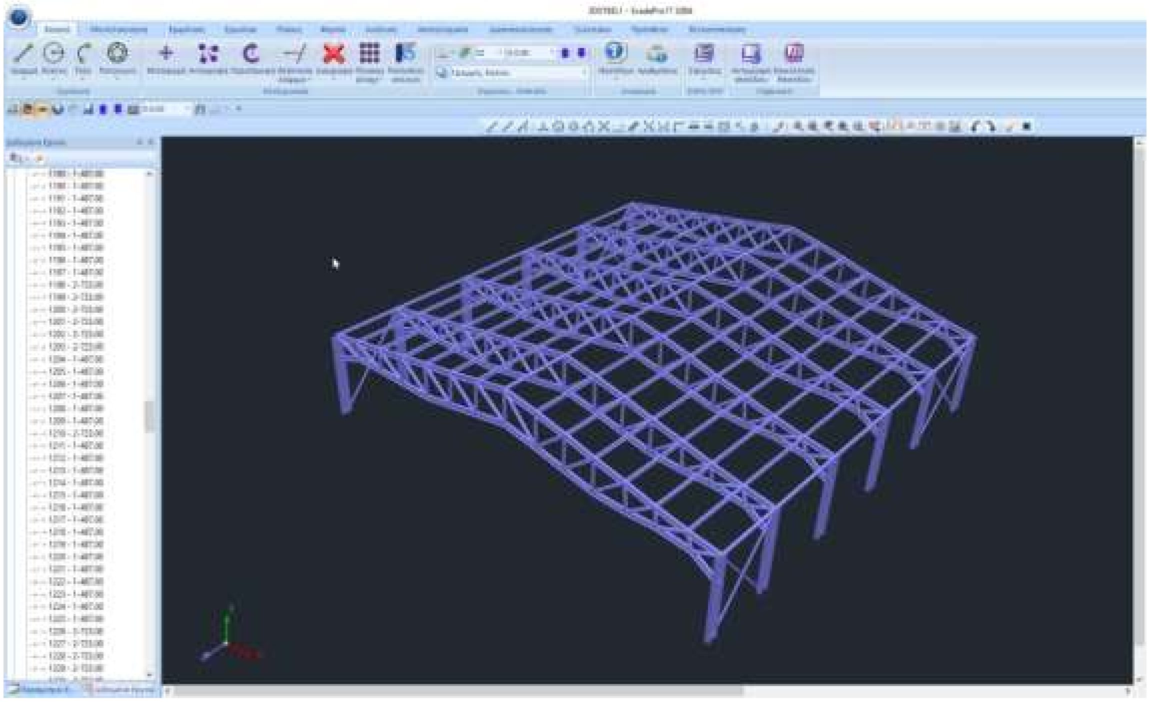


And come up with the following model:



and activating photorealism:

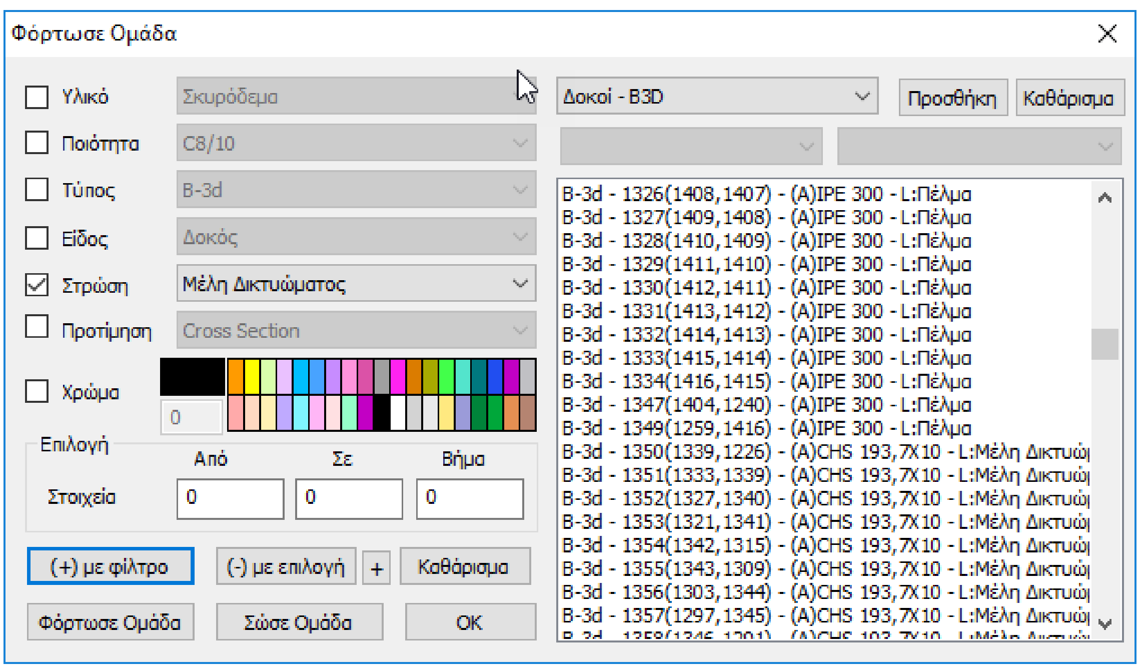
EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'



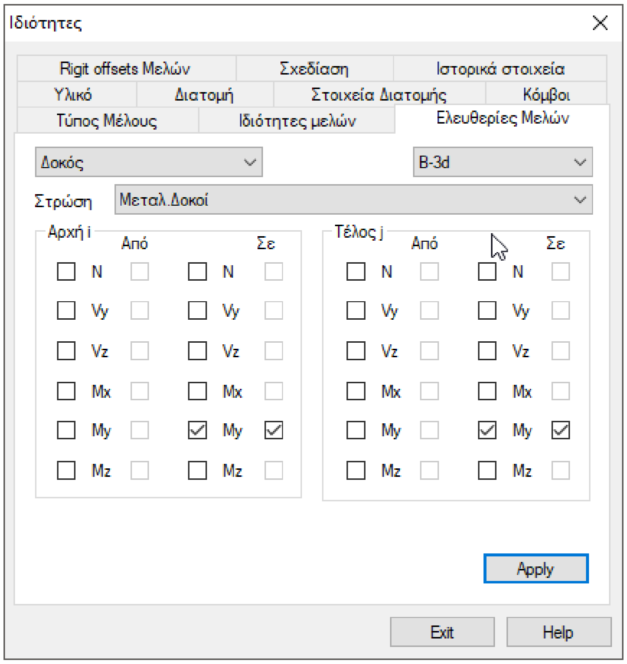
⚠ **EXCEPTION:**

The truss does not transfer torque, so we have to free its members from the main torque.

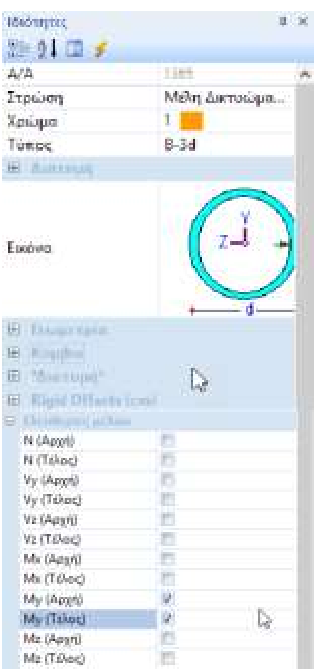
Using the Multiple Options command and the Filtered option, we select the Tread and (+) layer with filter and the Network Members and (+) layer with filter.



Ok and right click to open the Multiple Options window, and in the Member Freedoms check the start and end MyS for the Tread Layer and the Network Members Layer, and finish with Apply and Exit.

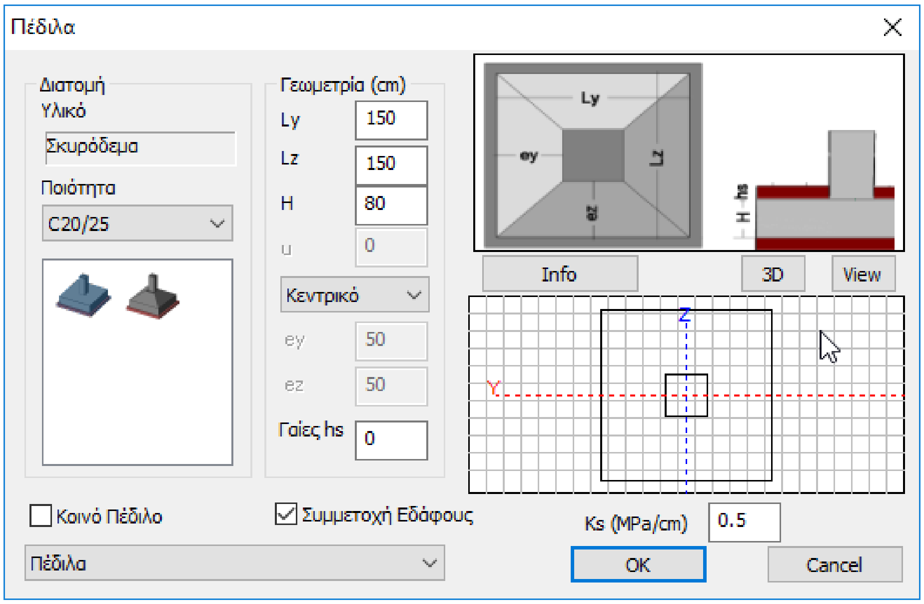


By left-clicking on an element of the network, in Properties you can see that the corresponding Member Freedoms have been updated:

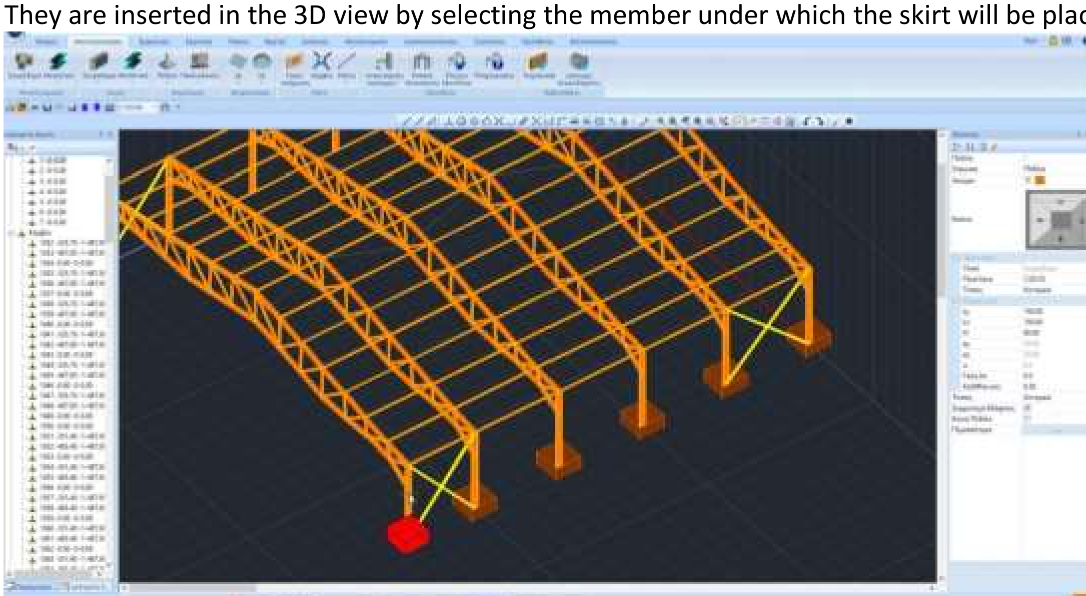


2.4 Installation of pedestals and connecting beams

From the Modeler, you select the command PEDILA and set the parameters.

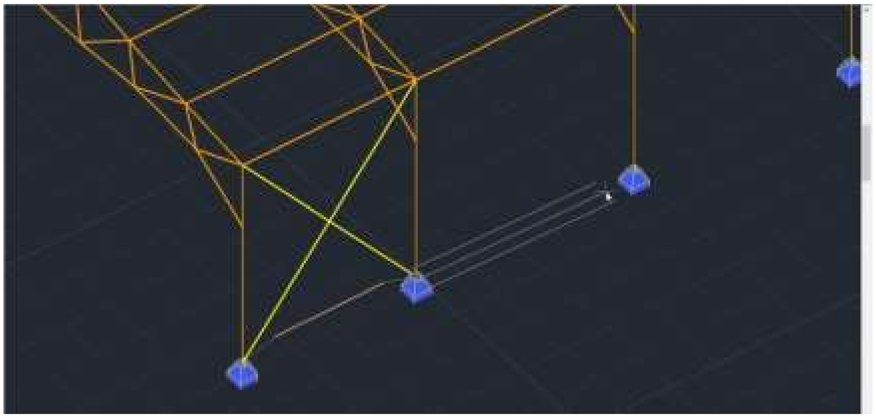


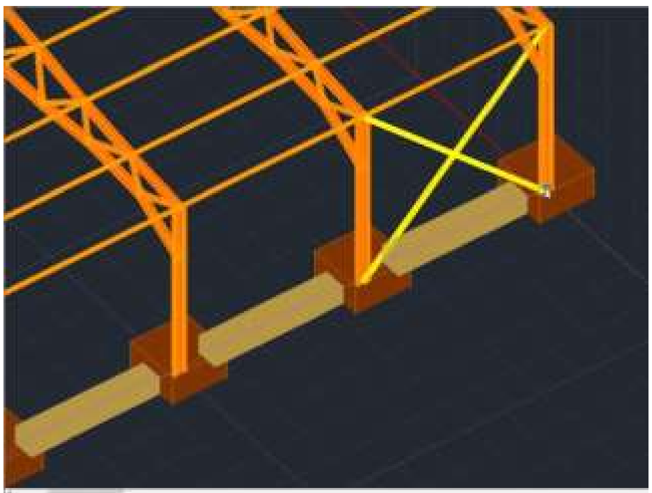
They are inserted in the 3D view by selecting the member under which the skirt will be placed.

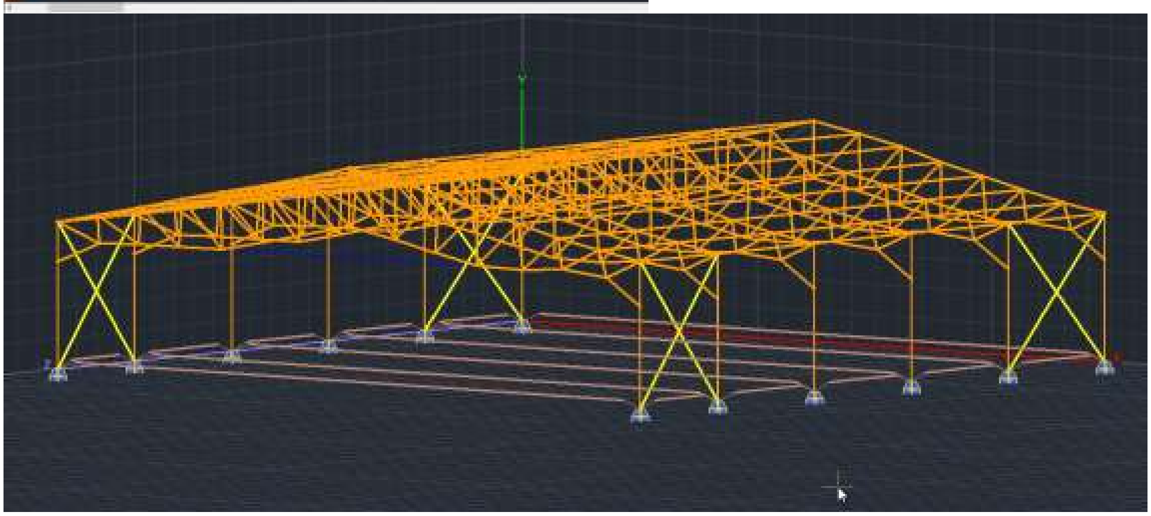


Similarly, for the connecting beams, you select the cross-section and pass them from node to node in either Mathematical Illustration or Physics. Automatically you also calculate the Mathematical Model of the beam:

EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'

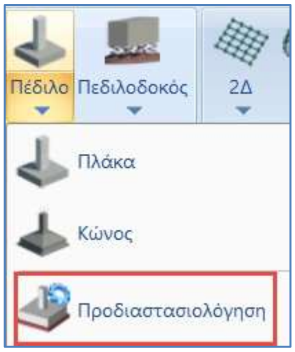


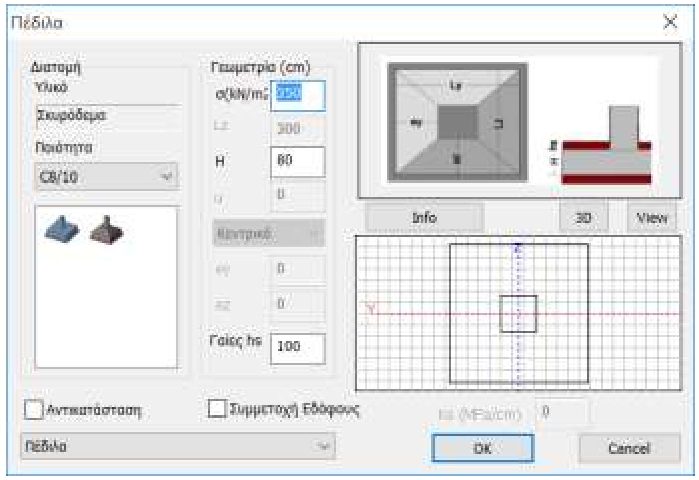




NOTE:

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.

3. IMPORTATION OF GOODS

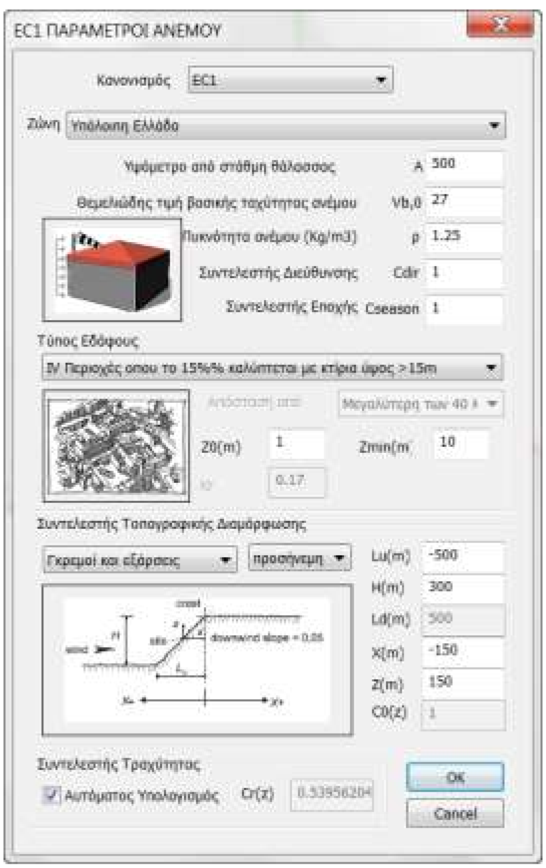
3.1 How to enter wind and snow loads in the automatic way based on Eurocode 1

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.

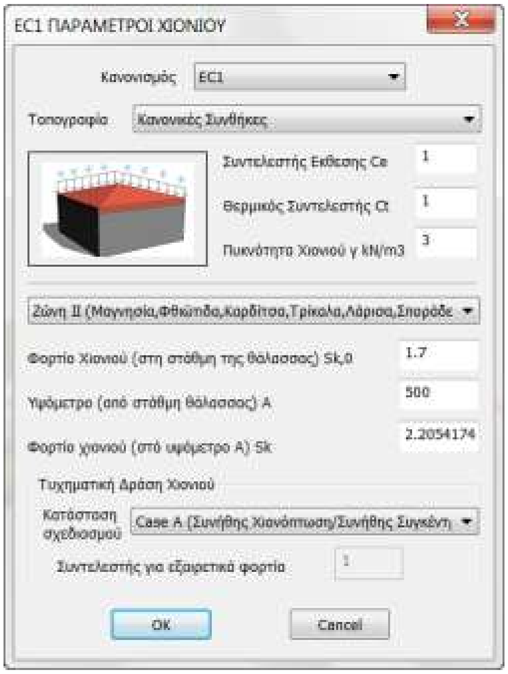
3.1.1 Parameters

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.

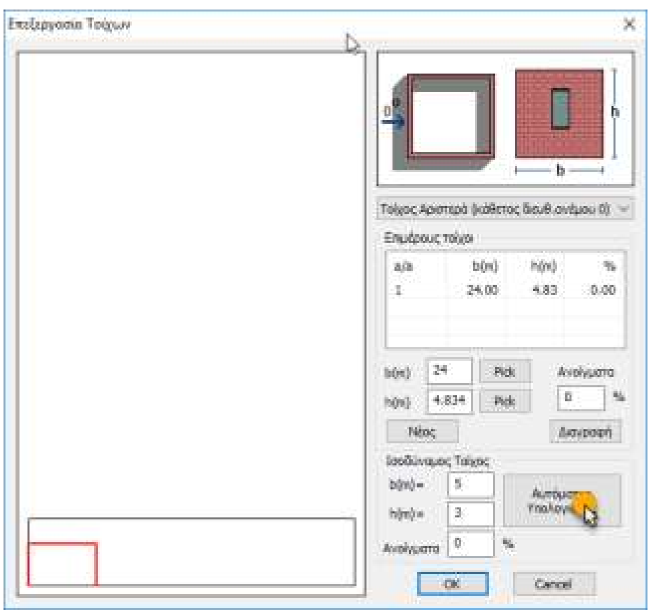
Similarly, in the snow parameters window:



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

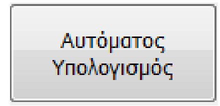
Then, via "Edit> "**Walls**", we define the walls per direction for calculation of the Equivalent Wall.



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

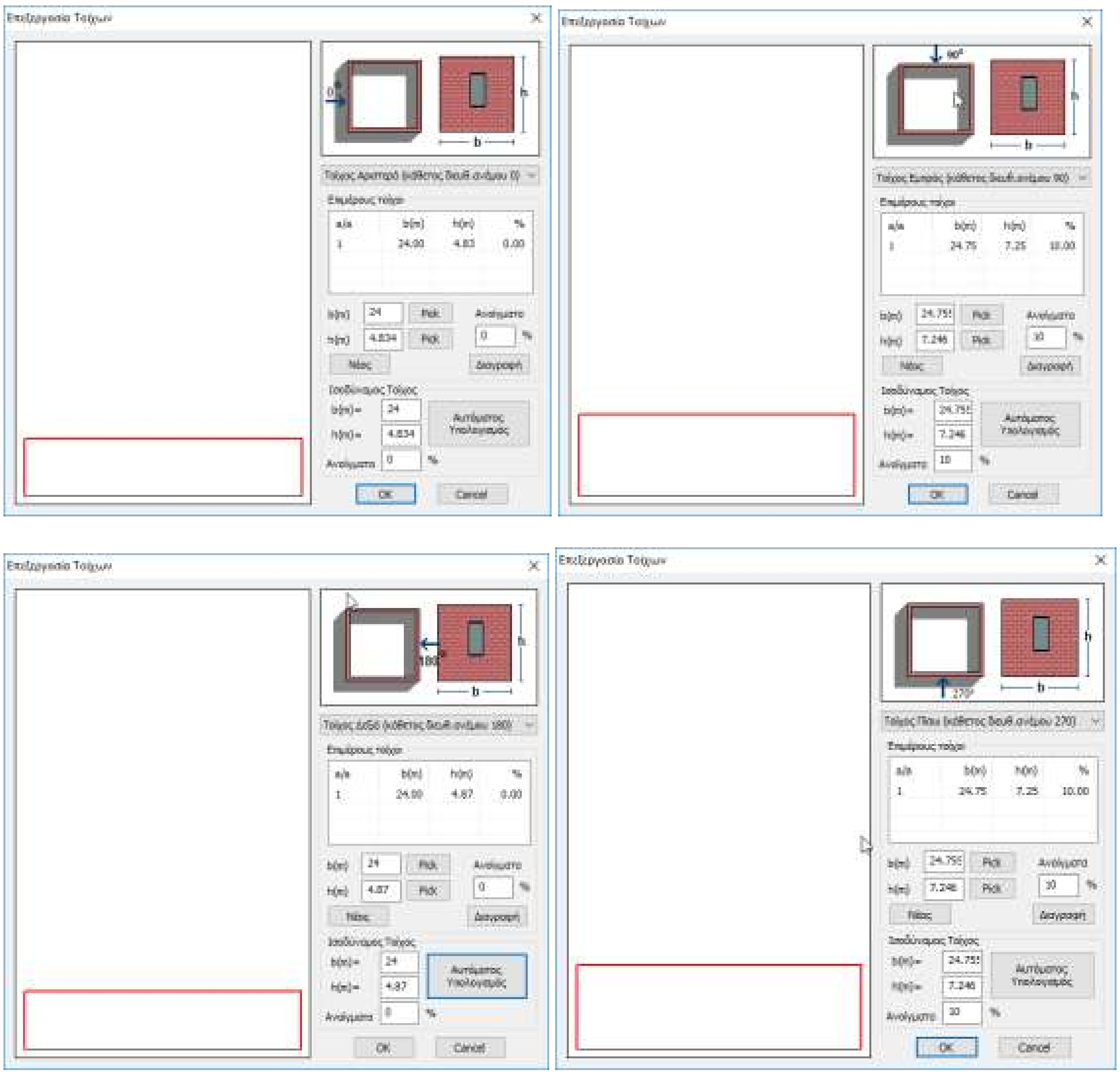
give the program the length (b) and height (h) for each wall (Left, Front, Right, Back), simply by clicking on Pick and selecting each time with the mouse the 2 ends of the wall in the corresponding direction, viewing the vector in 3D.

⚠ The height (h) is set from the bottom up, always starting from the foundation level.

Then, set the percentage of openings and press the button 

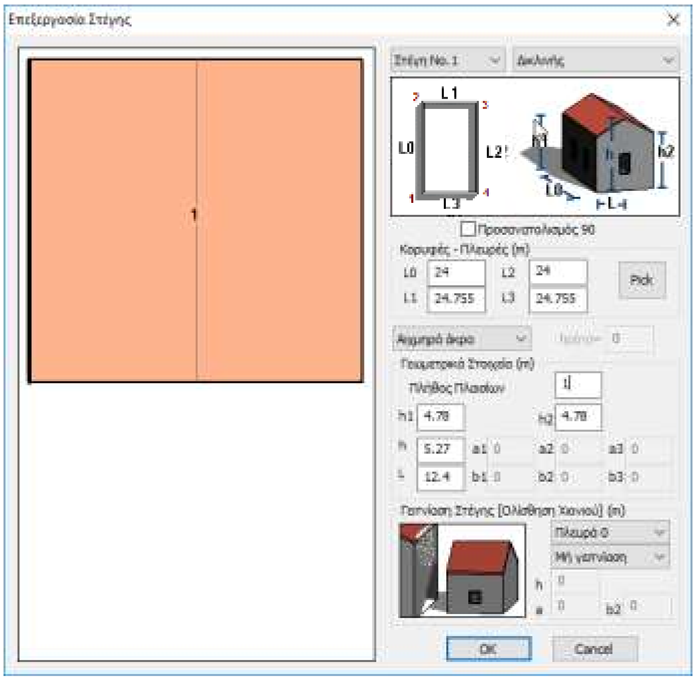
so that the program calculates the dimensions of the equivalent wall.

Repeat <u>for all four directions of the walls</u>.



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, as well as the Geometric Elements.

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 the EC1.

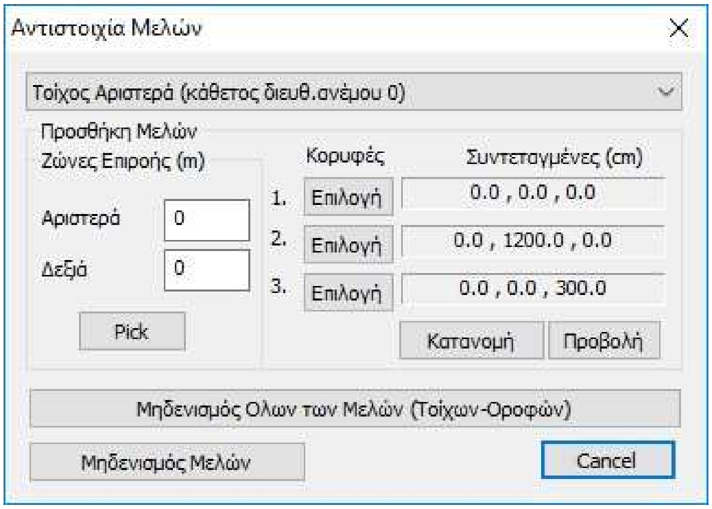
3.1.6 Matching members

The next step is to define the zones. This is to determine distribution of pressure (wind and snow) on the structural elements of each facade and roof.
Select the command and in the dialog box you select one by one the walls and/or roofs for the distribution.

In the new version of SCADA Pro, the automatic calculation of the influence surfaces for the linear members was completed and integrated in order to distribute the wind and snow loads.

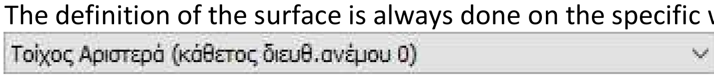
⚠ *Recall that until now automatic allocation was only done for constructions coming from the standard ones. It is now possible to perform this allocation on any surface defined by the designer.*

Selecting the command opens the following dialog box



In the part concerning the old definition of influence surfaces nothing has changed as well as the function of the "Pick" button where it hides the dialog box and displays the existing influence surfaces, has remained the same.

However, a part has been added on the right concerning the <u>definition of the surface with three points.</u>
The definition of the surface is always done on the specific wall that is active in the window



⚠ *It is advisable before starting either **manual** or **semi-automatic**, to reset everything that exists by pressing the "Reset Members" button.*

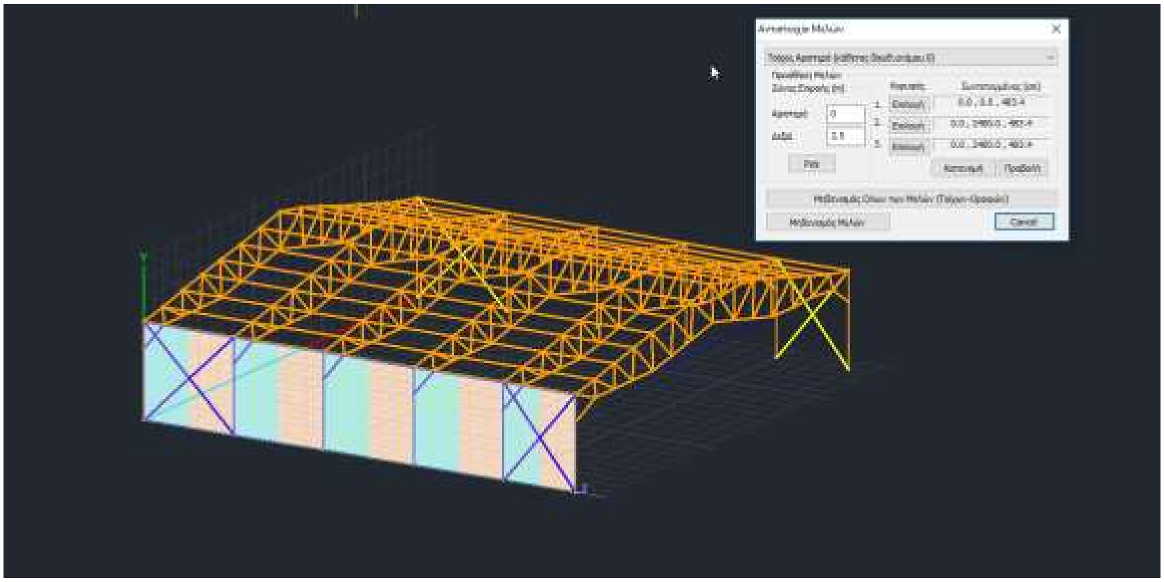
- **Semi-automatic Process**

The points are shown graphically with the following peculiarity:

- The first two points define the <u>direction in which</u> the automatic calculation of the influence surfaces is performed for the elements that are <u>parallel to</u> this direction.
  Note also that the distribution is done for all linear members belonging to this plane and as we said it is parallel to the first line.
- After defining the 3 points, we press the "Distribution" button and the program automatically executes the distribution and displays it.

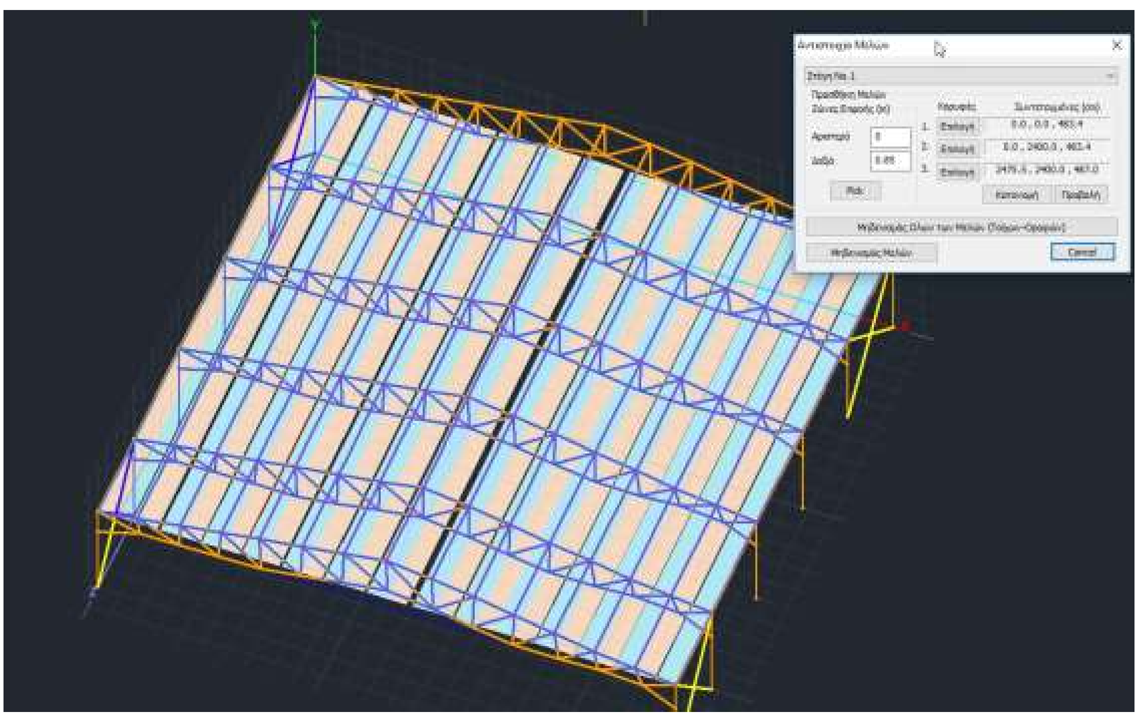
The same definition is made for the other walls.

EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'



As far as roofs are concerned, the definition can be done sequentially, i.e. after I first choose the roof to define

Στέγη No.1 it is obligatory to select the individual levels, i.e. in a two-pitch roof to select successively the two levels of the slopes, because as we said the logic is to define with three points a level for which and for those members belonging to it the calculation of the influence surfaces will be done automatically.
For example, I set the left slope first and then the right slope.

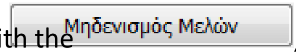


⚠ Finally, it is worth noting that if the walls are <u>correctly defined</u>, there is NO need to define the levels. We simply select each wall and pressing the "Allocation" button makes and simultaneously displays the allocation to the linear members belonging to that wall.

⚠ The same applies to roofs that, attention is on one level. For the rest of them (e.g. two-plane roofs), however, the procedure of defining the individual planes described above is needed.

⚠ **NOTE:** Pressing 

, the zones for all the zones are reset.

members of all facades of the building, while with the 

, only the zones of the members of the selected wall or roof are reset.

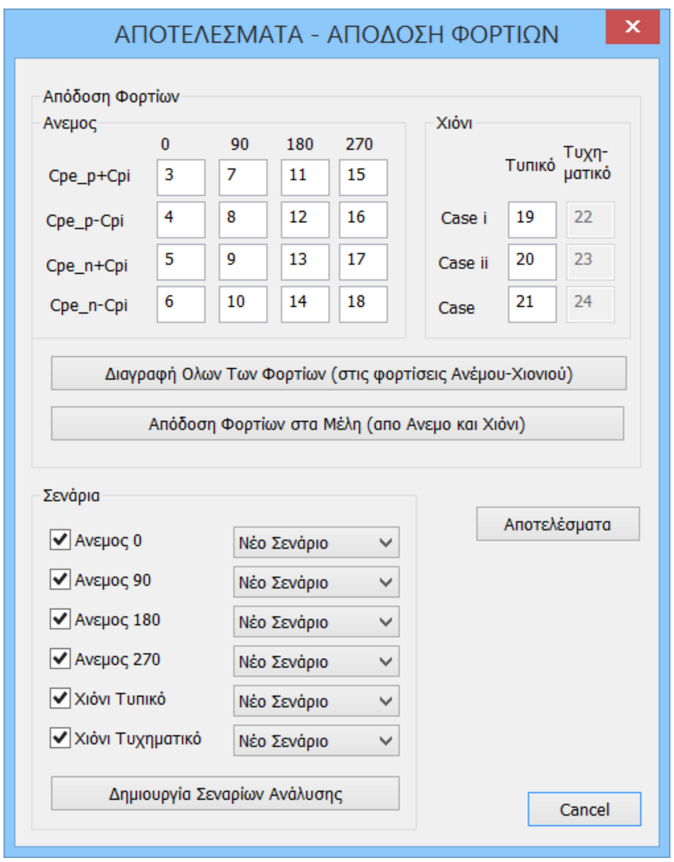
3.1.7 Results

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 loads for typical snowfall (random action applied in Greece).

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



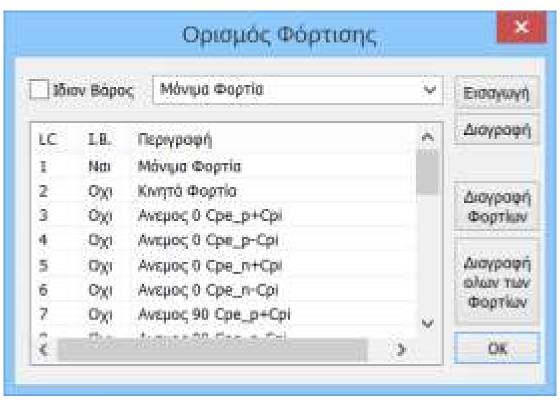
EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'

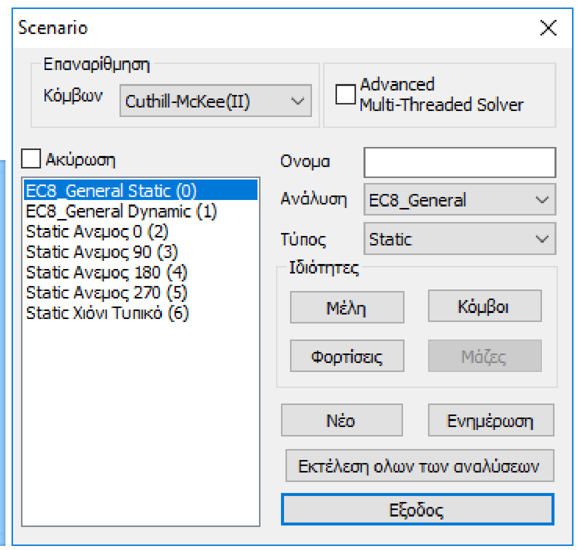
Recall that: Charge 1:
Permanently
Charge 2: Mobiles
and now another 16 loadings are added for wind (from 3 to 18) and 3 for snow (19, 20 and 21)

Select the command Απόδοση Φορτίων στα Μέλη (απο Ανεμο και Χιόνι) to attribute The loads of wind and of snow at members the construction, ή 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

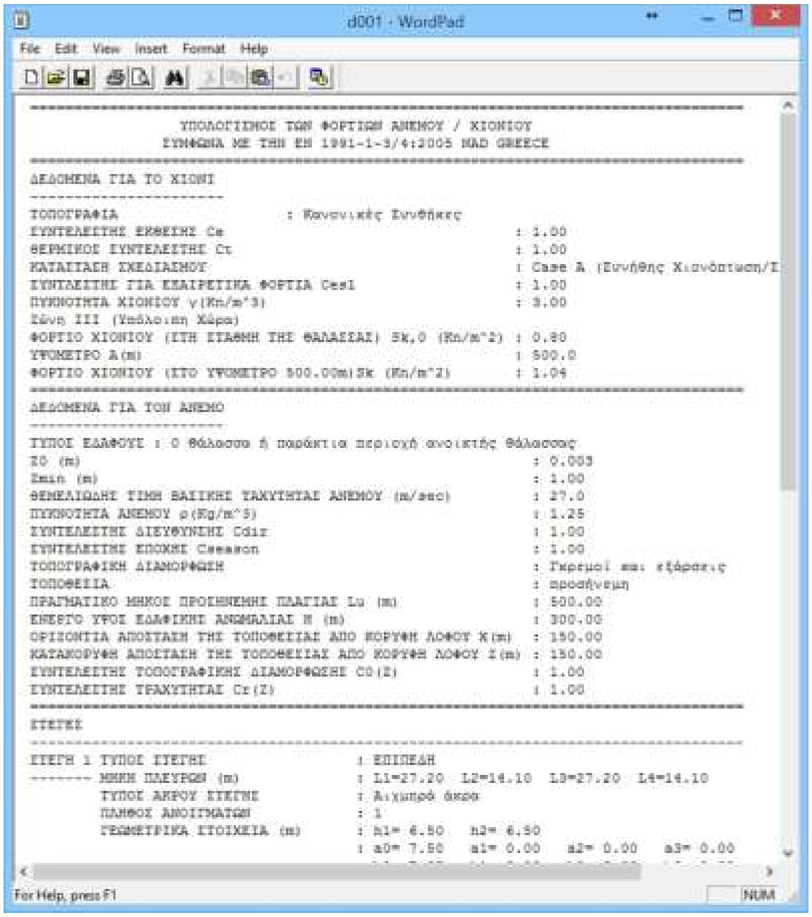
So SCADA Pro, in addition to automatically calculating distribution of wind and snow loads, automatically creates all the analysis scenarios.





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

EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'



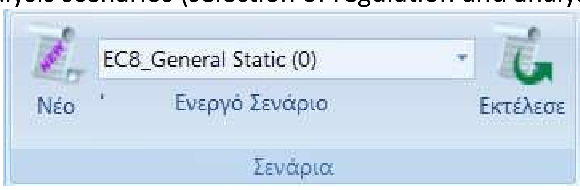
```
d001 - WordPad
File  Edit  View  Insert  Format  Help
===================================================================================
                 ΥΠΟΛΟΓΙΣΜΟΣ ΤΩΝ ΦΟΡΤΙΩΝ ΑΝΕΜΟΥ / ΧΙΟΝΙΟΥ
               ΣΥΜΦΩΝΑ ΜΕ ΤΗΝ ΕΝ 1991-1-3/4:2005 NAD GREECE
===================================================================================
ΔΕΔΟΜΕΝΑ ΓΙΑ ΤΟ ΧΙΟΝΙ
-----------------------
ΤΟΠΟΓΡΑΦΙΑ                    : Κανονικές Συνθήκες
ΣΥΝΤΕΛΕΣΤΗΣ ΕΚΘΕΣΗΣ Ce                                   : 1.00
ΘΕΡΜΙΚΟΣ ΣΥΝΤΕΛΕΣΤΗΣ Ct                                  : 1.00
ΚΑΤΑΣΤΑΣΗ ΣΧΕΔΙΑΣΜΟΥ                                     : Case A (Συνήθης Χιονόπτωση/Ι
ΣΥΝΤΑΕΣΤΗΣ ΓΙΑ ΕΞΑΙΡΕΤΙΚΑ ΦΟΡΤΙΑ Cesl                    : 1.00
ΠΥΚΝΟΤΗΤΑ ΧΙΟΝΙΟΥ γ(Kn/m^3)                              : 3.00
Ζώνη ΙΙΙ (Υπόλοιπη Χώρα)
ΦΟΡΤΙΟ ΧΙΟΝΙΟΥ (ΣΤΗ ΣΤΑΘΜΗ ΤΗΣ ΘΑΛΑΣΣΑΣ) Sk,0 (Kn/m^2) : 0.80
ΥΨΟΜΕΤΡΟ A(m)                                            : 500.0
ΦΟΡΤΙΟ ΧΙΟΝΙΟΥ (ΣΤΟ ΥΨΟΜΕΤΡΟ 500.00m)Sk (Kn/m^2)        : 1.04
===================================================================================
ΔΕΔΟΜΕΝΑ ΓΙΑ ΤΟΝ ΑΝΕΜΟ
-----------------------
ΤΥΠΟΣ ΕΔΑΦΟΥΣ : 0 Θάλασσα ή παράκτια περιοχή ανοικτής θάλασσας
Z0 (m)                                                     : 0.003
Zmin (m)                                                   : 1.00
ΘΕΜΕΛΙΩΔΗΣ ΤΙΜΗ ΒΑΣΙΚΗΣ ΤΑΧΥΤΗΤΑΣ ΑΝΕΜΟΥ (m/sec)          : 27.0
ΠΥΚΝΟΤΗΤΑ ΑΝΕΜΟΥ ρ(Kg/m^3)                                : 1.25
ΣΥΝΤΕΛΕΣΤΗΣ ΔΙΕΥΘΥΝΣΗΣ Cdir                                : 1.00
ΣΥΝΤΕΛΕΣΤΗΣ ΕΠΟΧΗΣ Cseason                                 : 1.00
ΤΟΠΟΓΡΑΦΙΚΗ ΔΙΑΜΟΡΦΩΣΗ                                     : Γκρεμοί και εξάρσεις
ΤΟΠΟΘΕΣΙΑ                                                  : προσήνεμη
ΠΡΑΓΜΑΤΙΚΟ ΜΗΚΟΣ ΠΡΟΣΗΝΕΜΗΣ ΠΛΑΓΙΑΣ Lu (m)                  : 500.00
ΕΝΕΡΓΟ ΥΨΟΣ ΕΔΑΦΙΚΗΣ ΑΝΩΜΑΛΙΑΣ H (m)                        : 300.00
ΟΡΙΖΟΝΤΙΑ ΑΠΟΣΤΑΣΗ ΤΗΣ ΤΟΠΟΘΕΣΙΑΣ ΑΠΟ ΚΟΡΥΦΗ ΛΟΦΟΥ X(m)   : 150.00
ΚΑΤΑΚΟΡΥΦΗ ΑΠΟΣΤΑΣΗ ΤΗΣ ΤΟΠΟΘΕΣΙΑΣ ΑΠΟ ΚΟΡΥΦΗ ΛΟΦΟΥ Z(m)  : 150.00
ΣΥΝΤΕΛΕΣΤΗΣ ΤΟΠΟΓΡΑΦΙΚΗΣ ΔΙΑΜΟΡΦΩΣΗΣ C0(Z)                 : 1.00
ΣΥΝΤΕΛΕΣΤΗΣ ΤΡΑΧΥΤΗΤΑΣ Cr(Z)                                : 1.00
===================================================================================
ΣΤΕΓΕΣ
-----------------------------------------------------------------------------------
ΣΤΕΓΗ 1 ΤΥΠΟΣ ΣΤΕΓΗΣ                  : ΕΠΙΠΕΔΗ
------- ΜΗΚΗ ΠΛΕΥΡΩΝ (m)              : L1=27.20  L2=14.10  L3=27.20  L4=14.10
        ΤΥΠΟΣ ΑΚΡΟΥ ΣΤΕΓΗΣ            : Αιχμηρό άκρο
        ΠΛΗΘΟΣ ΑΝΟΙΓΜΑΤΩΝ             : 1
        ΓΕΩΜΕΤΡΙΚΑ ΣΤΟΙΧΕΙΑ (m)       : h1= 6.50   h2= 6.50
                                      : a0= 7.50   a1= 0.00   a2= 0.00   a3= 0.00
For Help, press F1                                                          NUM
```

4. ANALYSIS

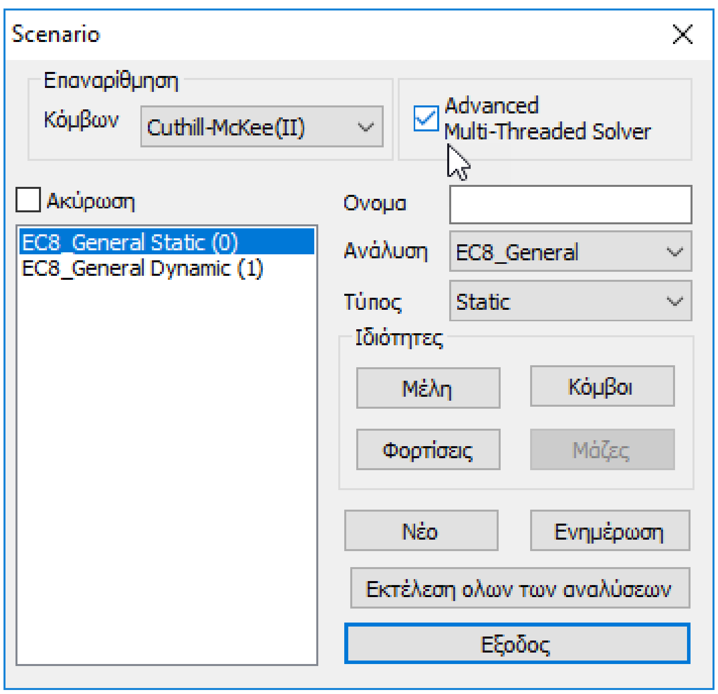
After the completion of the modeling of the structure and the input of the loads in the members, the analysis of the design based on the regulation you will define, the automatic creation of the load combinations and the results of the checks that will be obtained.

4.1 How to create an analysis script:

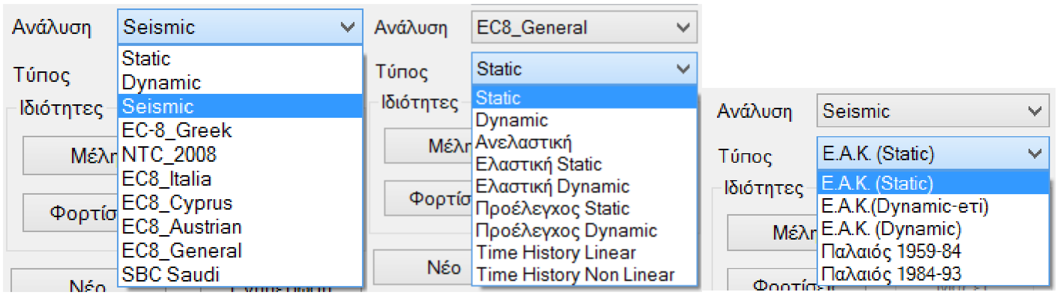
Within the "Analysis" Module, the commands of the "Scenarios" group allow the creation of the analysis scenarios (selection of regulation and analysis type) and their execution.



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: 'DESIGN OF METAL CONSTRUCTION'

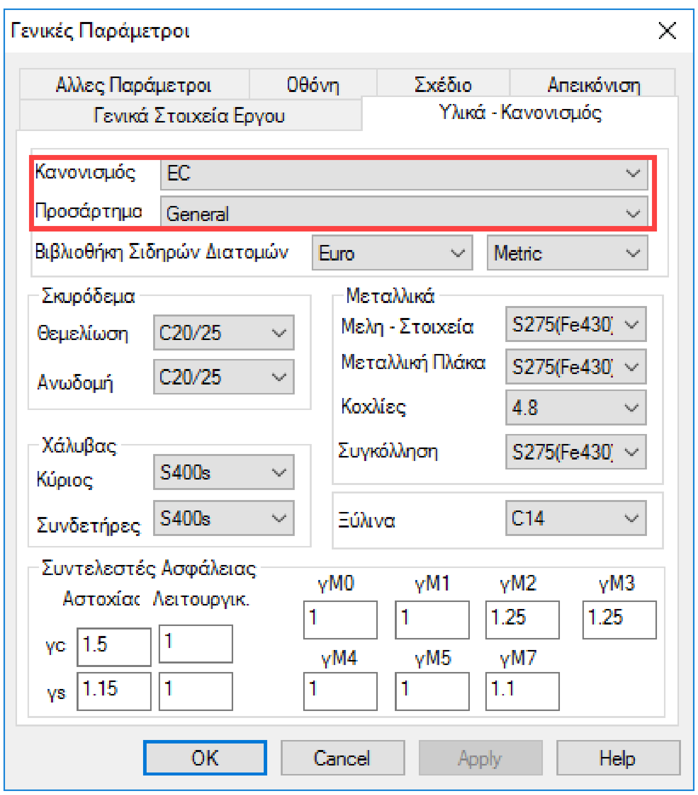


Select from the "Analysis" list and the corresponding "Type" list and click on 

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: 'DESIGN OF METAL CONSTRUCTION'

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 displacement 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 seismic involvement actions
- EC 8 Greek static	Structural analysis based on Eurocode 8 and the Greek Appendix
- EC8 Greek dynamic	Dynamic analysis based on the Eurocode 8 and the Greek Appendix
- EC 8 English Pre-test Static	Pre-testing based on the CAN.EPE
- EC8 Greek Pre-Control Dynamic	Pre-testing based on the CAN.EPE
- EC 8 Greek Time History Linear	Static analysis based on Erocode 8
- EC 8 Greek Time History Non Linear	Dynamic analysis based on the Code 8
- EC 8 English Elasticity	Anelastic seismic analysis based on the 8 or the EDPC.

For abroad:

ELASTIC - UNELASTIC

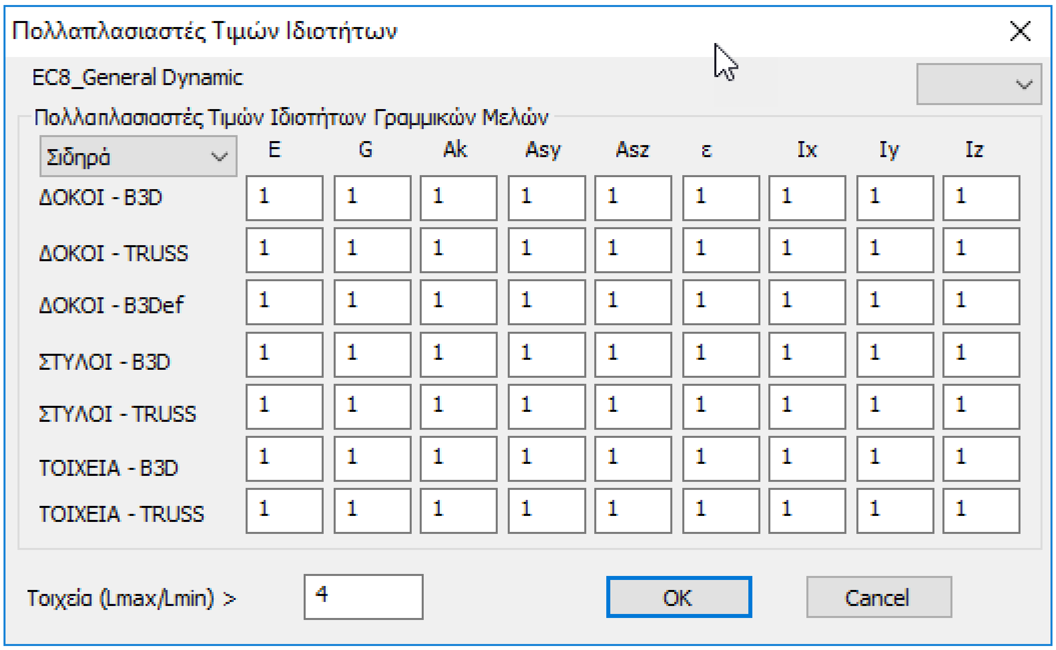
- NTC 2008	Seismic analysis based on the Italian 2008 regulation
- EC8 Italy	Seismic analysis based on the Eurocode 8 and the Italian Appendix
- EC8 Cyprus	Seismic analysis based on the Eurocode 8 and the Cyprus Appendix
- EC8 Austrian	Seismic analysis based on the Eurocode 8 and the Austrian Appendix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

EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'

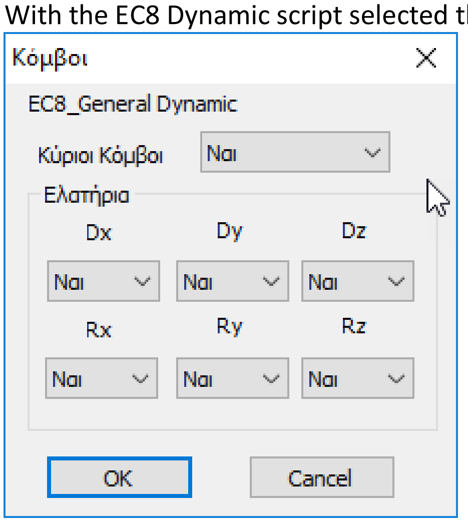
For this example you will run the EC8 dynamic scenario for the earthquake, as well the scenarios Snow Typical, Wind 0 and Wind 90, which were automatically created by the program in the previous step.

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:

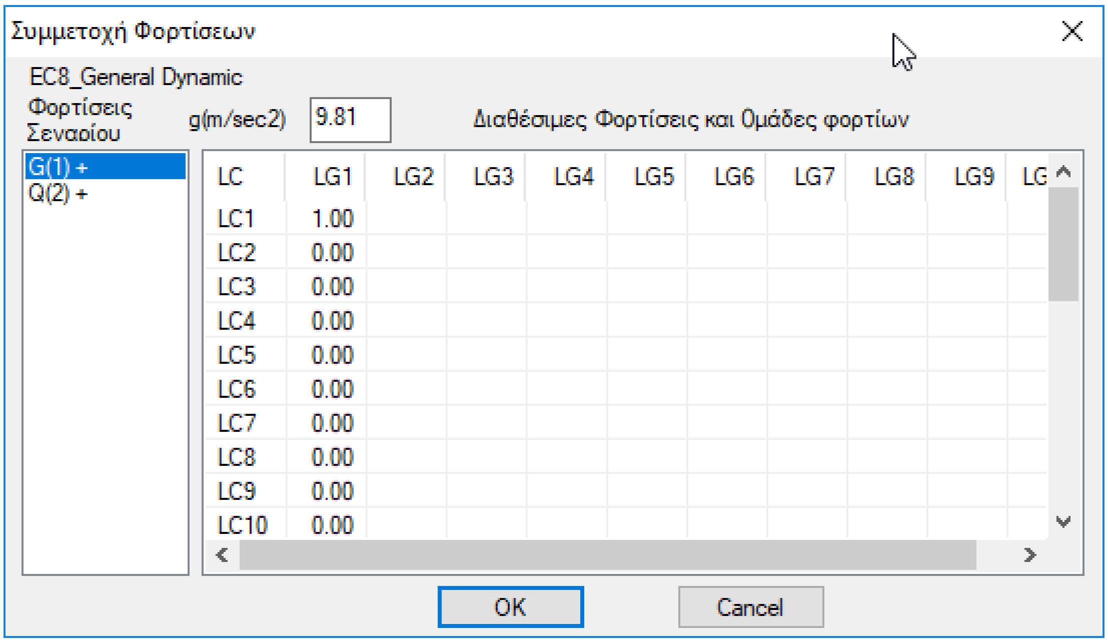


EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'

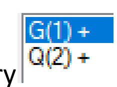
Here you can specify to resolve your girder without diaphragm mode altogether, even if there are diaphragm nodes, as well as to resolve it pressed (Spring No) even if elastic foundation is set.

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 

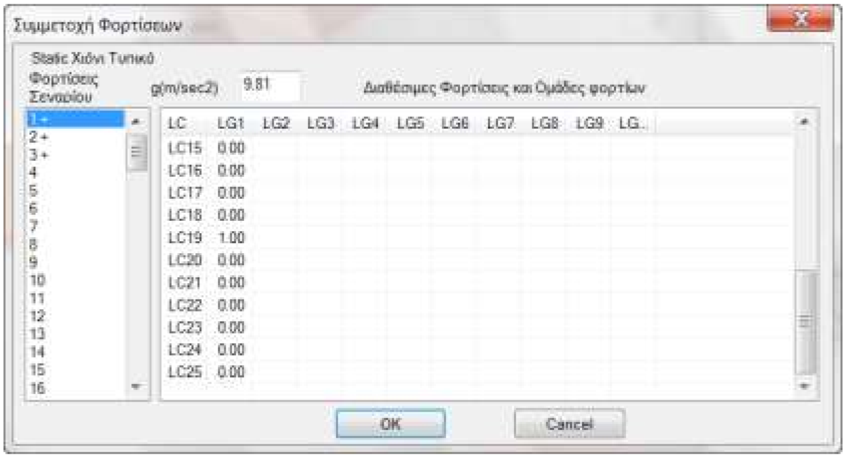
indicates that there is a charge participation for that particular charge.

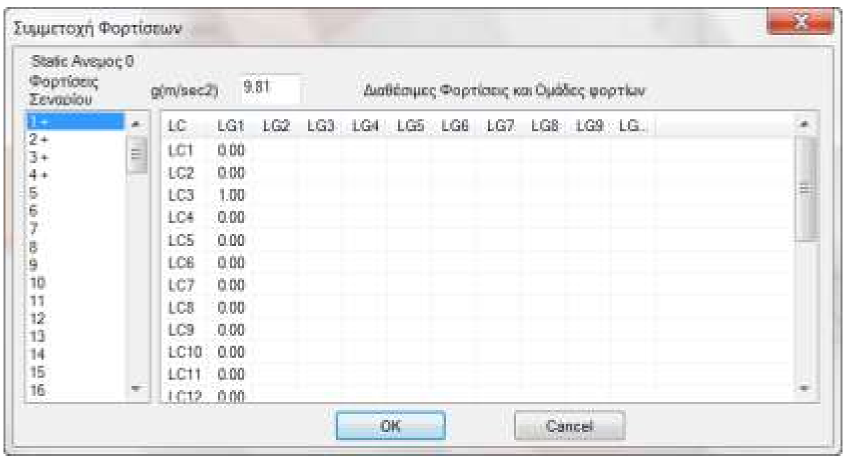
to update the script and register the changes.

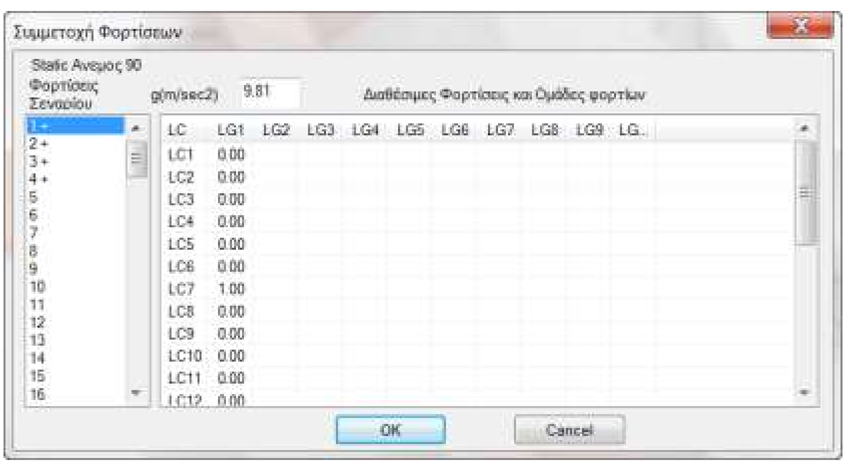
⚠ *The program automatically populates the unit in the corresponding charge, so any modification is, again, optional.*

EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'

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.





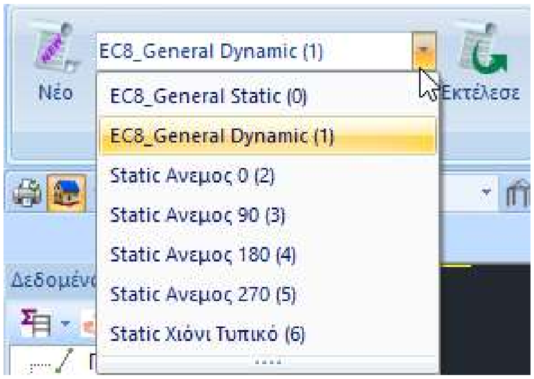


When each charge is activated, the+ symbol appears next to the charge number.

OBSERVATION

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

4.2 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:

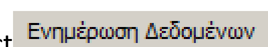
Alternatively, the new **Run all analyses** command 

allo

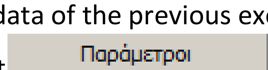
ws you to run all the scripts in the list with one click.

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 

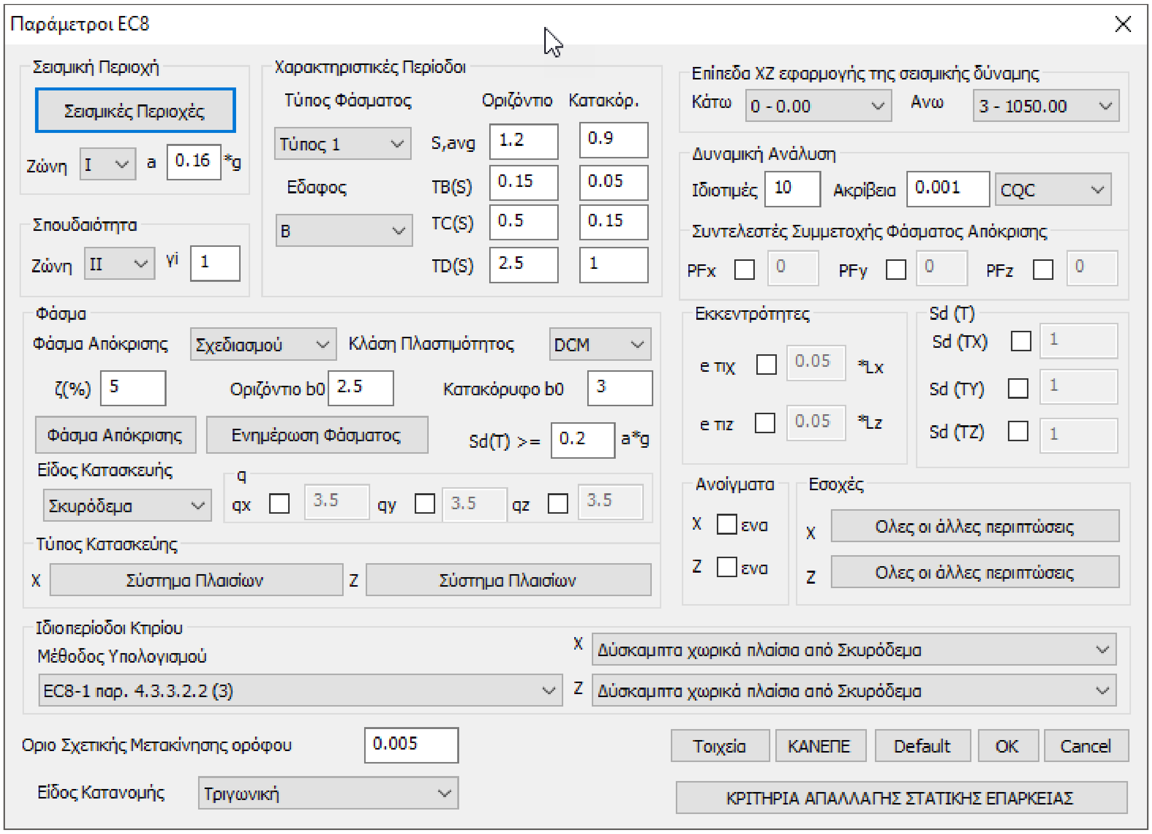
to update the parameters of the active script and delete the data of the previous execution process.

Then, select 

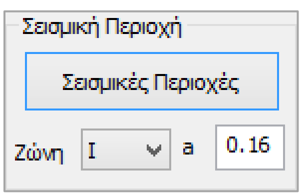
to set the parameters of the specific study.

⚠ *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:*

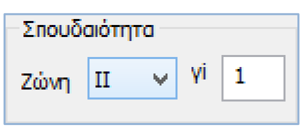
EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'



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.

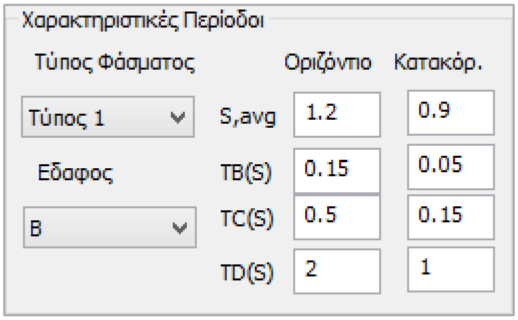


Select the seismic zone, after first checking the file that appears by clicking on "Seismic Areas" for the number of the zone corresponding to the municipality where your study belongs. Select the number from the "Zone" list and the factor "a" is automatically filled in.



Related select the "category importance" for to automatically fill in the importance factor "y".

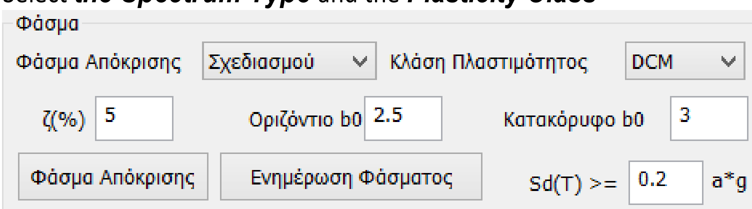
EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'



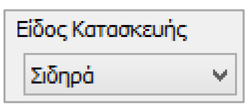
Then you define the type of spectrum (in Greece type 1 is used) and the soil category, so that the coefficients for the horizontal and vertical spectrum are automatically filled in.

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

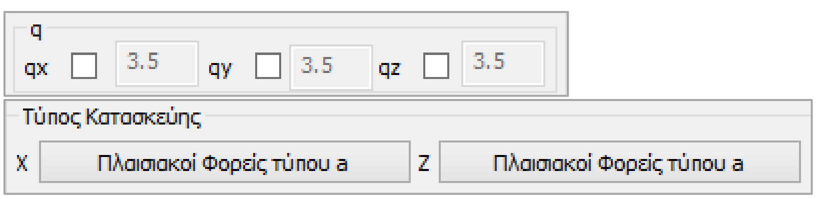
Select ***the Spectrum Type*** and the ***Plasticity Class***



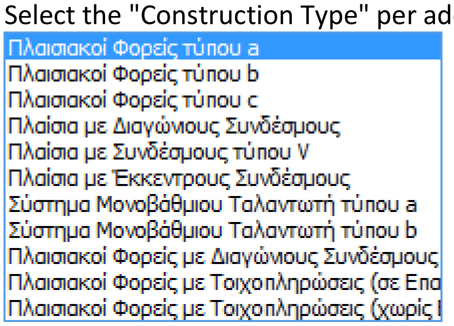
Select the ***Type of Construction***



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



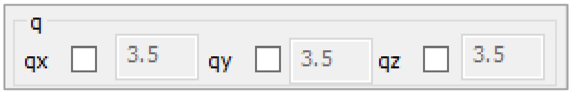
Select the "Construction Type" per address by selecting from:



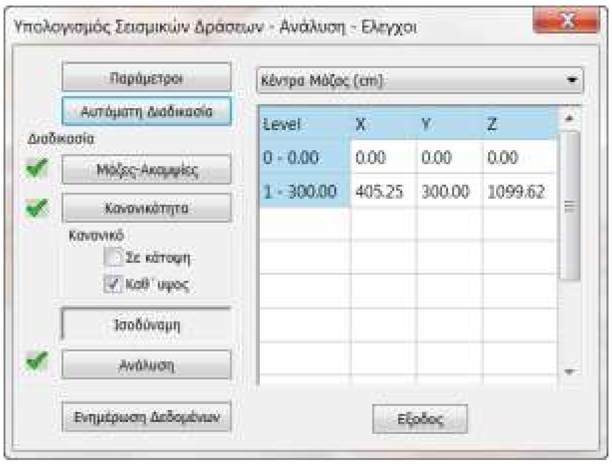
According to the Eurocode, the **"Seismic behaviour coefficient q"** is derived from a calculation and the **"Type of Construction"** from specific criteria.

⚠ *SCADA Pro automatically calculates the q and the type of construction. The procedure 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**.

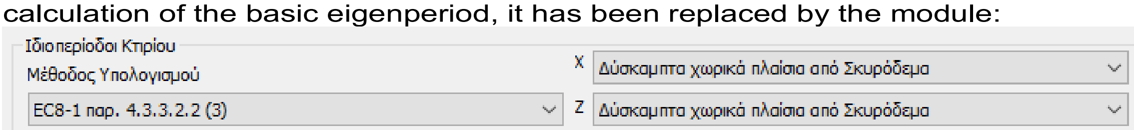


- Knowing all the previous parameters, the program calculates the "Seismic Coefficient q". Open the parameter dialog box one more time. In the "q" field read the values calculated by the program.

- 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).

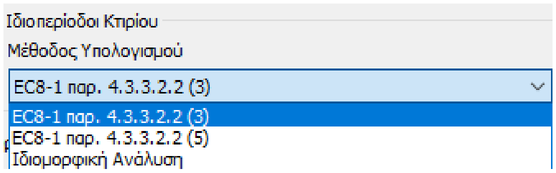
In the ***Building Properties*** field:

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.

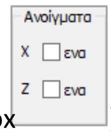
EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'



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 a single frame, the following shall be activated the corresponding checkbox in the "Openings" box



)

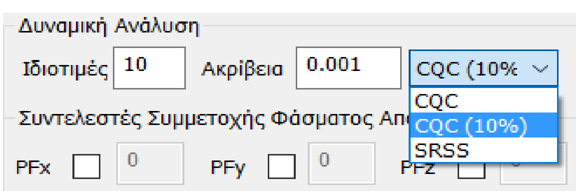
2. The second approximate method EC8-1 παρ. 4.3.3.2.2 (5) , is sufficient to be selected

and does not require any additional action.

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

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

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

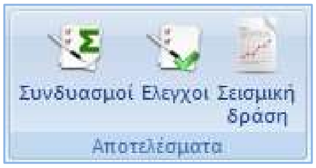


It is also possible to choose the mode of overlap of the eigenmodal responses either according Full Quadratic Parallelism CQC and CQC(10%) rule (3.6 EAK), or the Simple Quadratic Parallelism SRSS rule.

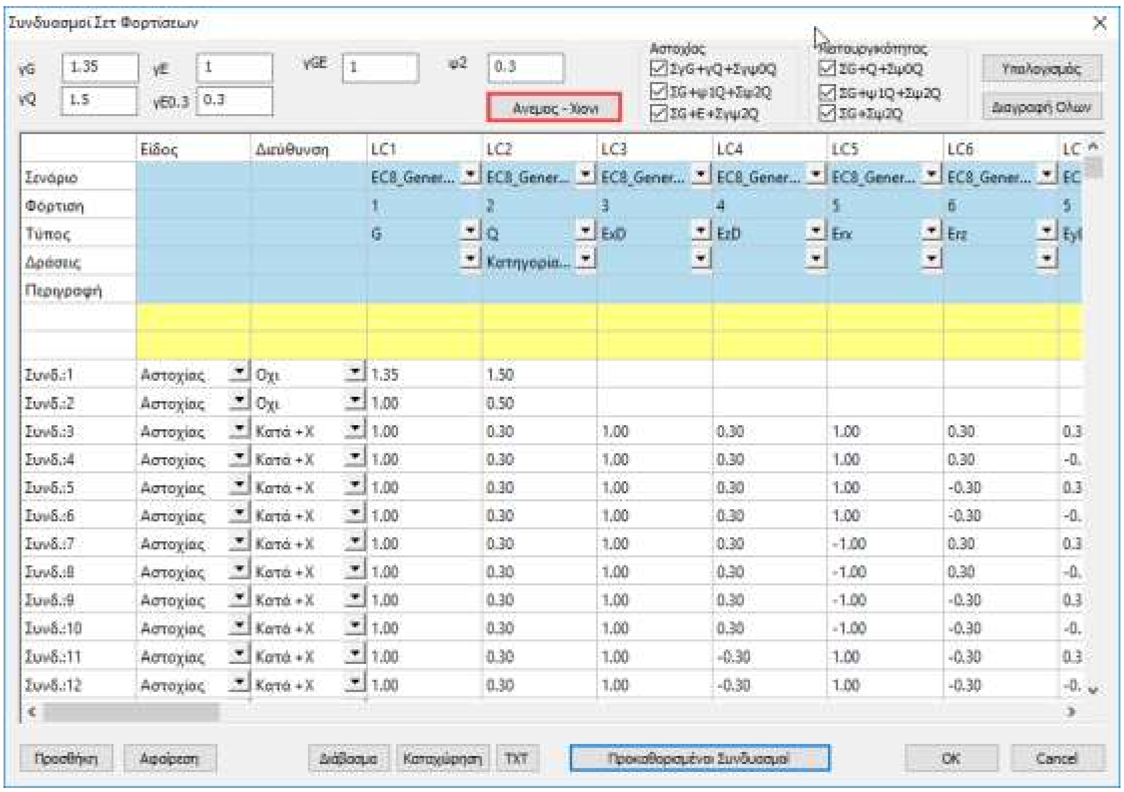
Also, the results of the seismic action now include the results of the eigenmode analysis for the static scenarios.

4.3 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:

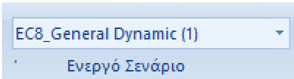


Selecting the "Combinations" command opens the "Load Set Combinations" dialog box where you can create your own combinations or call the predefined ones included in the program.



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

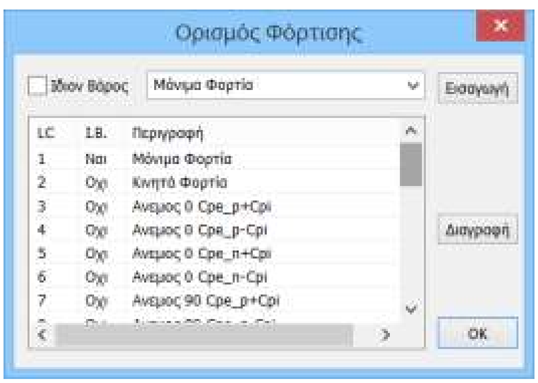


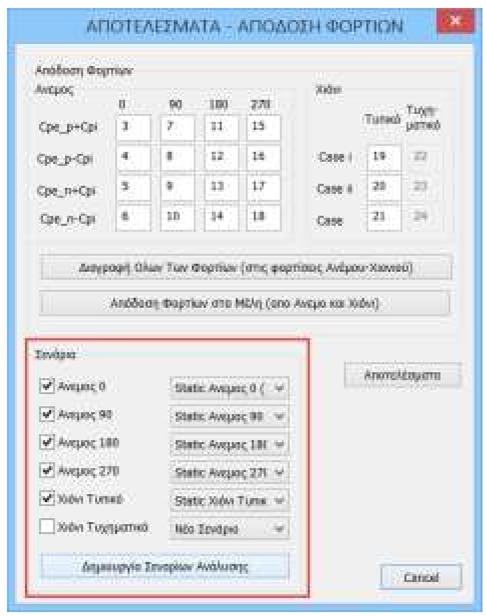
The predefined combinations of the "running" scenarios of the analysis are automatically entered by the program.

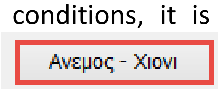
In addition to the predefined combinations, the designer has the possibility to create his own combination files, either by modifying the predefined ones, or by deleting all of them "Delete All" and entering his own values. The "Load Set Combinations" tool works like Excel page offering copy, total delete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

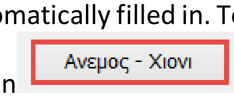
The predefined combinations refer to seismic scenarios. To create combinations of scenarios that do not contain an earthquake, both **automatic** and **manual** modes are available.

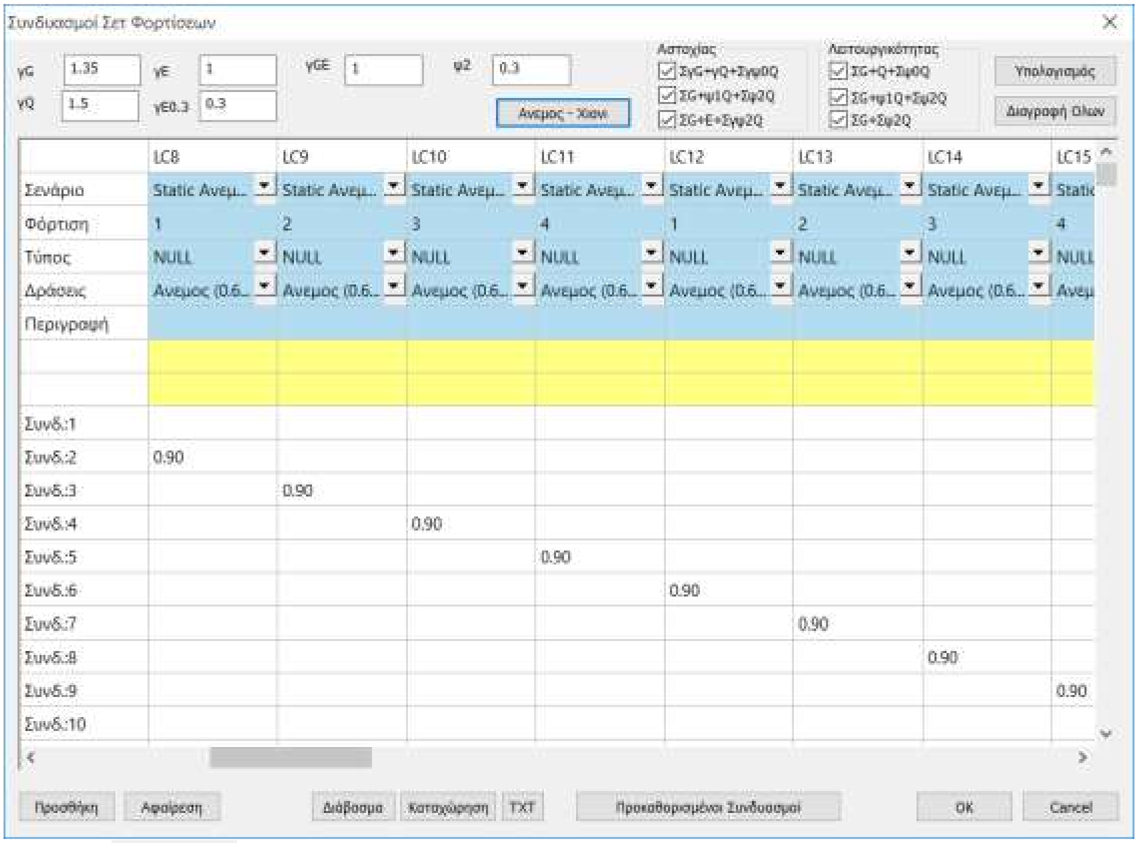
The **automatic** mode assumes that the automatic procedure for the calculation and distribution of wind and snow loads, the automatic creation of the loads and combinations, as in the example above, has been carried out beforehand.

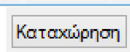




Subject to the above conditions, it is possible to create the wind and snow combinations automatically using the  command.

So, after first running the earthquake scenario and all the 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  . The coefficients of the wind and snow scenarios are automatically filled in, providing a complete file of combinations of all the study loads.



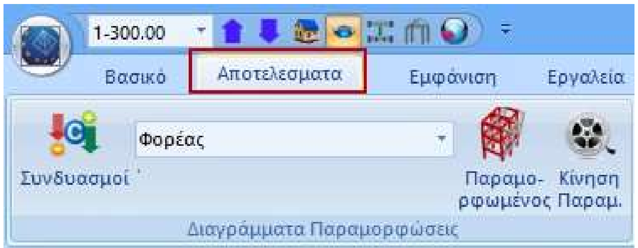
Select 

to save it so you can use it for sizing.

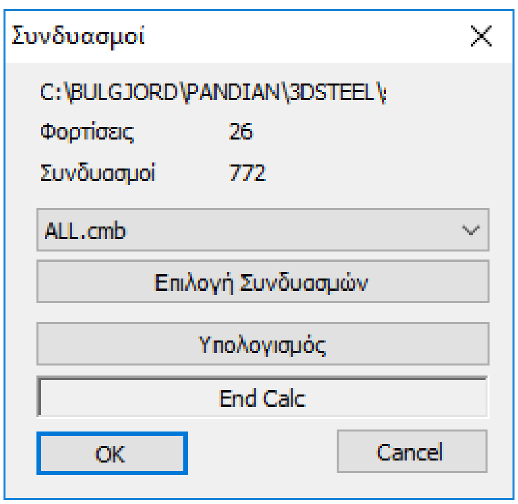
5. RESULTS

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.



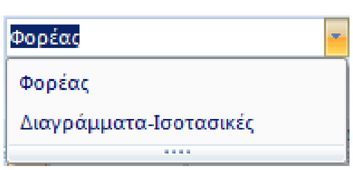
Depending on the results you want to see, from the "Combinations" command and within the dialog box:



- Select combined From from list which includes their combinations all of "running" analyses, and let them complete their calculation automatically, or

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

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

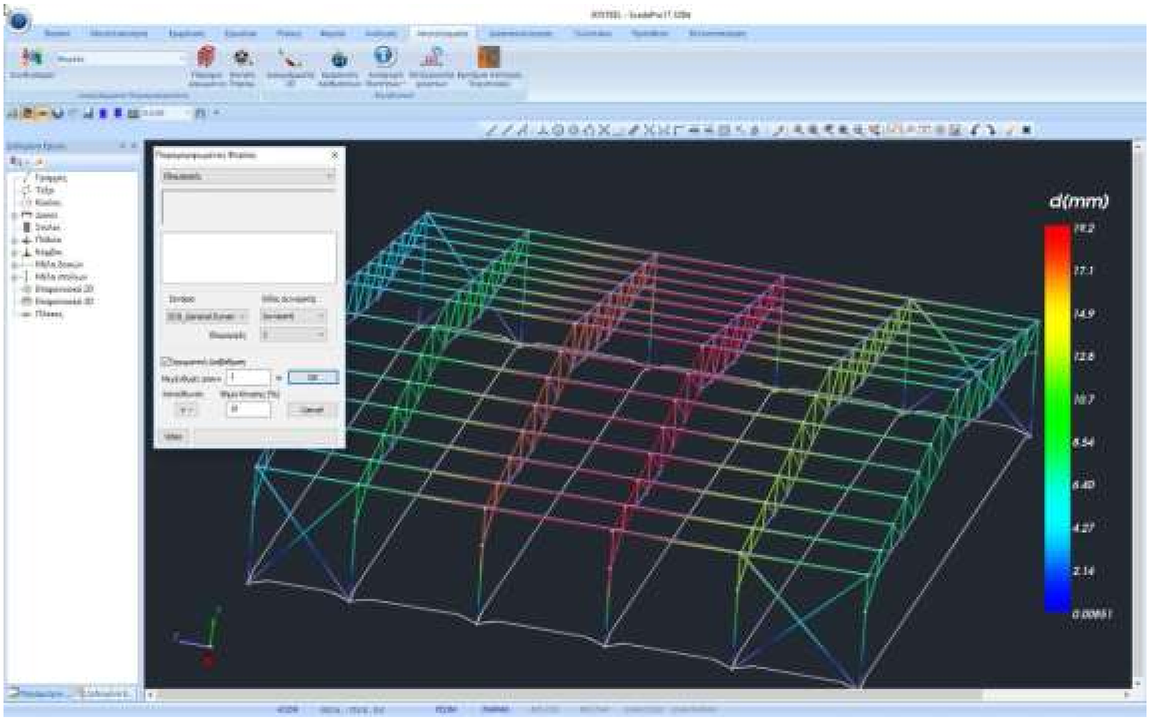


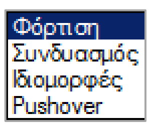
From the list on the right, depending on the results you want to see, select:

√ Institution or

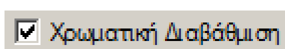
√ Charts-Important

5.1.1 Body+ "Deformed Body"



Select from the list 

select 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 

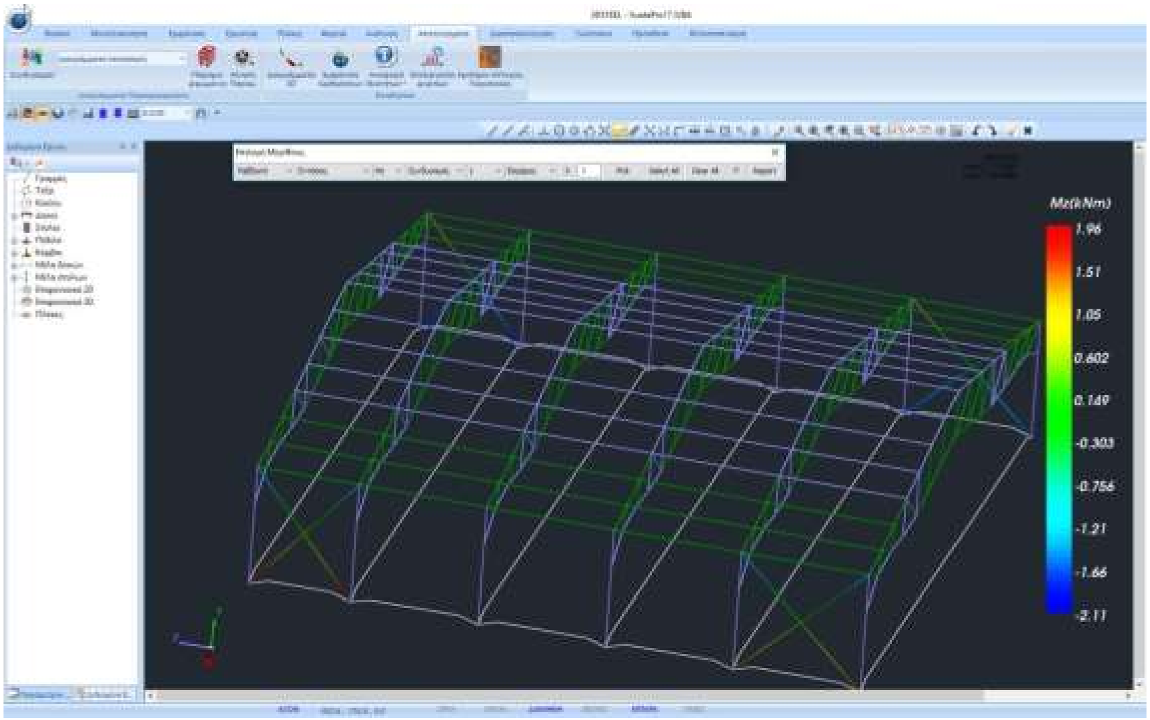
, modify the "Scale" and the "Motion Step" to see the best and most intuitive visualization.

At "Status bar" select (double click, blue=active, grey=inactive) the way to display the deformed vector.

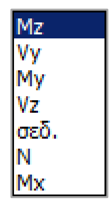
The color bar shows the configuration of the distortion value.

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.

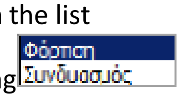
5.1.2 Charts - Equalisation



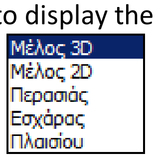
In this section you can see on the members the diagrams of the stresses for the linear members, and the isometric curves of stresses, strains and reinforcements for the finite surface elements. In particular, to see for the Ribbon elements

the diagrams of intensive sizes select the intensive size from the list 

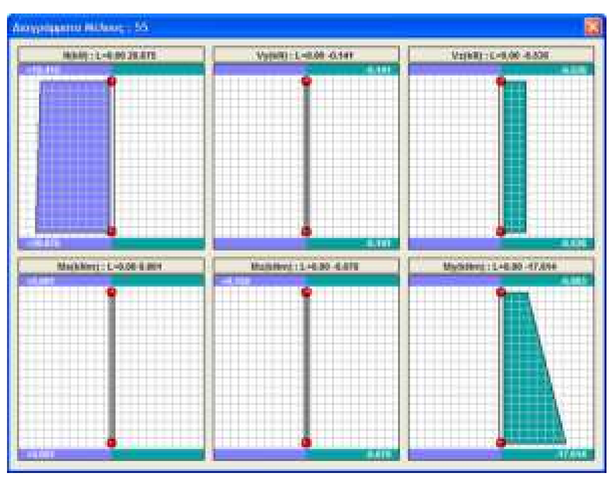
, in the

then select the type of charge or combination or surrounding 

and finally select the way to display the

chart 

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 .

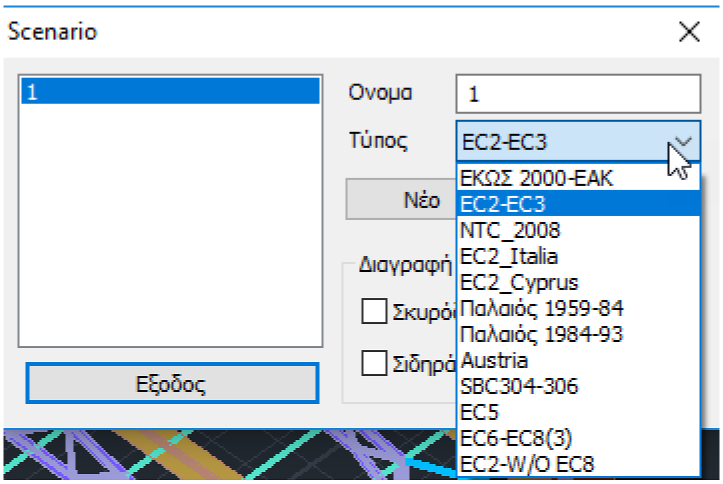


6. DIMENSIONING OF METALLIC SECTIONS

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

6.1 How to create dimensioning scripts :

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

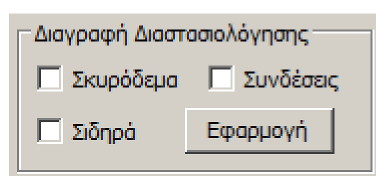


Type a name, select a type and New to populate the list of scenarios.

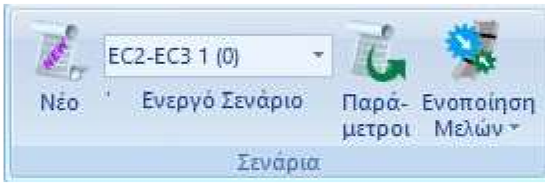
In this example, a Eurocode scenario was used.

Comment: For metals, EC3 is applied through the program and is included 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 dimension from scratch using other combinations, or parameters, or scenario, etc.

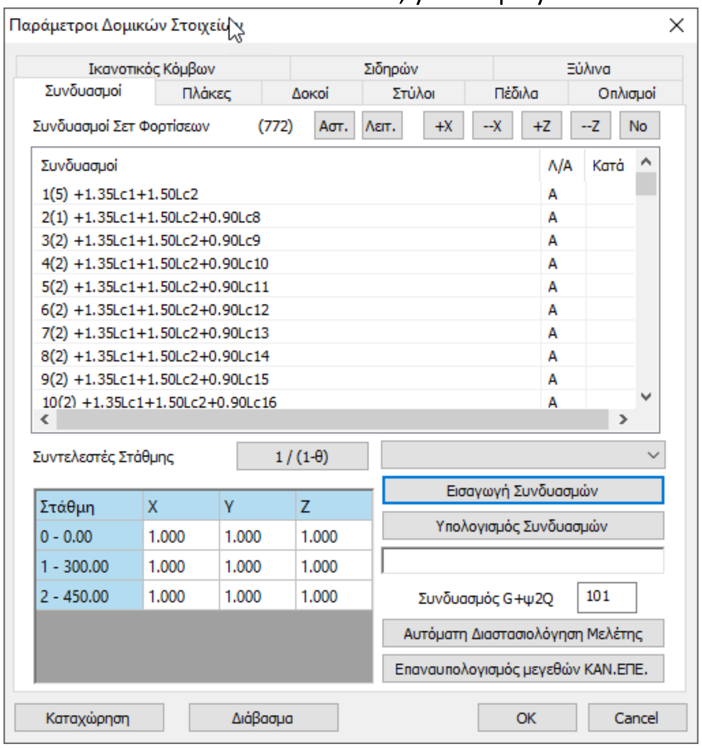


6.2 How to determine the parameters of the dimensioning metal sections:

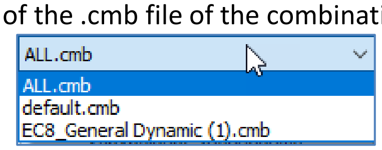


From the list of scenarios you have created, select the scenario you will use for sizing.

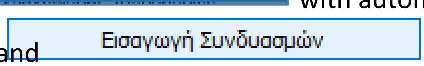
With the selected scenario active, you display the Parameters



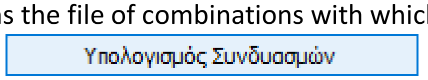
A prerequisite for sizing is the calculation of combinations.
The selection of the .cmb file of the combinations entered by the analysis is either:

- 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

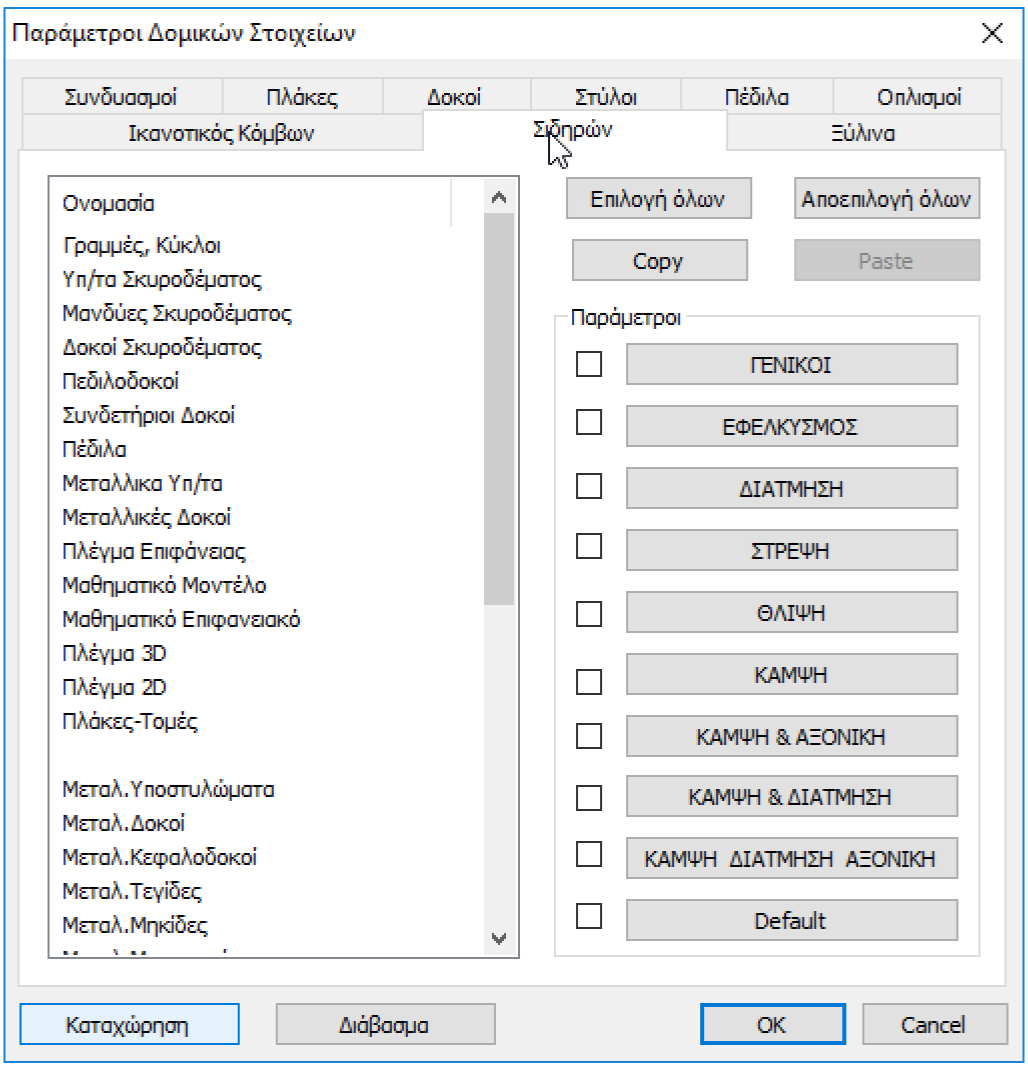
and then via the button 

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: 'DESIGN OF METAL CONSTRUCTION'

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 layers and on the right is a list of controls, each containing the corresponding parameters 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.



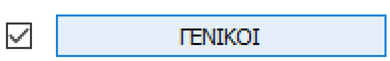
The definition of the sizing parameters of the metallic sections is done on a layer-by-layer basis. Select the layer whose parameters you want to set (e.g. Metallic Lodges) and select the layer you want to set the parameters for.

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, lets 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 you select the "Copy" button and select the Metal Beams layer and press the "Paste" button which is already 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.

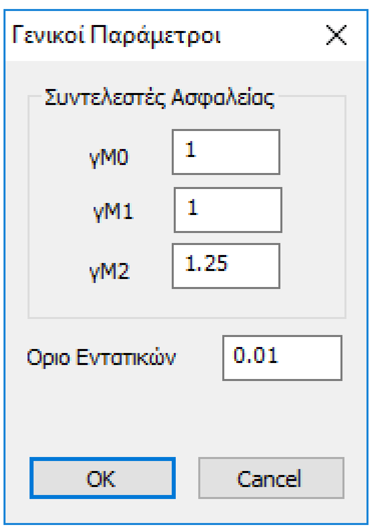
It should also be noted that to set parameters at least one (or more) layer must be selected.

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

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



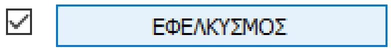
to set the safety factors γM:



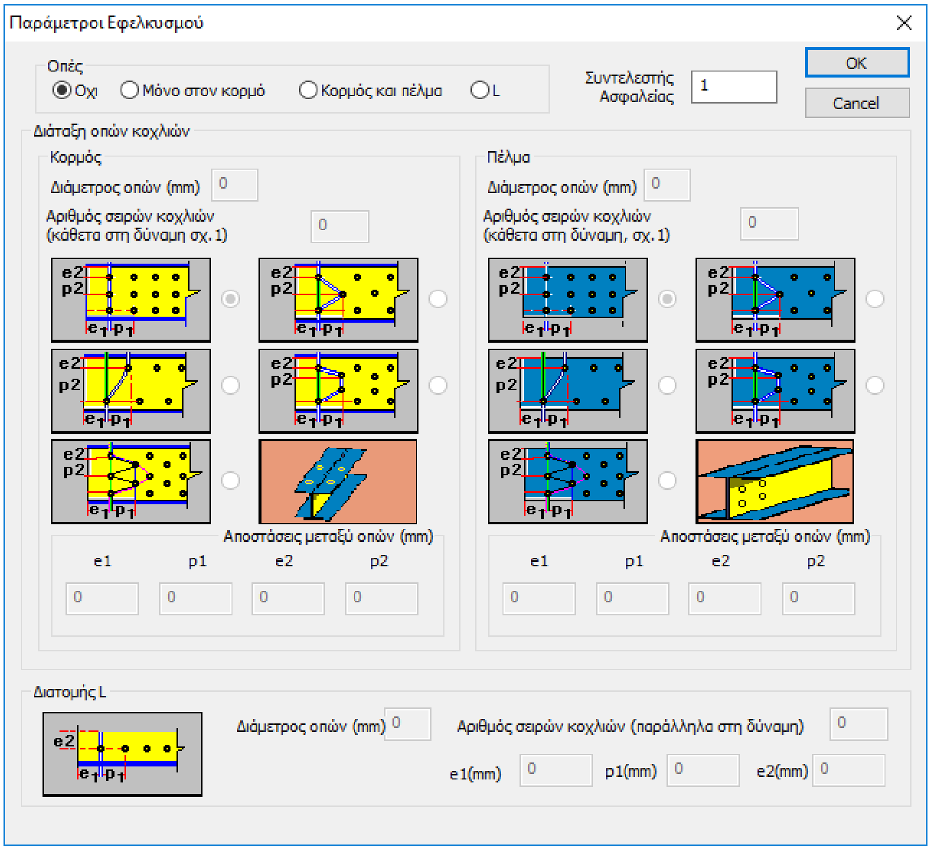
γM0= transverse stress resistance for each category of members γM1 = resistance to buckling based on tests cM2= tensile fracture strength of cross sections

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.

- "**EFFELCISM**"



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

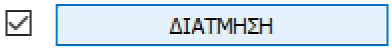


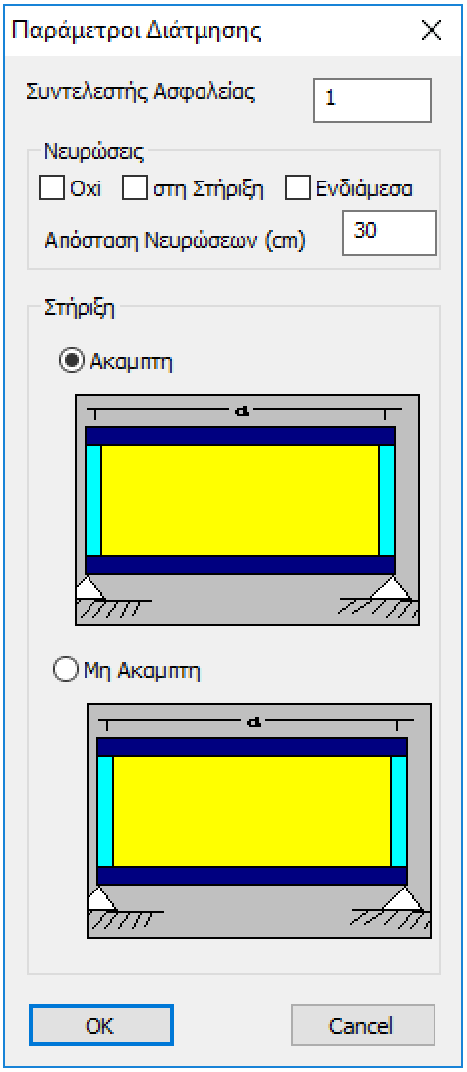
For the holes define the distances from the ends, the diameter and the number of rows on the torso and tread.
In the case of an L section, set the parameters at the bottom of the box.

The rationale here is to specify whether the application during the tensile test will 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.

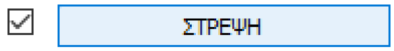
- "**DISTRICT**"

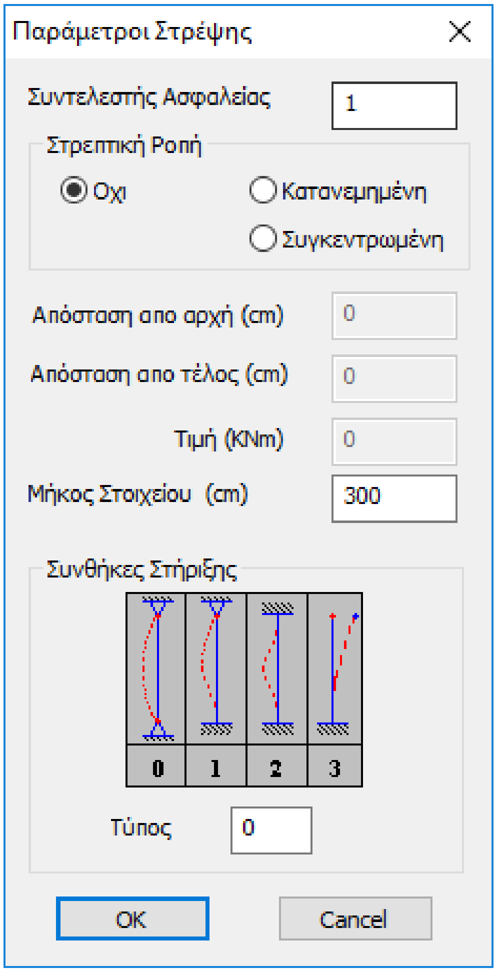




Here you define whether the elements of this Layer have ribs or not and if they do, where they are present (on the support and/or the trunk). You also define the spacing of the ribs as well as whether the support of an element is rigid or not.

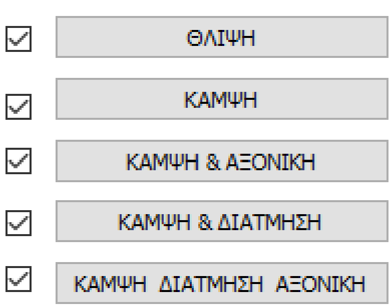
- "**STRIP**"

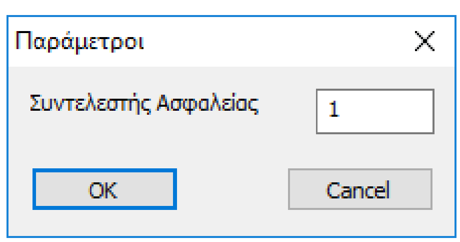




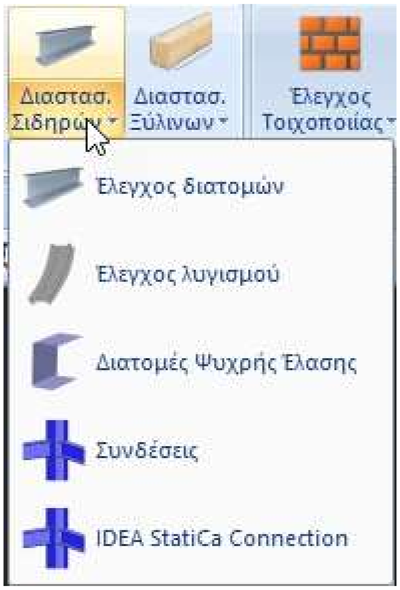
Here you specify whether the members of the layer are loaded by torsional moment (distributed or concentrated). If they are loaded, you define the elements of the loading. You also specify the support conditions of the members based on the support type shown in the graph.

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.





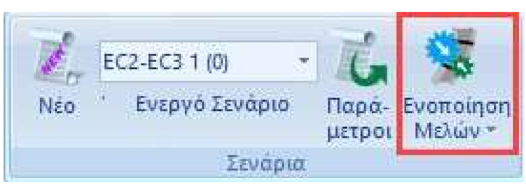
6.3 Dimensioning of steel sections:



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

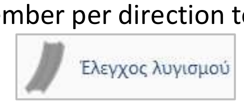
Before the Buckling Check and in order to obtain the correct buckling lengths of the members, a new set of commands is used to calculate them:

6.3.1 Consolidation of Members

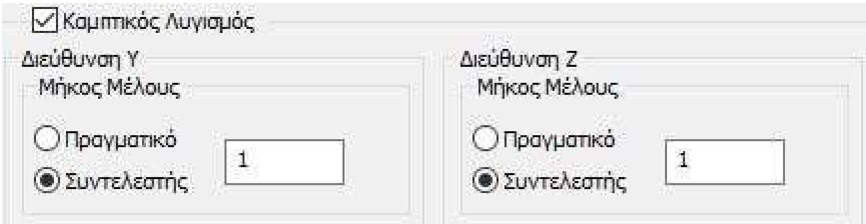


In the new version of the program a new group of commands has been added which concerns the consolidation of metallic members for the calculation and display of bending and deformation checks based on EC3.

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 the bending. 

⚠ This determination was previously made using the known rates:

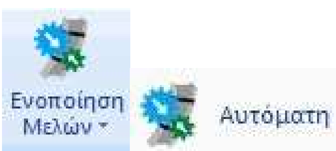


- Now with the use of the consolidation by direction, the process of the rates will not be needed, but the consolidation will be done, in most cases automatically.
- It should also be noted that with consolidation process the bending length is correctly calculated and in the printing of the results a 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, as a rule, we take **unified length Ly** in the direction where the local y-y axis is parallel to the elements supporting - securing the member, while in the other direction, if there are no elements, **the individual lengths** are taken as **Lz**.

We select the consolidation command group and 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.

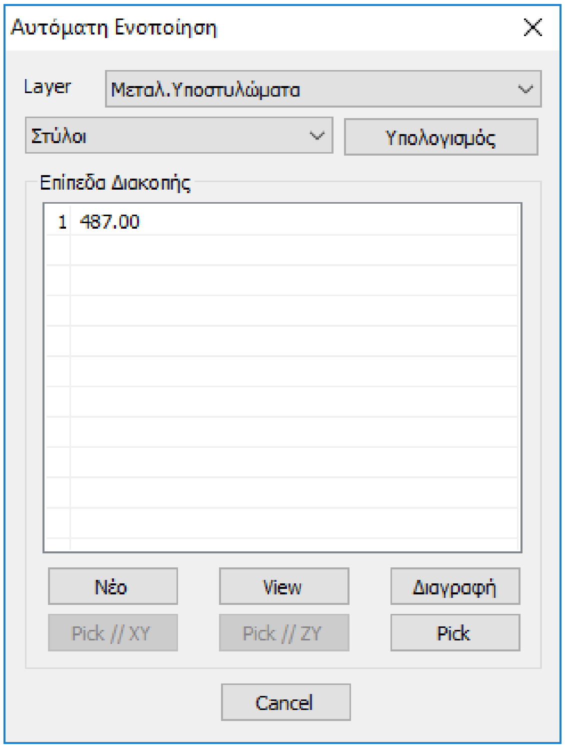
Break planes are horizontal or vertical planes that are used as breakpoints in the consolidation of a continuous element.
Thus, for vertical elements (Pillars) the stopping levels are horizontal levels which are defined, like the levels, with an altitude.

OBSERVATION

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

6.3.1.1 Automatic Consolidation

Using this command displays the following dialog box



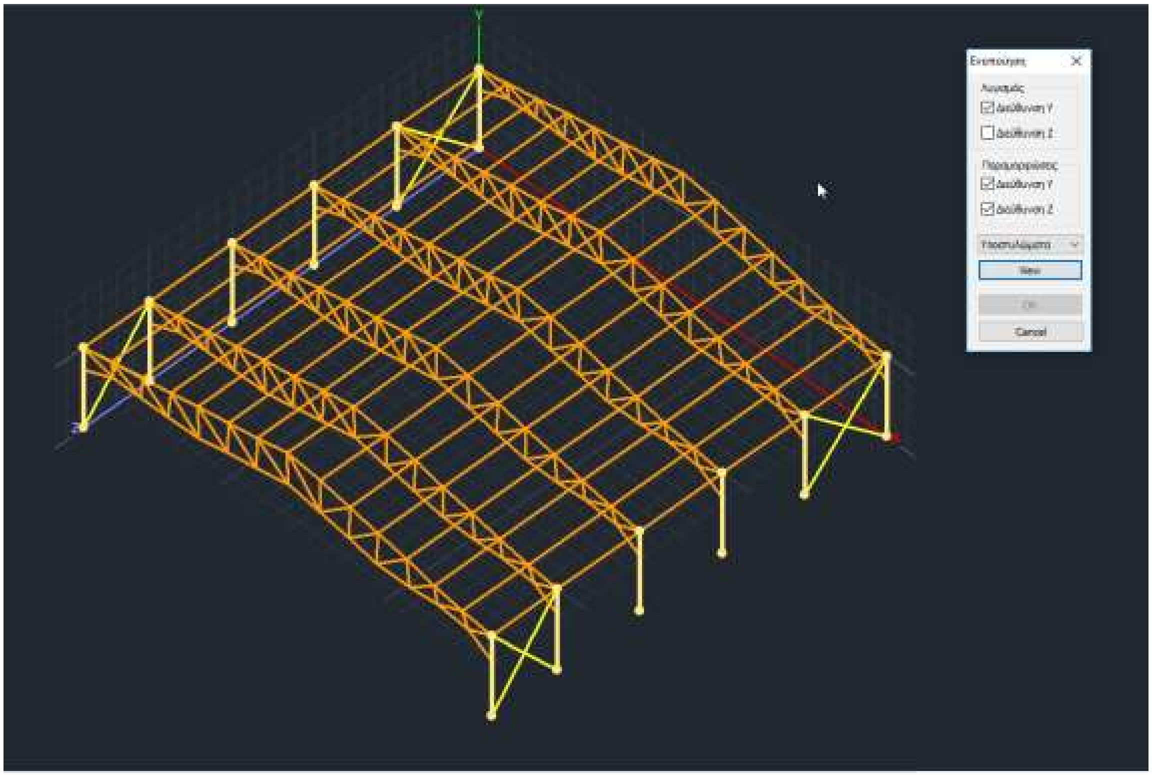
In the upper field you select the layer of the elements you want to consolidate. The

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.



<u>Interruption levels</u> are levels that are boundaries of the beams and poles where you want the integration for either one or the other direction to be interrupted.

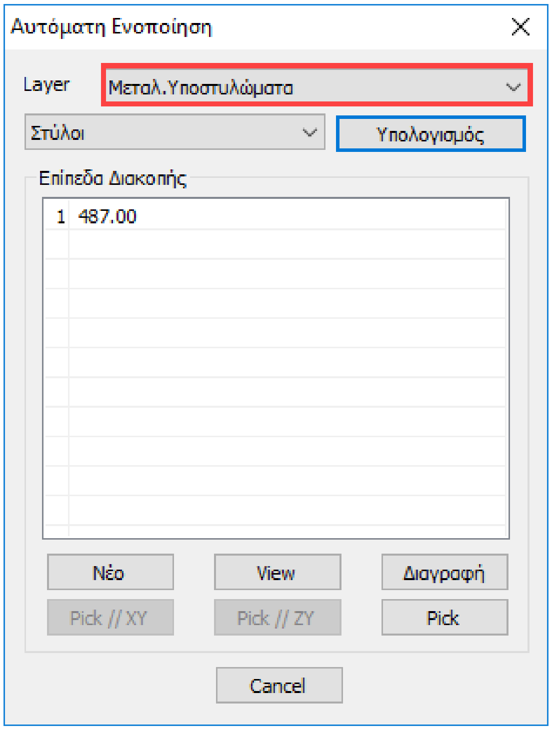
- For the poles, the stop levels are horizontal planes where they are defined by the altitude.
- For beams, the stop planes are always vertical planes defined by two points.

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.

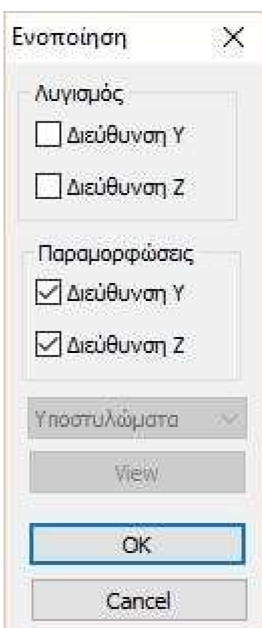
⚠ The default limits are not shown in the table of cut-off levels.

In this example, there are levels at 0.00, 487.00 and 725.00 in the table with the cut-off levels for the poles, only the level 487.00 will be indicated by default (i.e. only the intermediate level without the limits), on the grounds that if the poles are consolidated, it will be cut off at 487.00 cm, i.e. the pole will be consolidated from 0.00 to 487.00 cm.



6.3.1.2 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:

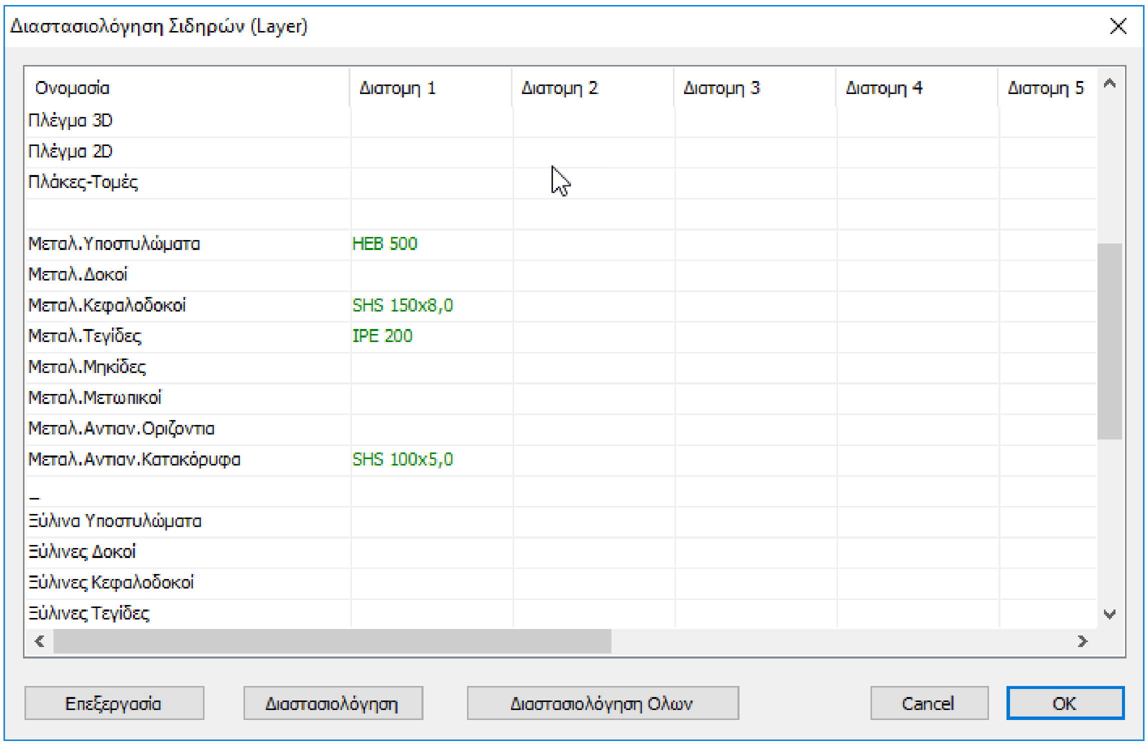


where you set the direction of unification for Bending and Deformation.

6.3.2 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.



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 HEB500 cross-section have been placed in the "Metallic Supports" layer. Similarly, the metal beams have been placed in the "Metal Beams" layer and the corresponding members in the other layers.

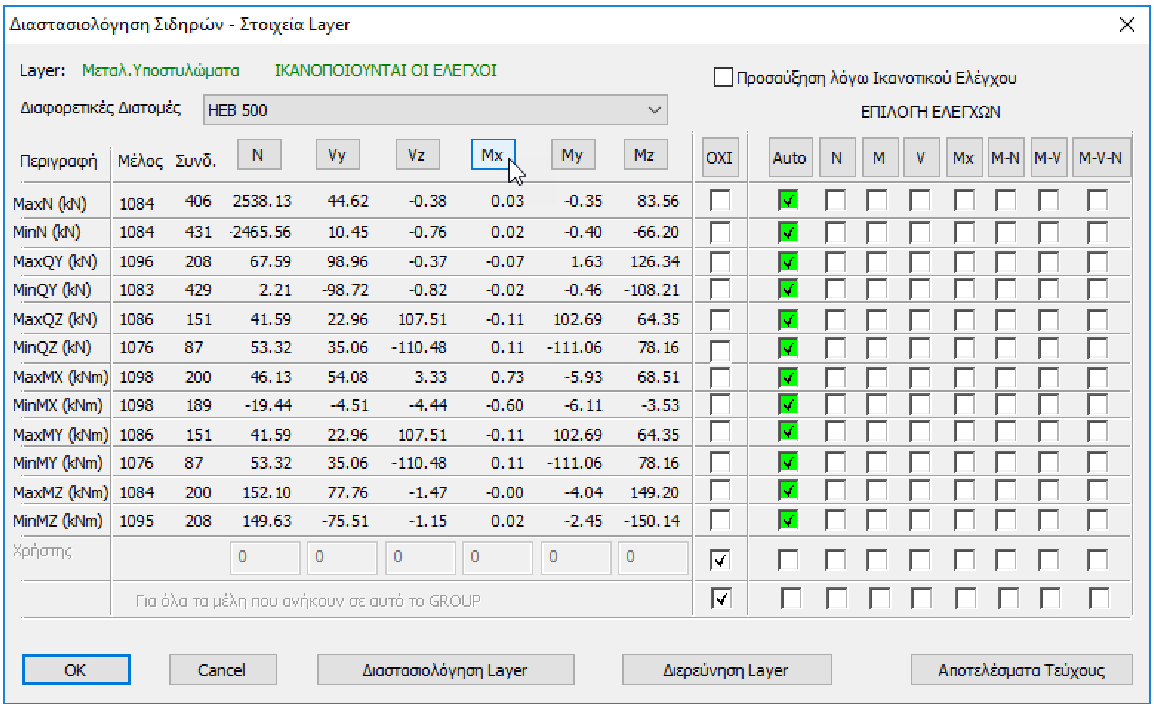
By selecting the command "**Dimension all**" cross-sections for all combinations will be automatically checked and the groups - layers in which no cross-section fails will be displayed in green and the groups in which even one cross-section has exceeded the unit, i.e. has failed, will be displayed in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

Select a 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).

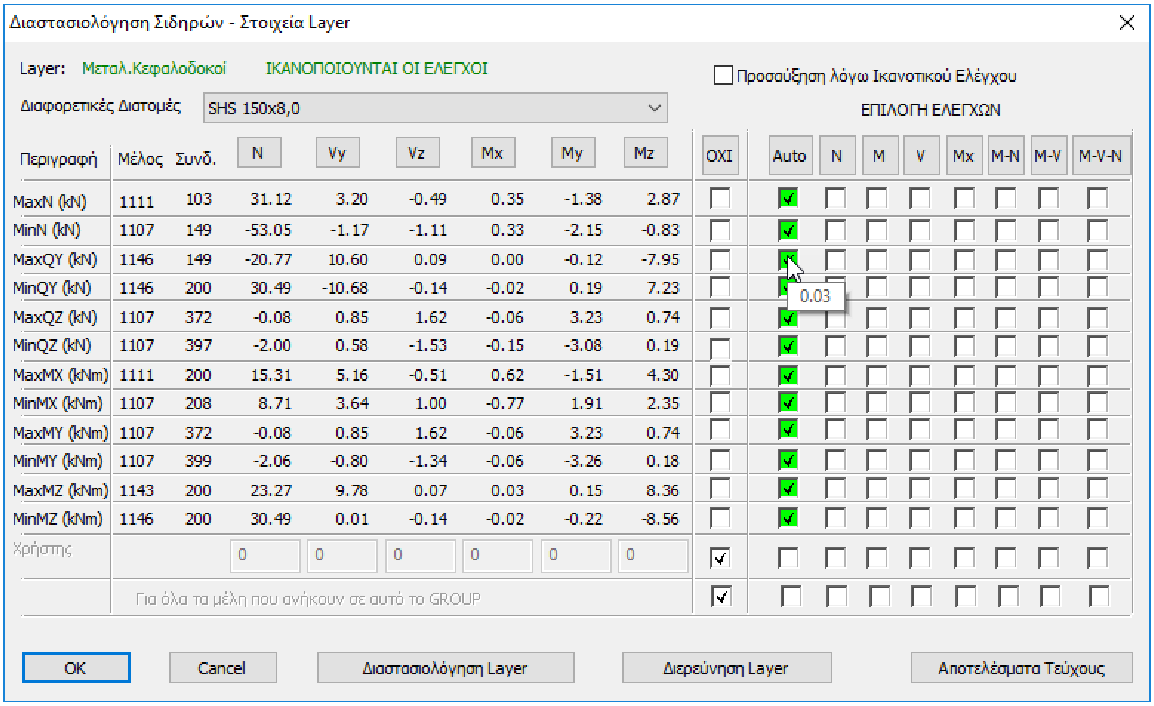


Περιγραφή	Μέλος	Συνδ.	N	Vy	Vz	Mx	My	Mz
MaxN (kN)	1084	406	2538.13	44.62	-0.38	0.03	-0.35	83.56
MinN (kN)	1084	431	-2465.56	10.45	-0.76	0.02	-0.40	-66.20
MaxQY (kN)	1096	208	67.59	98.96	-0.37	-0.07	1.63	126.34
MinQY (kN)	1083	429	2.21	-98.72	-0.82	-0.02	-0.46	-108.21
MaxQZ (kN)	1086	151	41.59	22.96	107.51	-0.11	102.69	64.35
MinQZ (kN)	1076	87	53.32	35.06	-110.48	0.11	-111.06	78.16
MaxMX (kNm)	1098	200	46.13	54.08	3.33	0.73	-5.93	68.51
MinMX (kNm)	1098	189	-19.44	-4.51	-4.44	-0.60	-6.11	-3.53
MaxMY (kNm)	1086	151	41.59	22.96	107.51	-0.11	102.69	64.35
MinMY (kNm)	1076	87	53.32	35.06	-110.48	0.11	-111.06	78.16
MaxMZ (kNm)	1084	200	152.10	77.76	-1.47	-0.00	-4.04	149.20
MinMZ (kNm)	1095	208	149.63	-75.51	-1.15	0.02	-2.45	-150.14
Χρήστης			0	0	0	0	0	0

In this example, the adequacy ratios are much smaller than unity. This means that we could use smaller cross-sections.

We select for example the Metal Layer. Headers:

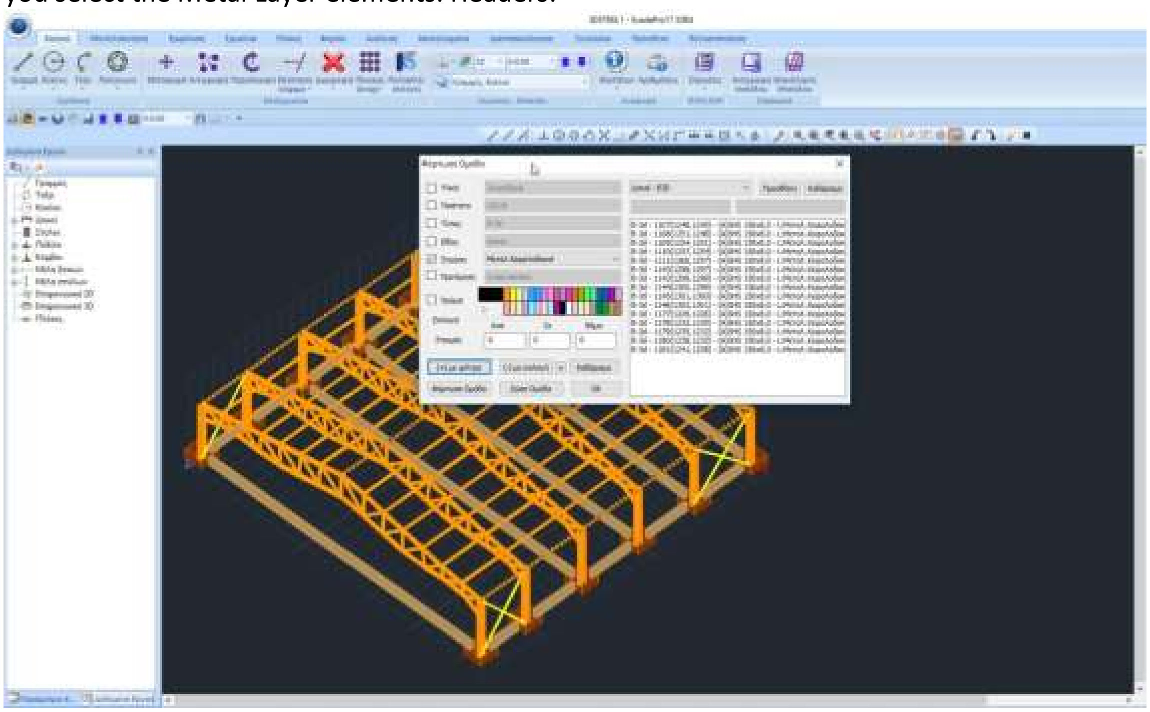
EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'



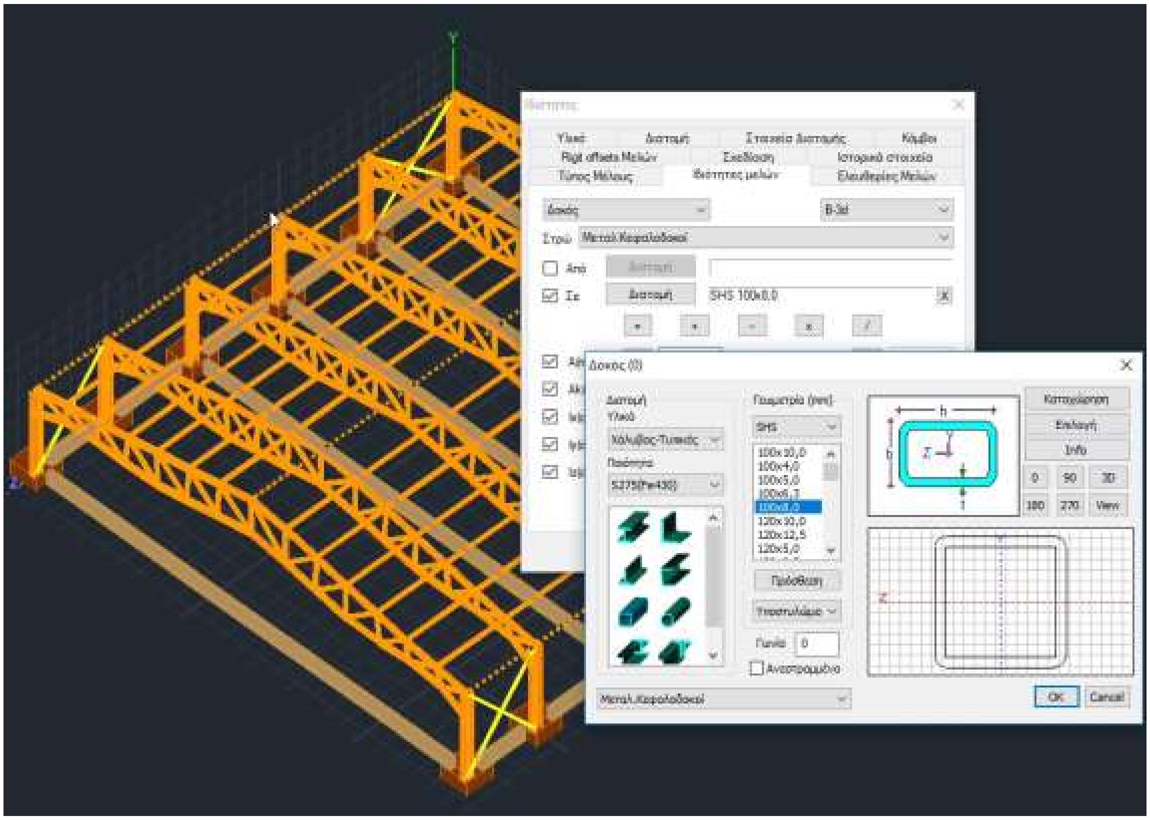
Περιγραφή	Μέλος	Συνδ.	N	Vy	Vz	Mx	My	Mz
MaxN (kN)	1111	103	31.12	3.20	-0.49	0.35	-1.38	2.87
MinN (kN)	1107	149	-53.05	-1.17	-1.11	0.33	-2.15	-0.83
MaxQY (kN)	1146	149	-20.77	10.60	0.09	0.00	-0.12	-7.95
MinQY (kN)	1146	200	30.49	-10.68	-0.14	-0.02	0.19	7.23
MaxQZ (kN)	1107	372	-0.08	0.85	1.62	-0.06	3.23	0.74
MinQZ (kN)	1107	397	-2.00	0.58	-1.53	-0.15	-3.08	0.19
MaxMX (kNm)	1111	200	15.31	5.16	-0.51	0.62	-1.51	4.30
MinMX (kNm)	1107	208	8.71	3.64	1.00	-0.77	1.91	2.35
MaxMY (kNm)	1107	372	-0.08	0.85	1.62	-0.06	3.23	0.74
MinMY (kNm)	1107	399	-2.06	-0.80	-1.34	-0.06	-3.26	0.18
MaxMZ (kNm)	1143	200	23.27	9.78	0.07	0.03	0.15	8.36
MinMZ (kNm)	1146	200	30.49	0.01	-0.14	-0.02	-0.22	-8.56
Χρήστης			0	0	0	0	0	0

Moving the mouse to the green indicator, we see that the adequacy ratios are very small and that there is room for reducing the cross-section.

So we return to the model and the Basic section. Using the Multiple Options and Filter command you select the Metal Layer elements. Headers.



And from the Member Properties, you choose to specify a smaller cross-section e.g. SHS100X8



In each case you will have to calculate the new intensities and therefore run the analyses again.

⚠ **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.

6.3.3 Buckling control of steel 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

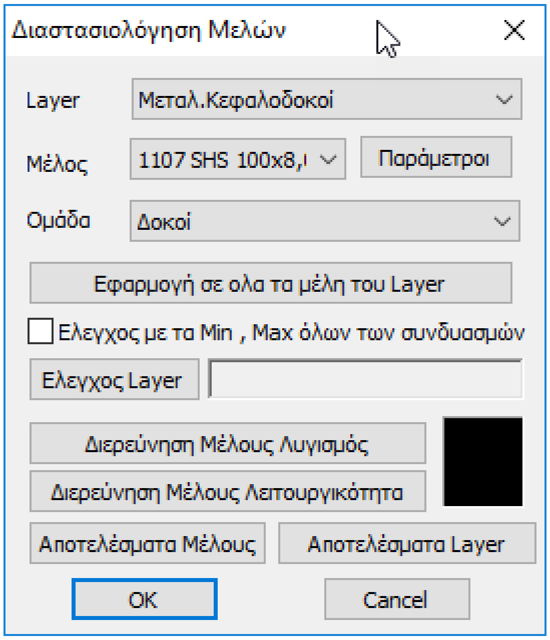
- Check for bending (lateral) buckling due to axial compressive force
- Torsional buckling check due to bending moment.
- Check for torsional bending due to the simultaneous presence of axial compressive force and bending moment.

Limit State of Functionality

- Member deformation control

- Edge (node) movement control

Using the command, the following dialog box appears:



The check is done per layer.

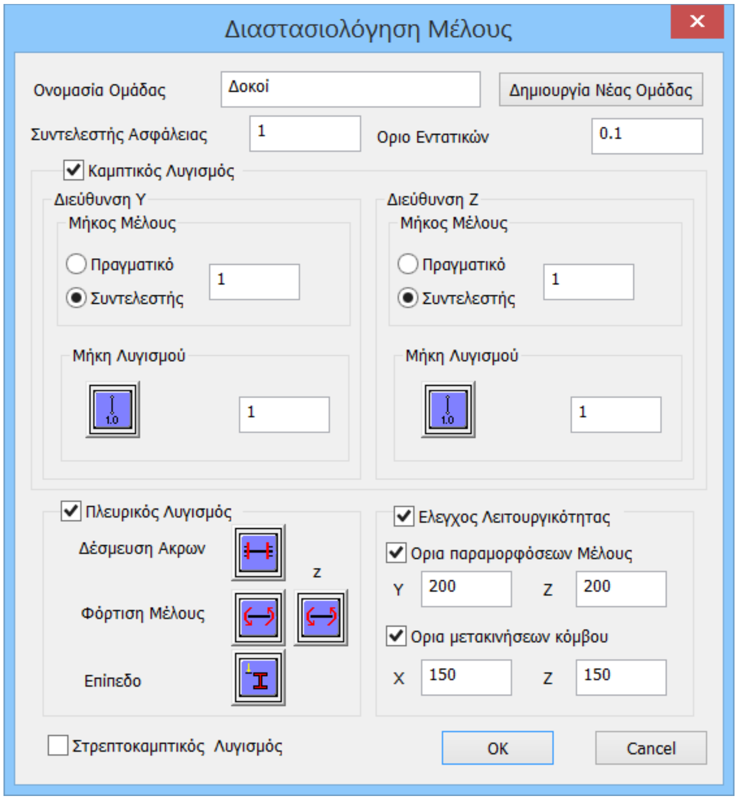
By selecting the layer, all the members of that layer appear in the "Member" list and their cross-section.

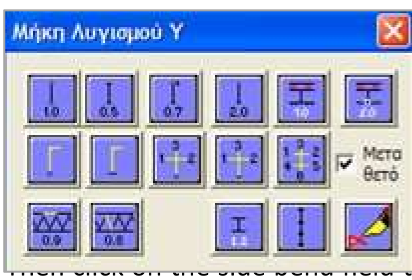
The first step in dimensioning the layer is to define the dimensioning **parameters.** Because it is possible that for some of the members of the layer you may want to define different parameters, it is possible that within the same layer you can define different parameter groups to which the members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

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

Since the "Consolidation of Members" has been done, the definition of Member Length at the 2 addresses is no longer relevant. The program will take into account in the bending length, the length of the member resulting after the Consolidation.

For this example, select the parameters shown below:





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.

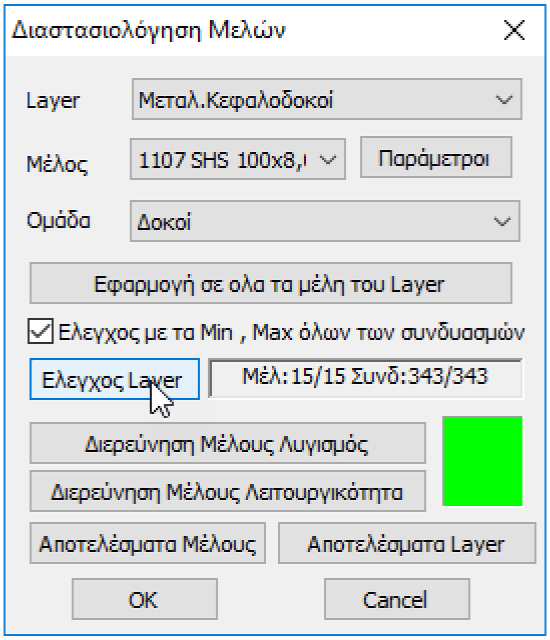
Then click on the side bend field to have the program perform the corresponding check. Here you need to describe the "**Edge Binding**" , the "**member loading**" form by y and z, and the "**loading level**" .

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

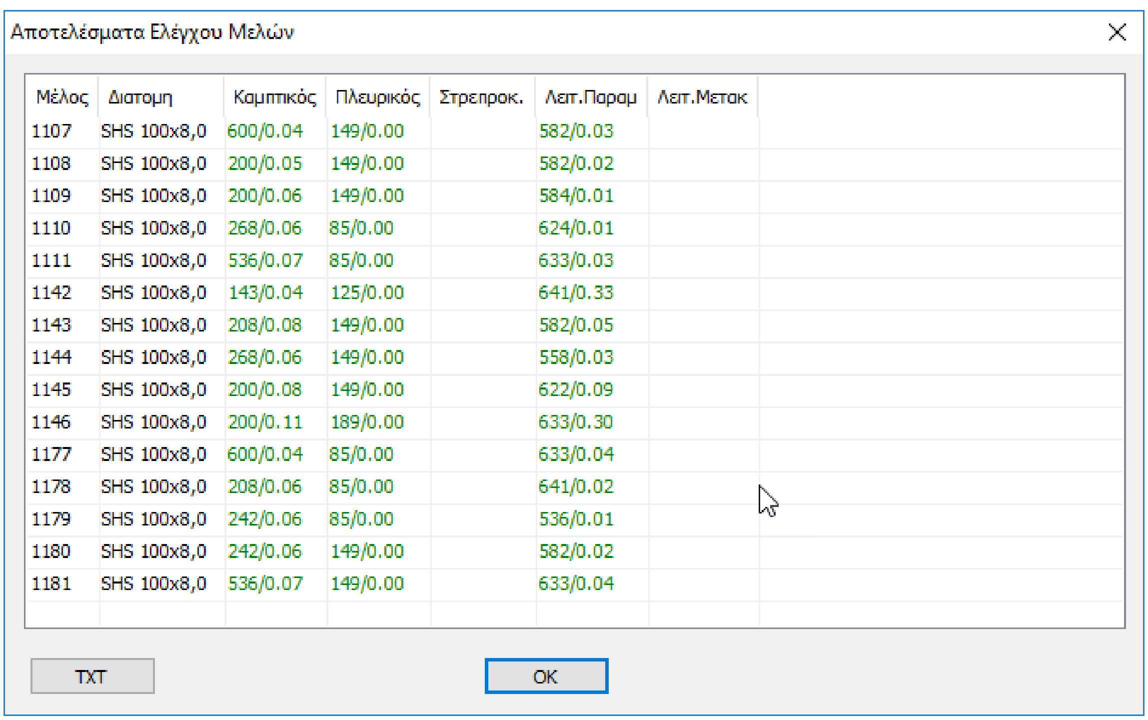
⚠ The parameters for torsional flexure are the same as those you gave for flexural flexure.

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

By activating ☑ Ελεγχος με τα Min , Max όλων των συνδυασμών and selecting the command "**Check layer**" , the calculation of all members starts taking into account only the maximum and minimum of the combinations, which makes the process faster. At the end a green or red square appears which, if you click on it, you will see the ratios resulting from the bending checks of each member.



⚠ The results of the buckling tests can be viewed in a table, as well as summarised or broken down by member and for the whole Layer.


Αποτελέσματα Ελέγχου Μελών

Μέλος	Διατομη	Καμπτικός	Πλευρικός	Στρεπροκ.	Λειτ.Παραμ	Λειτ.Μετακ
1107	SHS 100x8,0	600/0.04	149/0.00		582/0.03	
1108	SHS 100x8,0	200/0.05	149/0.00		582/0.02	
1109	SHS 100x8,0	200/0.06	149/0.00		584/0.01	
1110	SHS 100x8,0	268/0.06	85/0.00		624/0.01	
1111	SHS 100x8,0	536/0.07	85/0.00		633/0.03	
1142	SHS 100x8,0	143/0.04	125/0.00		641/0.33	
1143	SHS 100x8,0	208/0.08	149/0.00		582/0.05	
1144	SHS 100x8,0	268/0.06	149/0.00		558/0.03	
1145	SHS 100x8,0	200/0.08	149/0.00		622/0.09	
1146	SHS 100x8,0	200/0.11	189/0.00		633/0.30	
1177	SHS 100x8,0	600/0.04	85/0.00		633/0.04	
1178	SHS 100x8,0	208/0.06	85/0.00		641/0.02	
1179	SHS 100x8,0	242/0.06	85/0.00		536/0.01	
1180	SHS 100x8,0	242/0.06	149/0.00		582/0.02	
1181	SHS 100x8,0	536/0.07	149/0.00		633/0.04	

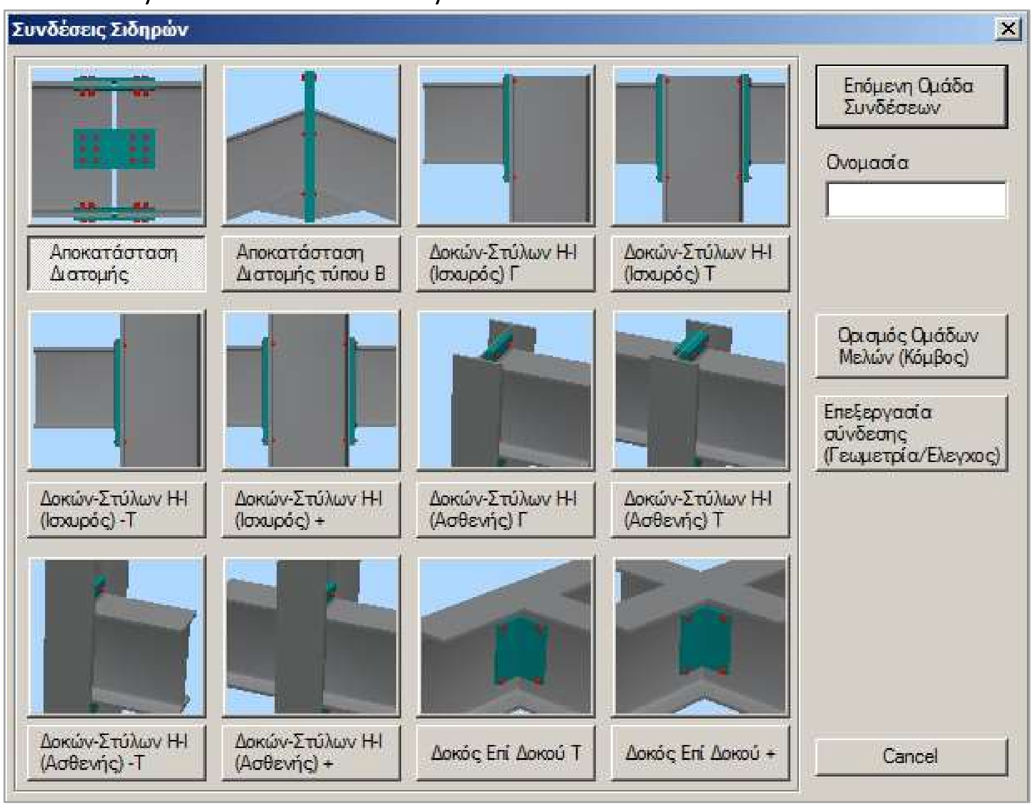
Check the other Layers accordingly.

7. Dimensioning of connections

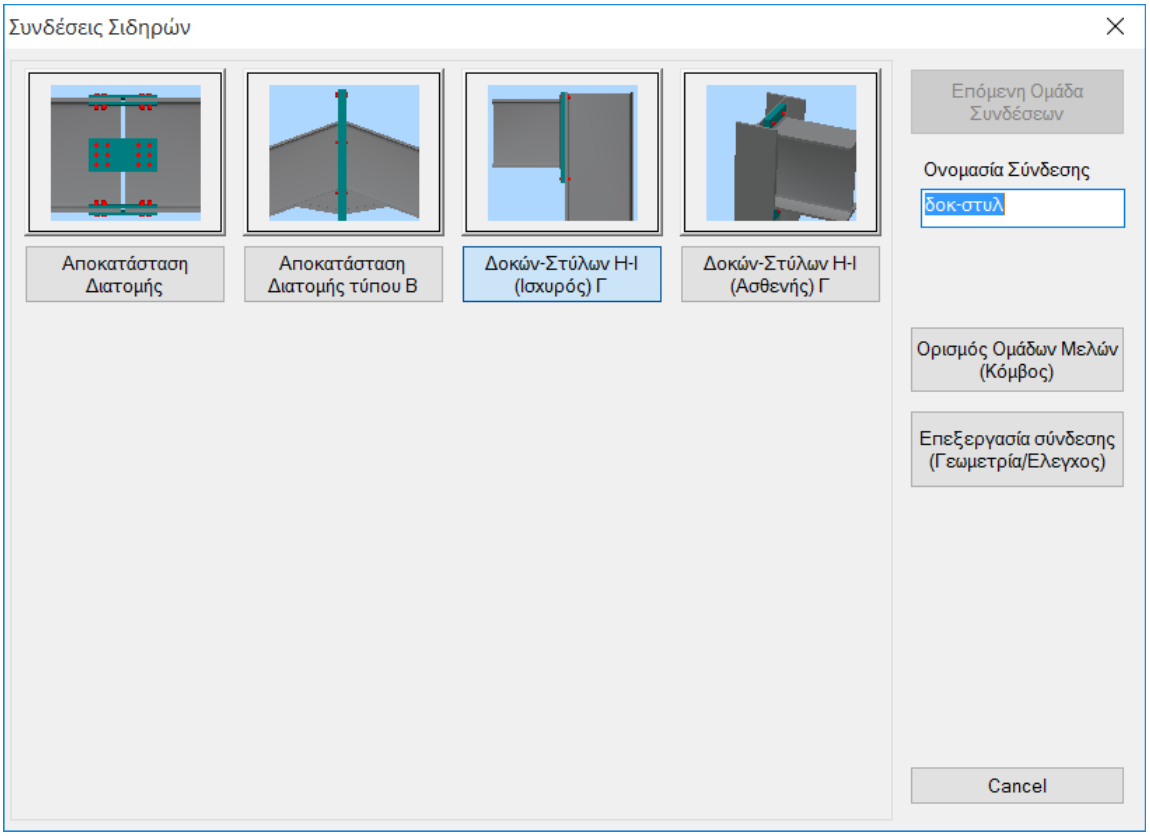
7.1 How to dimension the connections of the metal members:

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 to 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.



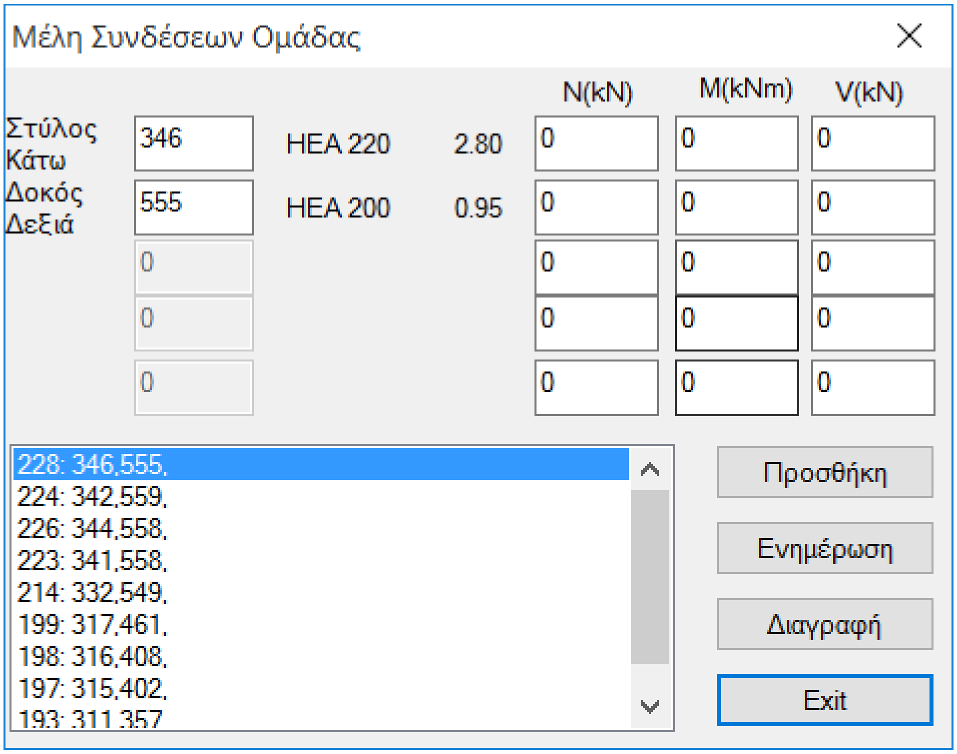
B) Alternatively, you can click on the "Links" command and then left-click on the members you wish to link. Right-clicking then displays a window in which only possible connections consisting of only two members included



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.

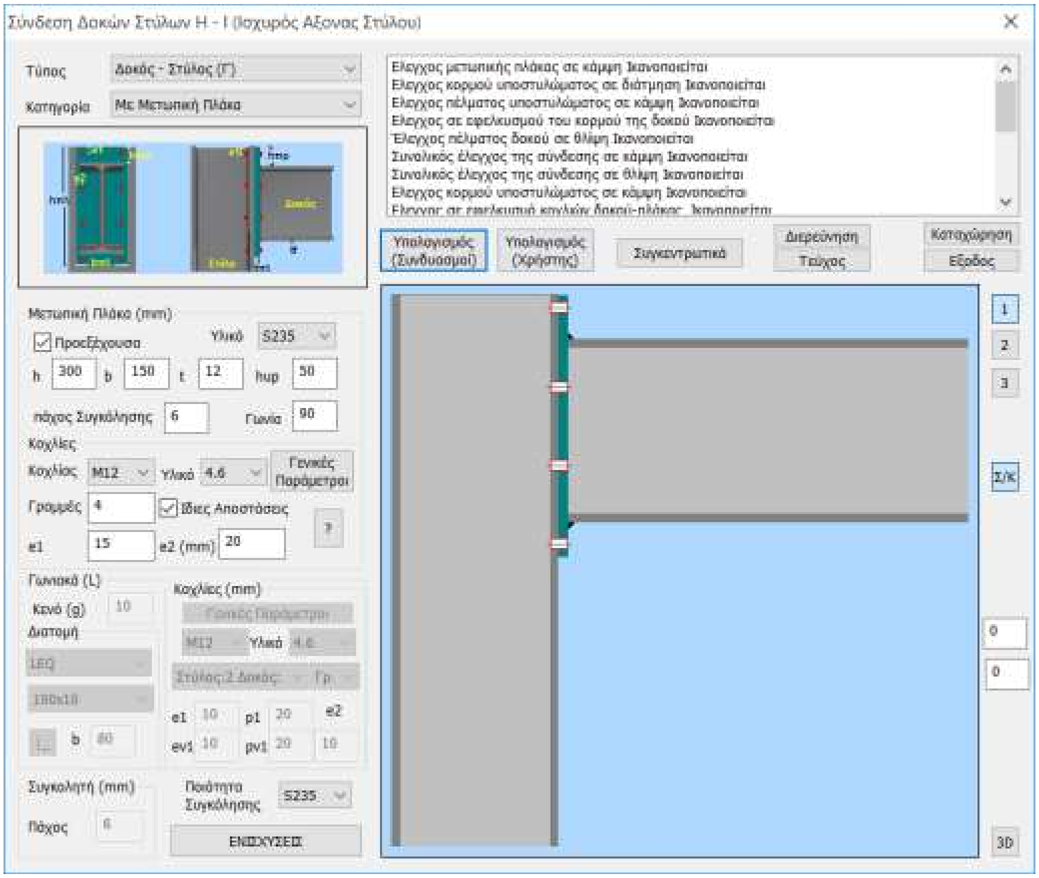
Then select the "Define member groups" command and in the dialog box you can add other similar cross-section pairs (column - beam) or add your own values for the N,M,V intensive sizes to the existing pair. To add other similar pairs, click on the "Column Bottom" field and then select Subcolumn 24 on 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.



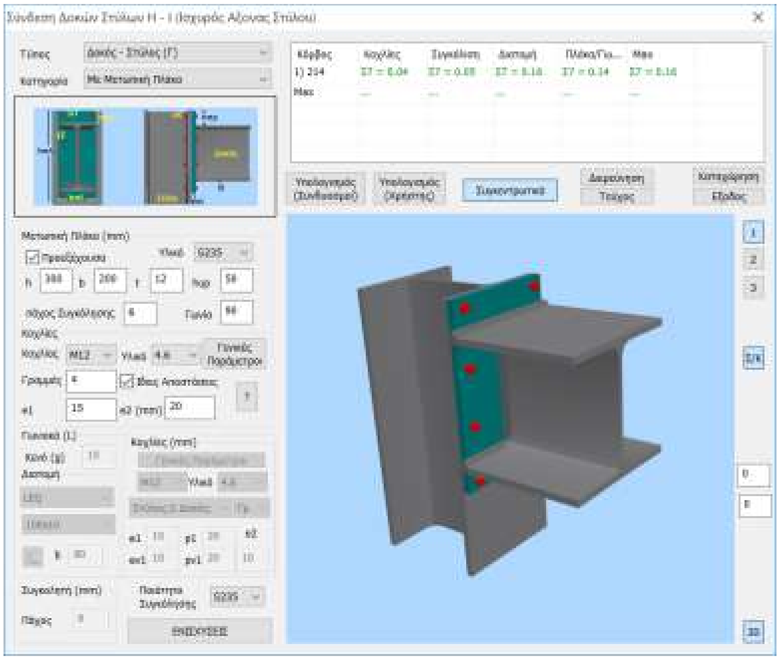
Essentially, way, you can mass dimension all the connections of the beam-post members of the girder that are connected to the weak axis in the same way (bolts or welds, plate geometry, etc.) and that have common cross-sections. The program will automatically calculate the intensive sizes of each pair 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 on 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: 'DESIGN OF METAL CONSTRUCTION'



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 in the 3D view the welds and bolts.



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.

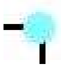
8. FIELD SIZING

8.1 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.

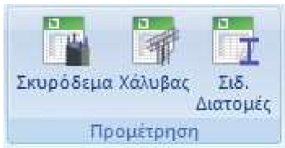
 The skirt missed. The type of failure is also indicated as a symbol above the failure indication. The failure indications are respectively the letter "Z" which means failure at limit load, the letter "e" which means failure due to load eccentricity and the letter "s" which means exceeding of the developed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

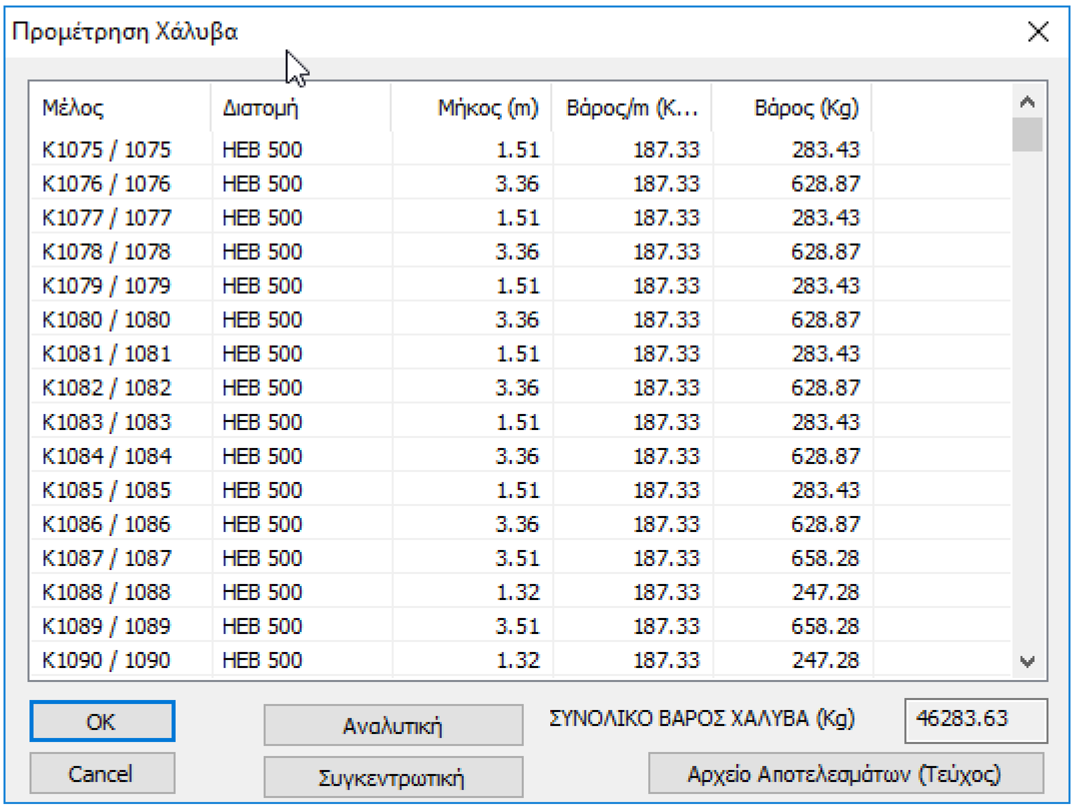
As is well known, seismic intensities derived from dynamic analysis are unlabeled because they result from the superposition of the eigenmodal responses. In the diagrams and wherever there is a necessity to superimpose them, they are always used with positive values. And for the dimensioning of the elements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

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.

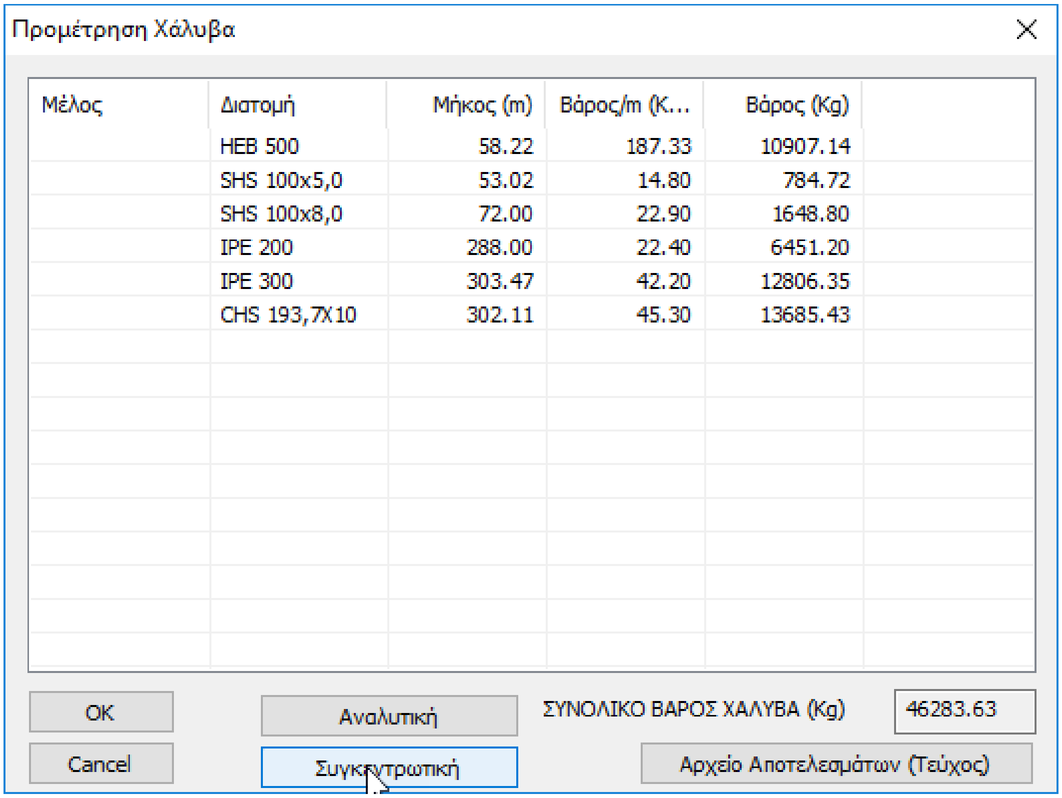


Μέλος	Διατομή	Μήκος (m)	Βάρος/m (K...	Βάρος (Kg)
K1075 / 1075	HEB 500	1.51	187.33	283.43
K1076 / 1076	HEB 500	3.36	187.33	628.87
K1077 / 1077	HEB 500	1.51	187.33	283.43
K1078 / 1078	HEB 500	3.36	187.33	628.87
K1079 / 1079	HEB 500	1.51	187.33	283.43
K1080 / 1080	HEB 500	3.36	187.33	628.87
K1081 / 1081	HEB 500	1.51	187.33	283.43
K1082 / 1082	HEB 500	3.36	187.33	628.87
K1083 / 1083	HEB 500	1.51	187.33	283.43
K1084 / 1084	HEB 500	3.36	187.33	628.87
K1085 / 1085	HEB 500	1.51	187.33	283.43
K1086 / 1086	HEB 500	3.36	187.33	628.87
K1087 / 1087	HEB 500	3.51	187.33	658.28
K1088 / 1088	HEB 500	1.32	187.33	247.28
K1089 / 1089	HEB 500	3.51	187.33	658.28
K1090 / 1090	HEB 500	1.32	187.33	247.28

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".

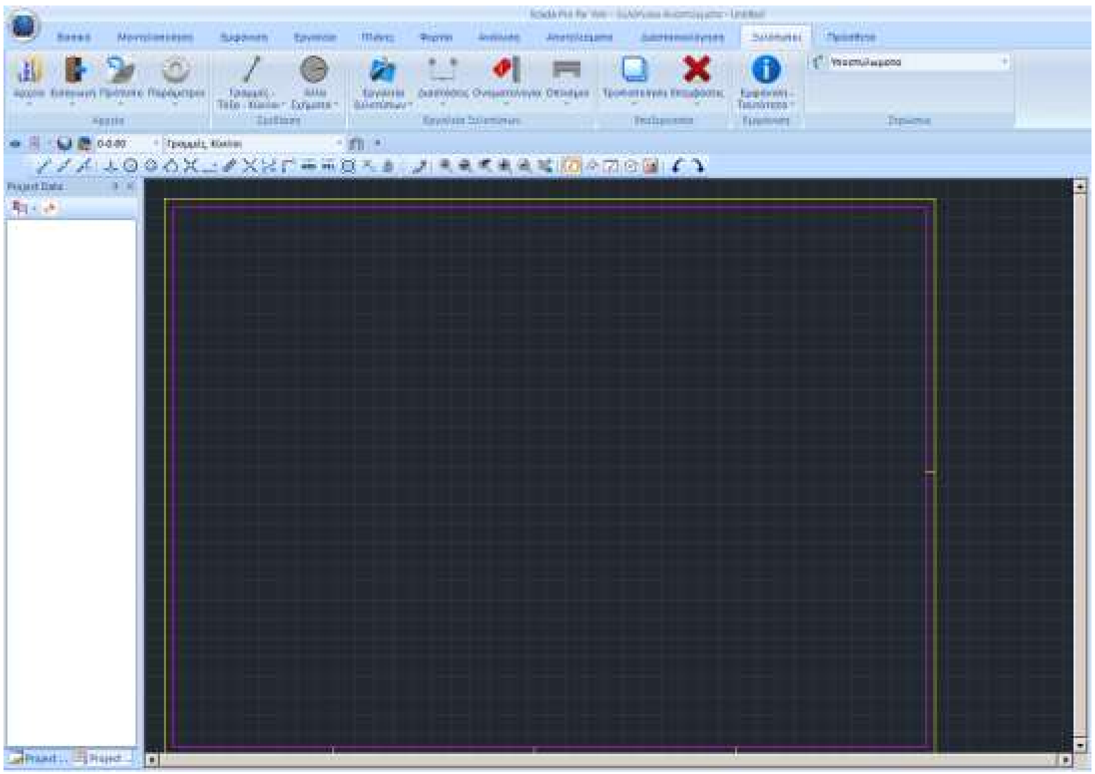
EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'



10. DESIGN

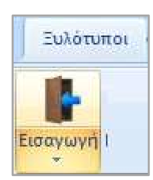
After completing the dimensioning of the carrier and the creation of the connections for the metallic ones, in the Timber Formwork Module you enter, modify and finally create the drawings of the formwork and its details.

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

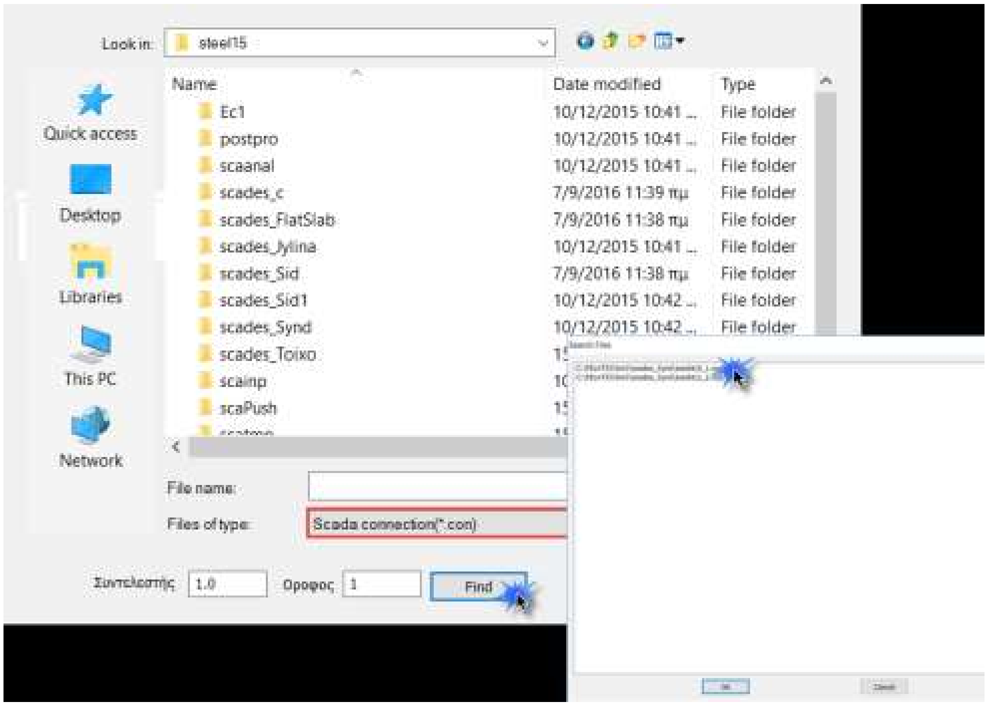


10.1 How to enter the plans of the links:

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: 

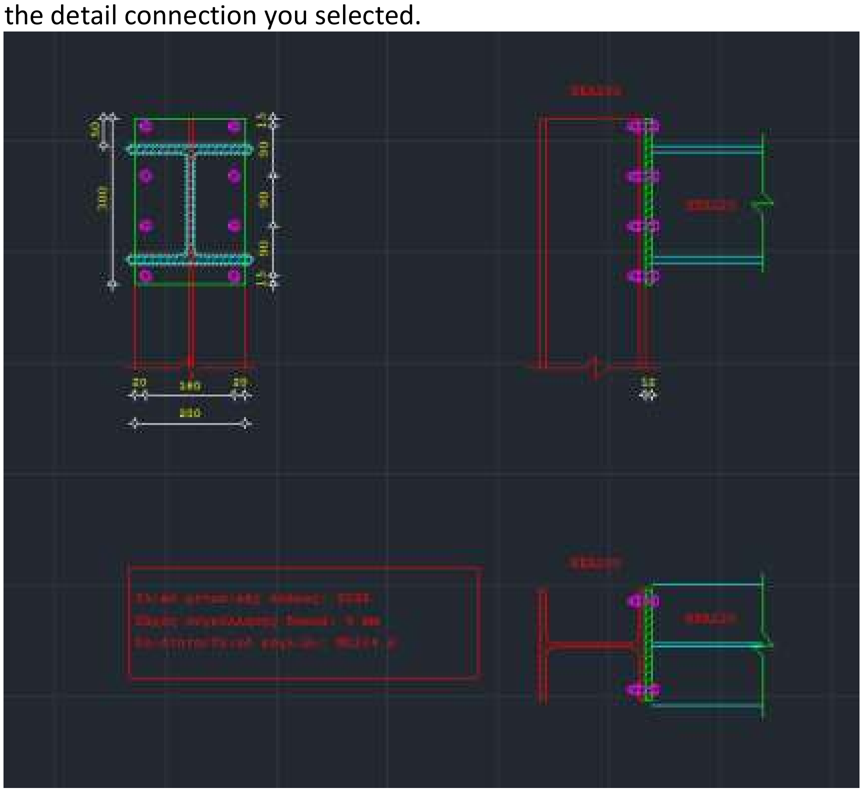
EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'



In the dialog box:

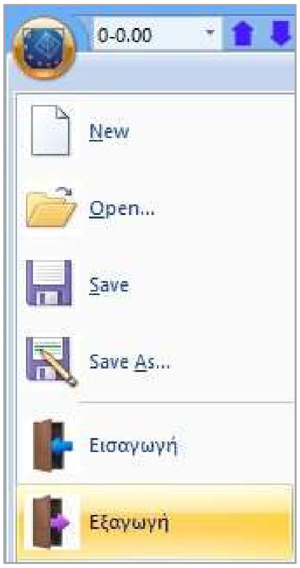
- in Files of Type select **Scada Connection(*.con)**

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 detail connection you selected.

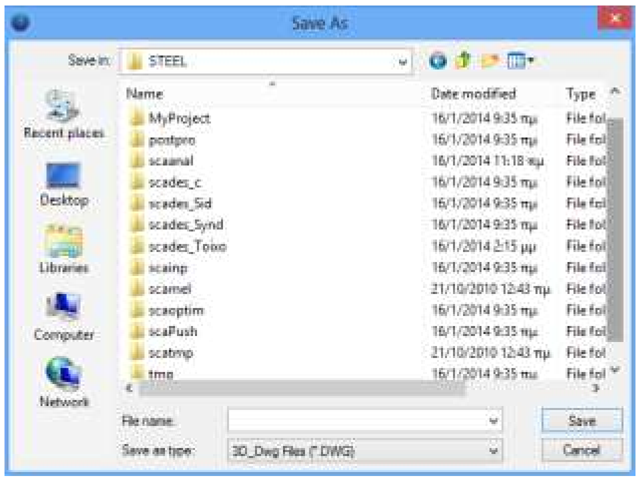


EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'

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 have 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.



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, even visualize your carrier in 3D and with photorealism.

11. COPY

11.1 How to create the study issue:

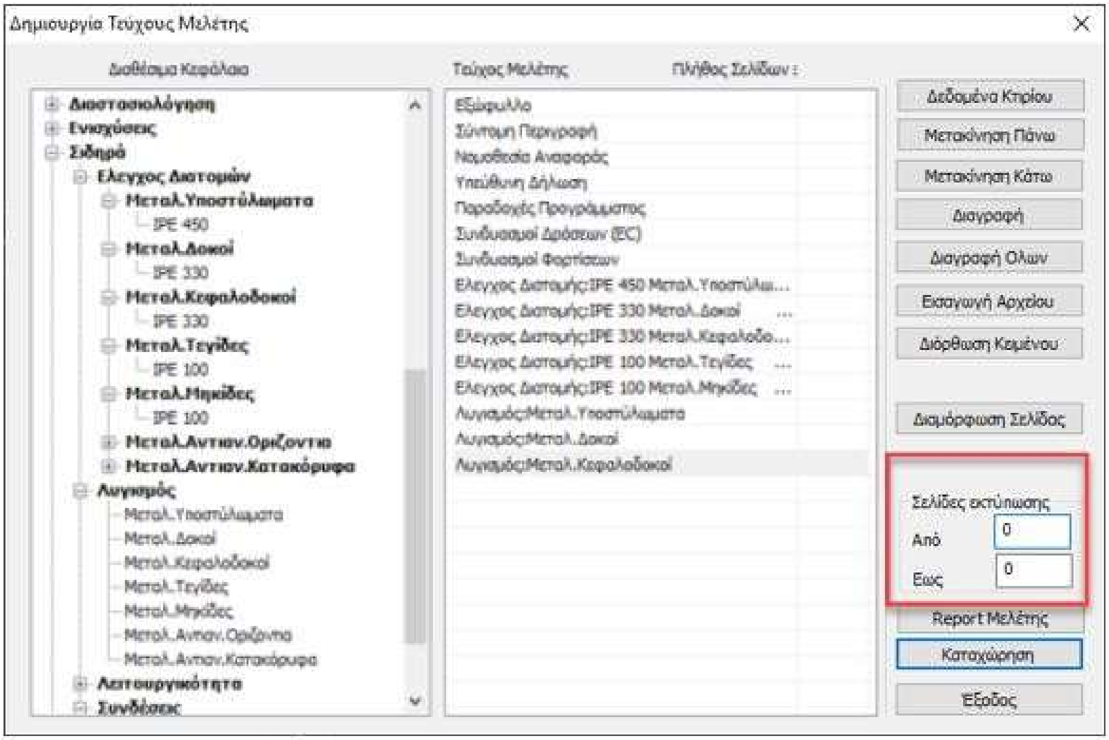
To create the study booklet, open the "Extras" section and select the Print command.
In the "Create Study Sheet" dialog box, the list of chapters available for printing appears on the left. The right list, with the chapters to be included in the booklet, is completed by selecting them from the left list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

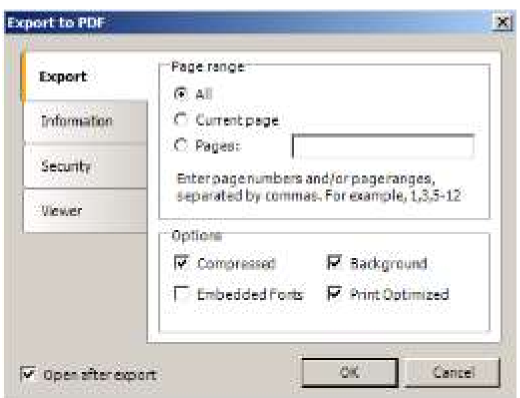
In the new version of SCADA Pro all the printouts of the study results booklet have been redesigned and implemented with modern tools in order to offer you a new tabular, easy-to-read study booklet with the addition of diagrams and images. You also now have a full preview of your issue as well as the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

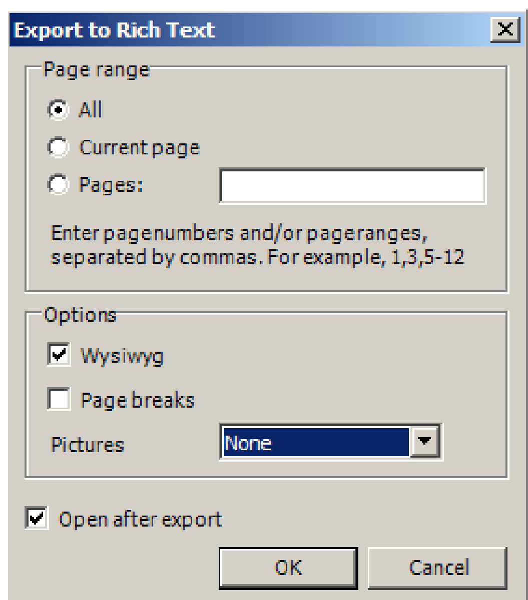
In addition, the ability "break" the study book into individual sections has been added, a useful and practical feature especially for the easy management of multi-page studies.

EXAMPLE: 'DESIGN OF METAL CONSTRUCTION'



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





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