SCADA Pro™
Structural Analysis & Design

Example 2

Formal Study

Metal Construction

ACE HELLAS
INTEGRATED SOLUTIONS

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

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 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

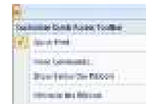
This manual was created to guide the designer in his first steps in the new SCADA Pro environment. It is divided into chapters and based on a simple example guide.
Each chapter contains information useful for understanding both the commands of the program and the procedure to be followed in order to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

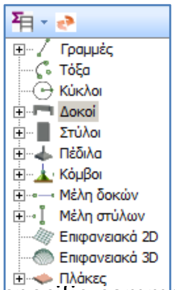
THE NEW ENVIRONMENT

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

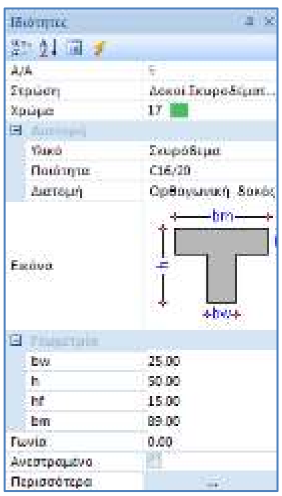
EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

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





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. It is also possible to apply specific commands to each element of the selected tree. The command menu is displayed with the right mouse button, and this menu changes depending on the section of the program that is active.



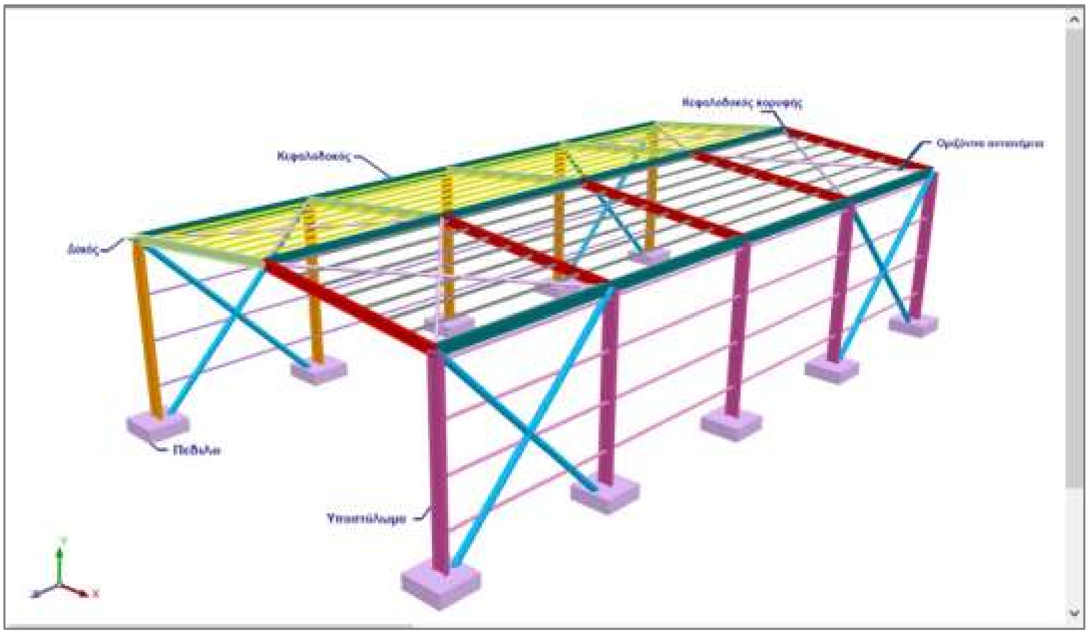
The "Properties" list on the right is automatically populated by selecting an element of the entity. It informs the user of its attributes, as well as allowing changes to them.

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

1. GENERAL DESCRIPTION

1.1 Geometry

The metal shed under study consists of five frames of one span (7.05+7.05=14.10m). The frames are spaced 6.80m apart. The roof is pitched with a pitch of $5.33^{(o)}$. The ridge is at a height of 7.50m. The height of the poles is 6.50m. The frame structure is formed of steel frame, while the foundation is formed of individual reinforced concrete footings in both directions. For the complete geometry see the figure below:



1.2 Materials

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

1.3 Regulations

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

1.4 Cross sections

Pillars: IPE450
Crunch: IPE360
Vertical Windward: CHS219.1/6.3
Horizontal Windward: CHS114.3/5.0
Headstocks: HEA180 Top
Headers: HEA180
Tigers: IPE100
Mecids: IPE100

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 ± (epicentric torsional moment loads resulting from the epicentric forces of the earthquake XI displaced by the random eccentricity ±2eτzi).
(6) Erz±(epicyclic torsional moment loads resulting from the epicyclic forces of the earthquake ZLI displaced by the random eccentricity ±2eτxi.
(7) EY (vertical seismic component -earthquake by y- from dynamic analysis). To these, for this example, we will add the 3 below:
(8) S (snow)
(9) W0 (wind in direction x)
(10) W90 (wind in the y 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 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 **User Manual** that accompanies the program.

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

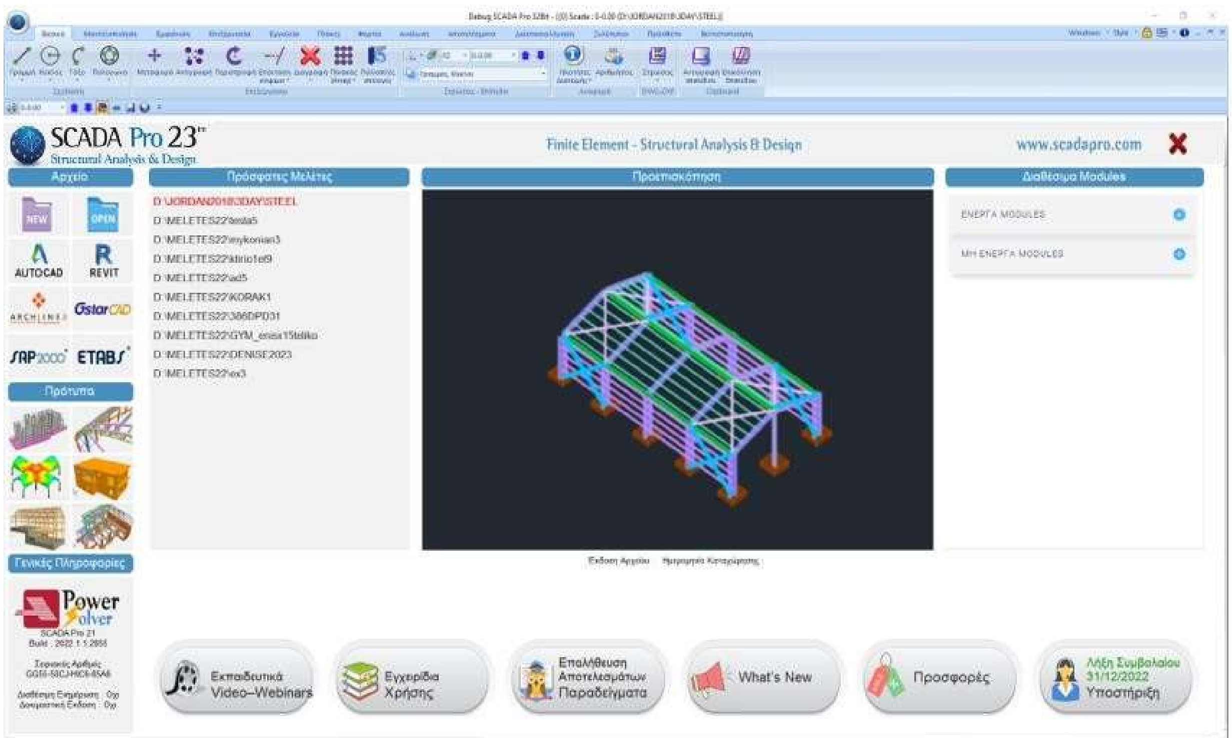
2. DATA IMPORT - MODELLING

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 conjunction with the architect, the shape of the floor plan, the choice of using linear and/or
finite elements, etc.

> ⚠ This example will detail how to use standard structures to model a metallic structure.

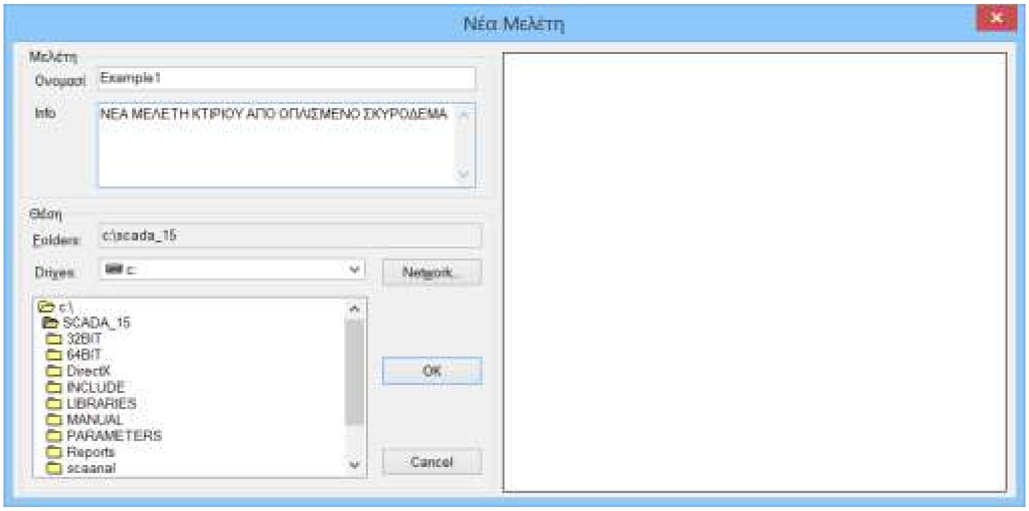
Upon opening the program, the startup window appears on the screen, which includes a set of commands to start the program:



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's commands.

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

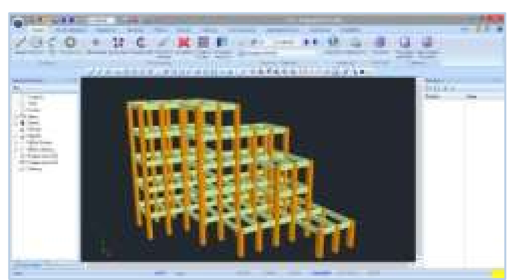


The file name must consist of ***a maximum of 8 Latin characters and/or numbers, without spaces and without the use of special characters (/, -, _)*** *(e.g. ARXEIO1). The program automatically creates a folder where it enters all your study data. The "Location" of the folder, i.e. the place where this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

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

"**new**": usually used when there is no auxiliary file in electronic form. The startup is done in a blank interface. The designer starts by defining the stations and importing the cross-sections, using the
modeling commands and with the help of the canavan's pulls.

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

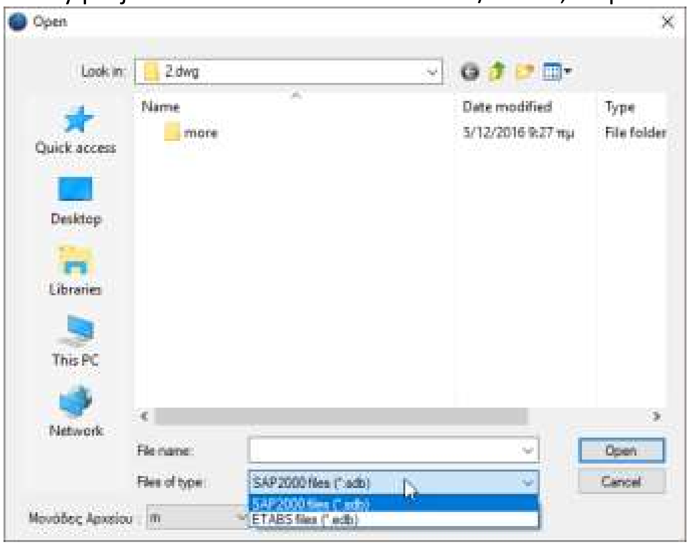



"**ArchlineXP**": read xml files from ArchlineXP.

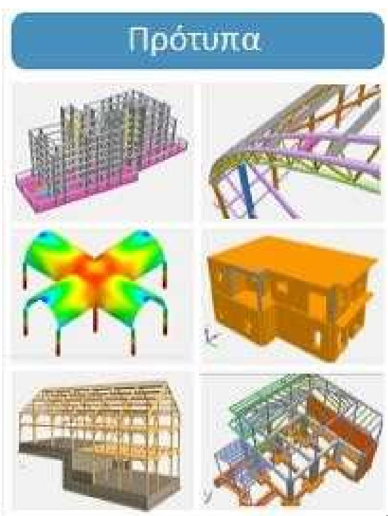
"**ETABS, SAP2000**": reading .edb & .edb files .sdb from the static programs ETABS & SAP2000 .

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

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



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



concrete metal

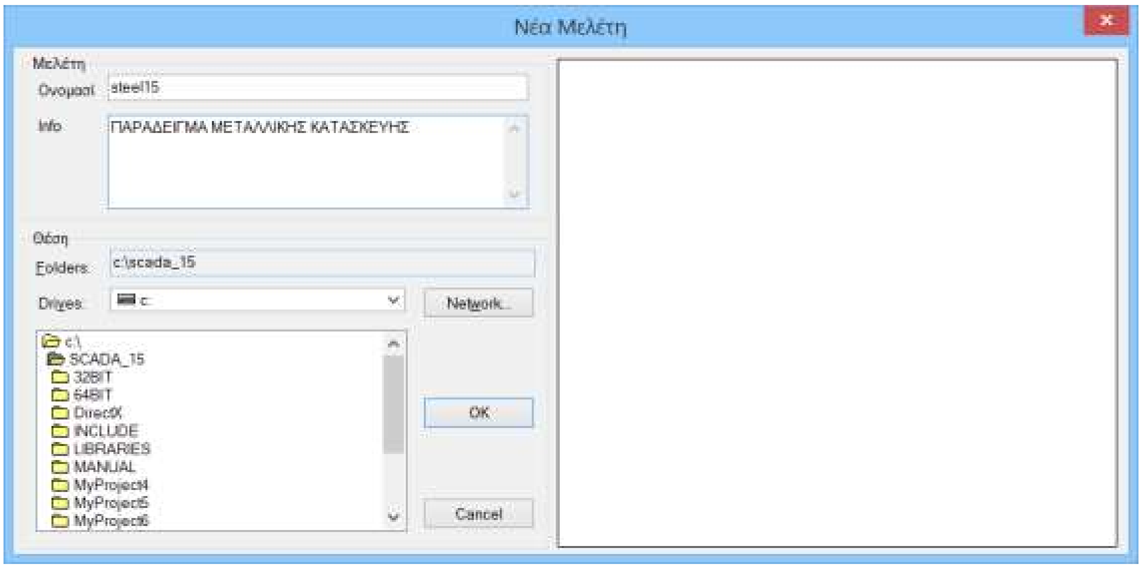
surface connections(IDEA StatiCa)

wooden masonry

- The usual metal carriers are typical structures with continuous frames in one or both directions with a pitched roof. They may include thimbles and purlins,
  windbreakers and frontal columns. In cases using the standard structures you manage to model the carrier with a single movement! But also in the case of more complex structures, the use of standard structures can provide the foundation which to build a complex structure.

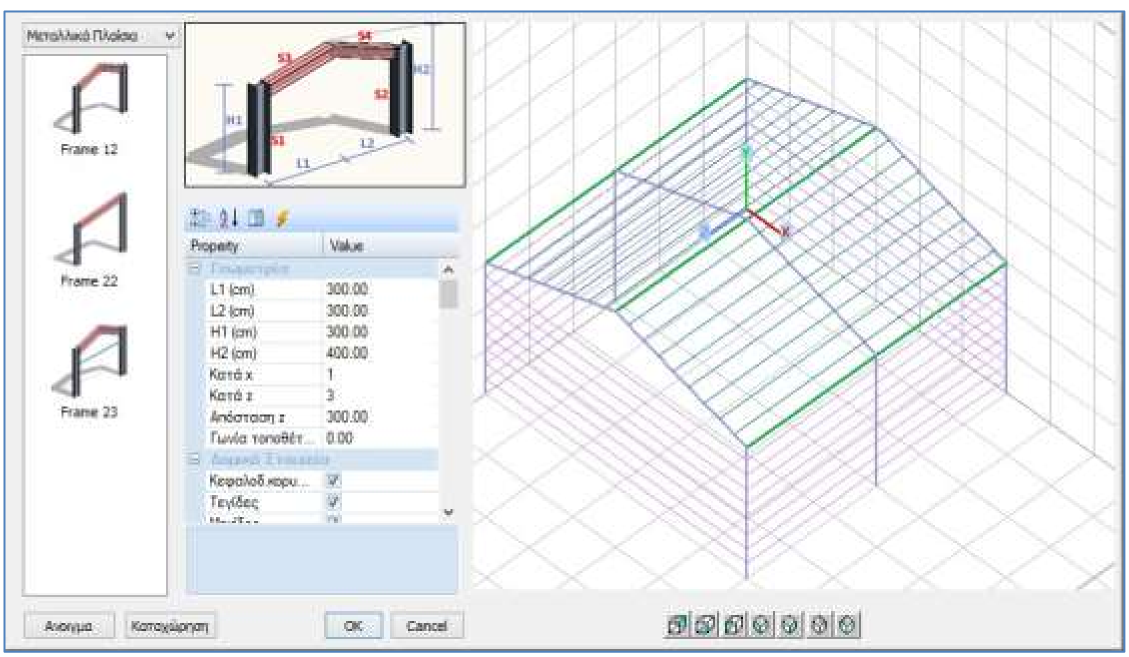
2.2 Standard Structures - Metal Frames

Select the relevant icon  and in the dialog box



give a "**Name**" to the study. If you wish, enter some information about the study in the "**Info**" field and OK.

Automatically, a new window opens through which you can define, by going from top to bottom, the geometry, the cross-sections and the foundation of the carrier in a single step.

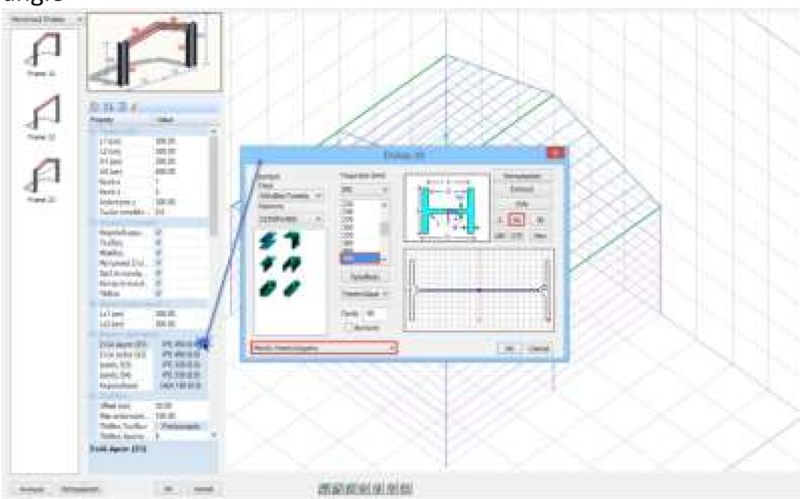


EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

Going down and following the data of the study, determine the distances between the main panels = 680 cm.

OBSERVATION:
Here it is worth noting that you can choose different distances between the frames (Lz1, Lz2 etc.).

As the main cross-sections of the frames you choose :
Pillars S1-S2: IPE450 with 90° angle
Beams S3-S4: IPE360 with 90° angle
Header beams: HEA180 with 90° angle



OBSERVATIONS:

- *To modify a predefined cross-section, left-click on it and the corresponding dialog box opens. Select the desired cross-section and its angle. In addition, it is necessary (for dimensioning, see below) to select from the list at the bottom of the window, the layer to which it will belong.*
- *In addition to the Layers predefined by the program, the user has the ability to create his own layers and transfer to them the elements of his choice.*
- *It is worth mentioning that the program has the ability to automatically calculate the angle that the frames should have by calculating the angle of inclination of the frames. So even though we put an angle of 90° on the tetrahedrons the program calculates their exact angle and places them in the 3d correctly oriented. This is only valid for standard constructions. In every other case the members are inserted with the orientation you give.*

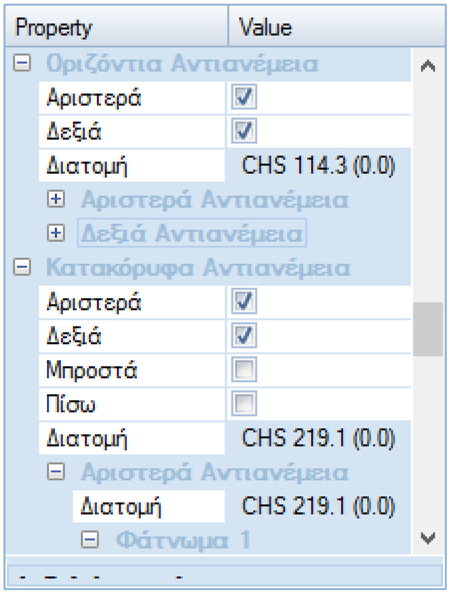
Then you select the number, the positions and the cross-sections of the **pegs** and **pins**. You can enter the maximum distance between consecutive purlins or thimbles and the program will automatically calculate how many purlins or thimbles fit in the beam or post. The offset indicates the distance that the first purlin or cotter pin will be from the header.

In this particular example:

- select 30 cm as offset and type

- as a number of 8 (IPE 100 with 90° angle-left and right) and
- as a number of 4 (IPE 100 with 0° angle-left and right).

Then you can insert the frontal poles (in this example there are no frontal poles) and the **headwinds**.

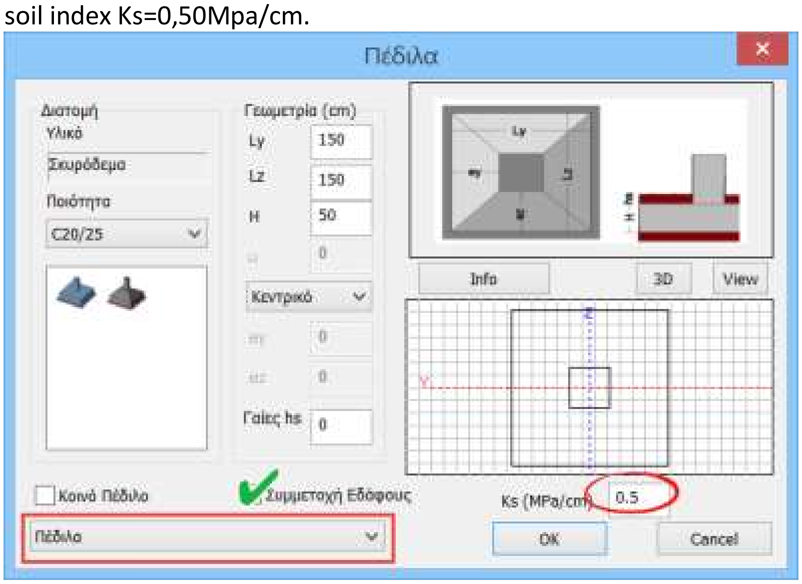


For the **horizontal wind resistors** you choose cross-section CHS114.3/5.0 and place them on the first and last facade.
In the places where these will be inserted you have 4 options. You choose the "<u>upper and lower poles</u>".

Thus, the windbreaks start from pole nodes and end at nodes of the ridge, running through the intermediate tectiforms without being connected. Also, if you don't select the "intersection" the antifans will not split in the middle resulting in 2 cross members. Otherwise we would have 4 which would connect in the middle to form a single node.

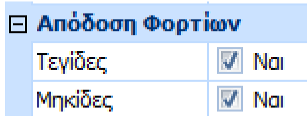
For the **vertical upwinds** on the left and right (not front and back for the example) you select cross sections CHS219.1/6.3 and positions "upper and lower posts".

Finally, select the cross-sections of the **pedestals** (150cmx150cm rectangular cross-section pedestals) and the soil index Ks=0,50Mpa/cm.



EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

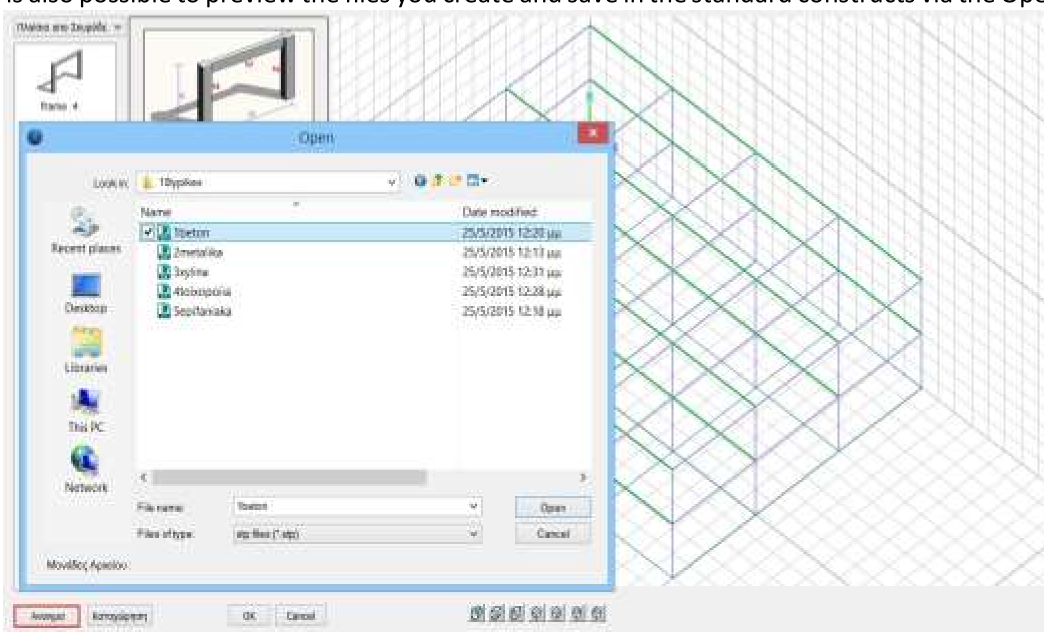
⚠ With the "Tags" and "Mikes" enabled in the "Load Performance" field of the "Standard Construction"



, program automatically calculates the zones of influence by distributing the pressures to all the tectonic plates and tectonic plates (see: Wind-Snow Loads>Member Matching).

⚠ By selecting "entry" you can save the construction you have created in a folder and in this way create your own library with ready-made standard frames which you can use again for any other study.

⚠ It is also possible to preview the files you create and save in the standard constructs via the Open command:



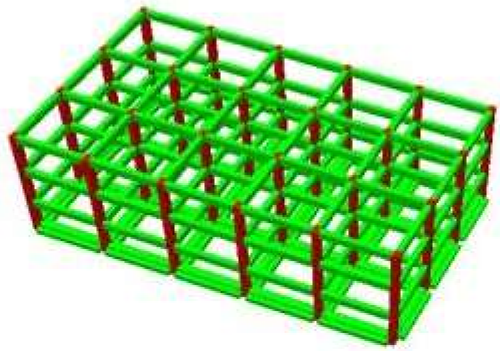
⚠ By pressing "ok" you can see the carrier in 3 dimensions and even with the physical cross-sections.

⚠ *You can use as many ready-made standard frames as you want in one file to create more complex constructions. Just select the frame you have entered with the open command and then click on a point in the canvas or a node by pressing "ok", if another construction has already been entered.*

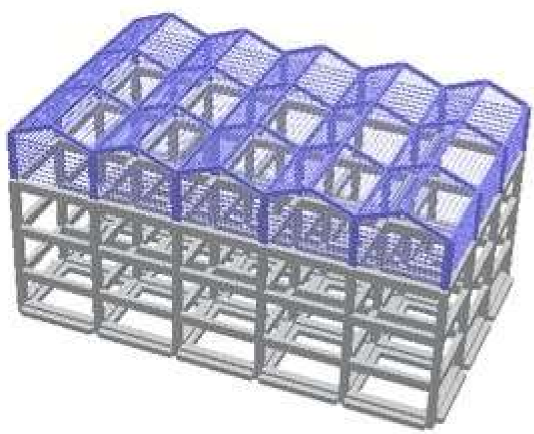
⚠ *In this way you have the possibility to insert as a background any concrete structure and then insert as a superstructure a ready-made standard steel frame, selecting as an insertion point a node belonging to the concrete carrier.*

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

For example, if you insert a metal structure on top of a concrete structure (which you have already inserted in the design environment) and click on the top left node of the latter, then the steel structure will be placed



as you can see in the figure below and indeed where there are common nodes of concrete and steel columns these will be automatically identified by the program.



OBSERVATION:

The above feature is a very useful tool especially for composite structures, since the time required for the description of the structure is reduced to a minimum, thus increasing the productivity of the designer.

2.3 How to modify a standard construction

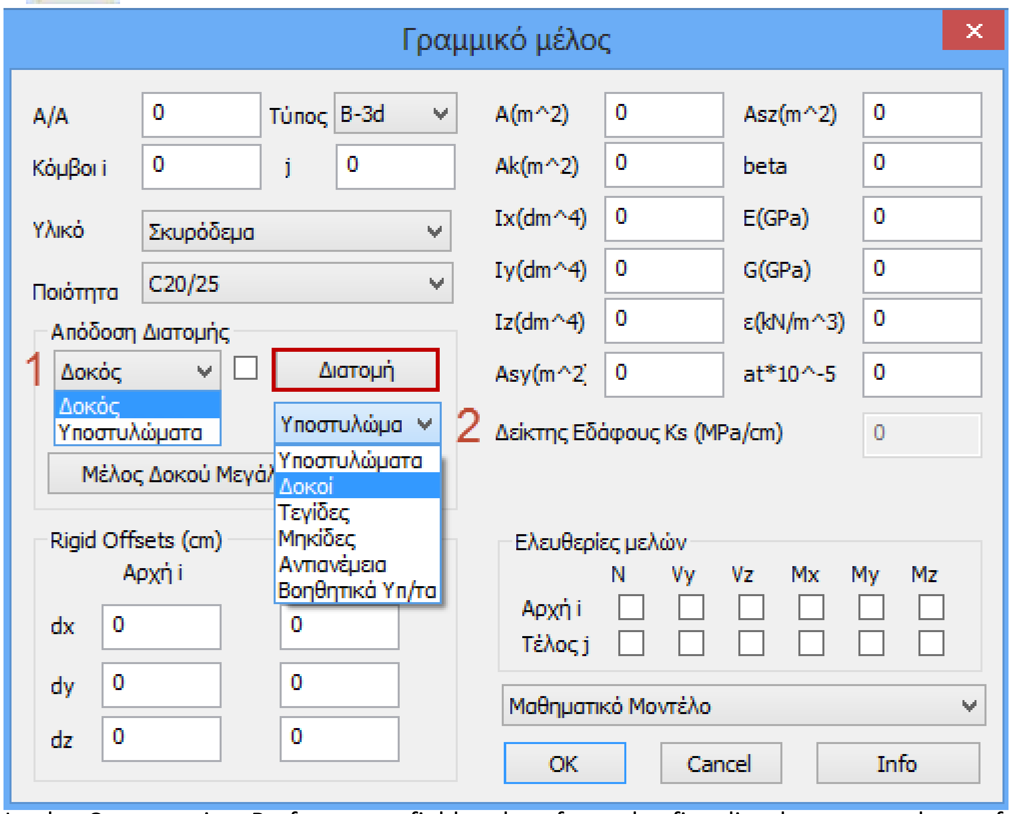
After importing the standard construction, you can make possible modifications or additions to create even the most complex constructions.

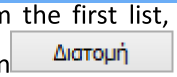
OBSERVATION:

⚠ *Note that, the members of the bodies created through standard constructions are* ***mathematical members with cross-sectional performance*** *and* *<u>not physical members</u>.*

You can delete members and nodes with the "Delete" command in the "Basic" section

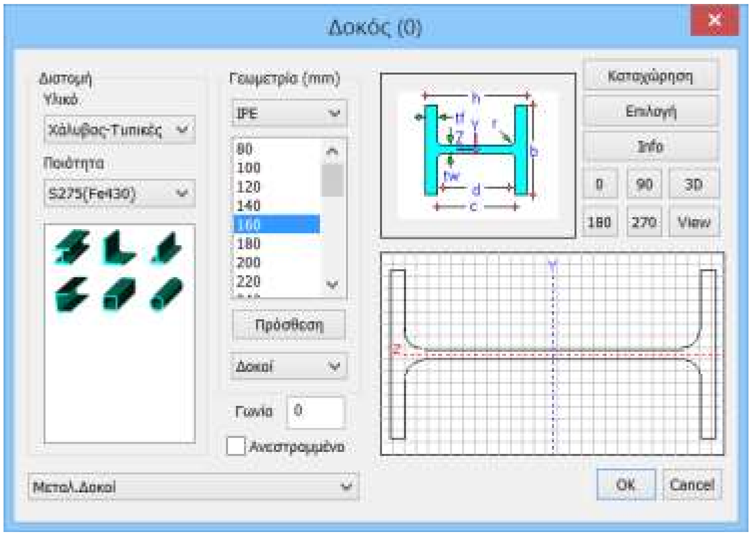
You can add new members with the "Member" command in the "Modelling" section. Selecting the "Member" command opens a window where you can select the cross-section, type and even the freedoms a member will have:



In the Cross-section Performance field, select from the first list, beam or column, from second list the compared type of cross-section and press the button 

. In the new window that opens, set the characteristics of the cross-section.

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'



To insert it into your construction you click on "ok" and then select the start and end node of that member on your 2D or 3D construction. This gives you the ability insert any member into the space and create even the most complex constructions.

3. IMPORTATION OF GOODS

3.1 How to import loads in the general way of importing loads :

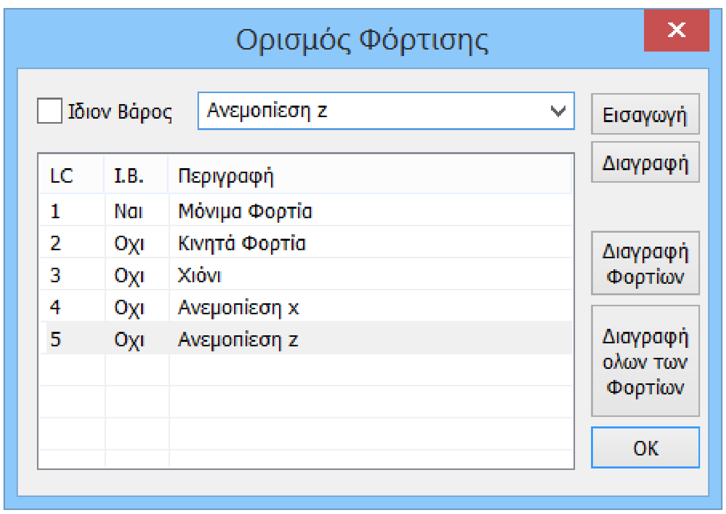
⚠ *Below, for educational purposes, the methodology for importing loads is listed with the general way of importing loads in the program.*

⚠ *The study will eventually use the new tool of automatic input of wind and snow loads according to EC1. Therefore, you can skip the following and go directly to page 23.*

In the "Loads" section and in the "Loads" command you first define the type of loads of the carrier. Automatically a window opens, in which you can enter additional loadings from a ready-made library.

As default loads there are permanent (which includes the same weight with the possibility to exclude or include it in another load) and mobile.

For this example you should click on the third line and enter the snow from the list (Load case 3). Similarly on the fourth line and enter the wind by x (Load case 4) and respectively by z (Load case 5) as shown below:



After defining the load cases to be included in the structure, you proceed to enter specific load values to the members of the structure.

For the purposes of this study you will provide:
- **Permanent loads** :0,20 KN/m on all the tesserae
- **Mobile loads** :0,10 KN/m on all tesserae
- **Snow load** :0,15 kN/m on all troughs

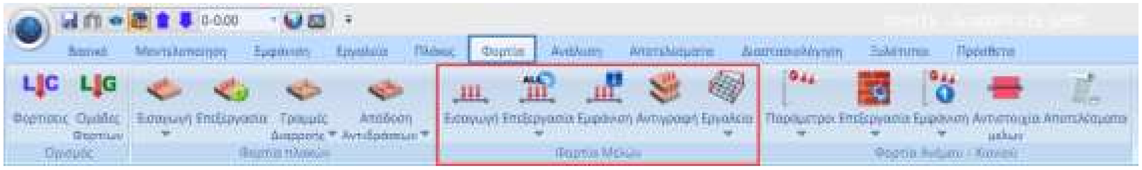
**alternatively you could put snow as a load on the beams*

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

- **Wind load at x** : 8KN/m on light red coloured columns and 2 KN/m on dark red coloured columns.
- **Wind load at z** : 3KN/m in all columns.

**Alternatively you could put the wind as a load on the thwarts*.

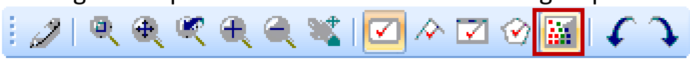
In detail :

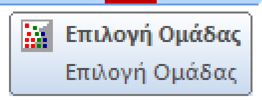


Within the "Loads" section and the "Member Loads" command group, select the "Import" command. and then left-click on the members you want to give loads to.

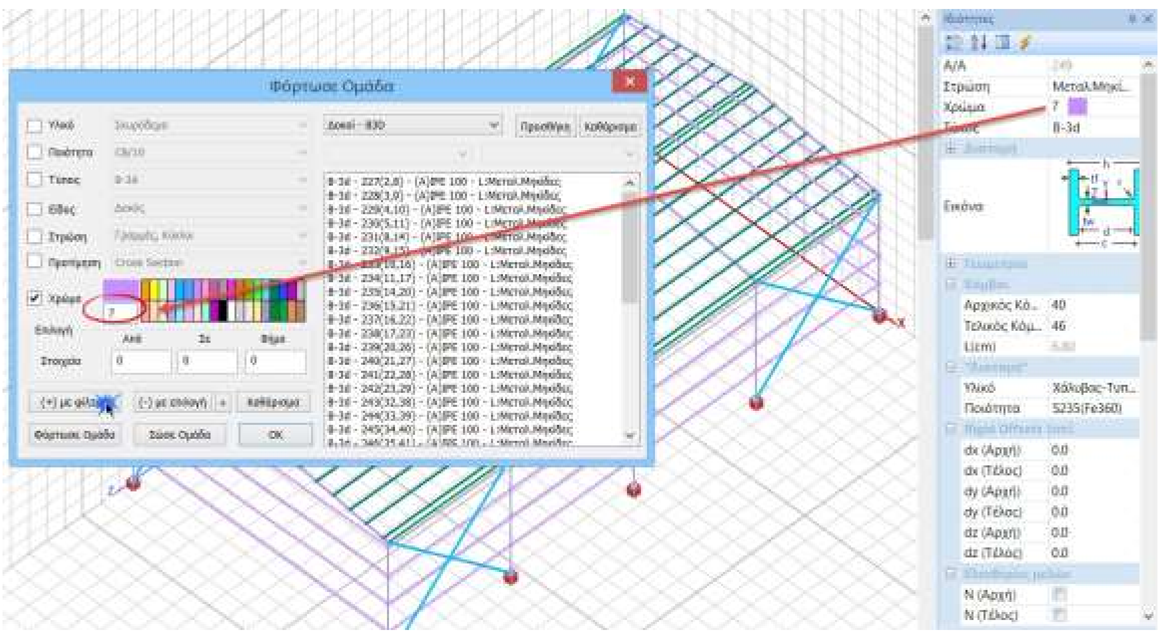
You can select them either one by one, or collectively through filters. Here, we will use method 2[h]. So, after clicking on "Import " 

then click on the "group selection" button



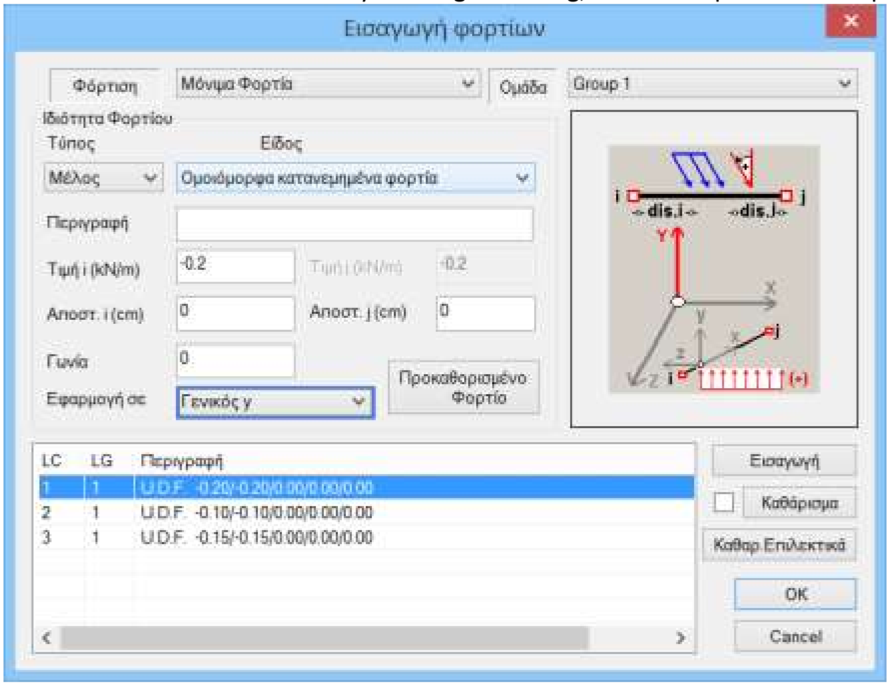


which opens a new window:



The "Group Selection" is a very useful tool which allows you to select elements of the construction based on specific criteria which are displayed on the left of the above window. In this example, colour filter will be used next, i.e. the selection of elements to impose loads based on their colour.

So in this window check the color filter and select color or write its corresponding number, which appears in the properties by selecting a tick, in this case the color number 7. Then click on the "**(+) with filter**" command. Automatically in the right area of the window all the roof's tectrics are added to the list, i.e. those with colour 7, and then you click **ok.** You will notice that the members of the τεγίδες present in the vector are shown with a dotted line. By then right clicking, the load input window opens.



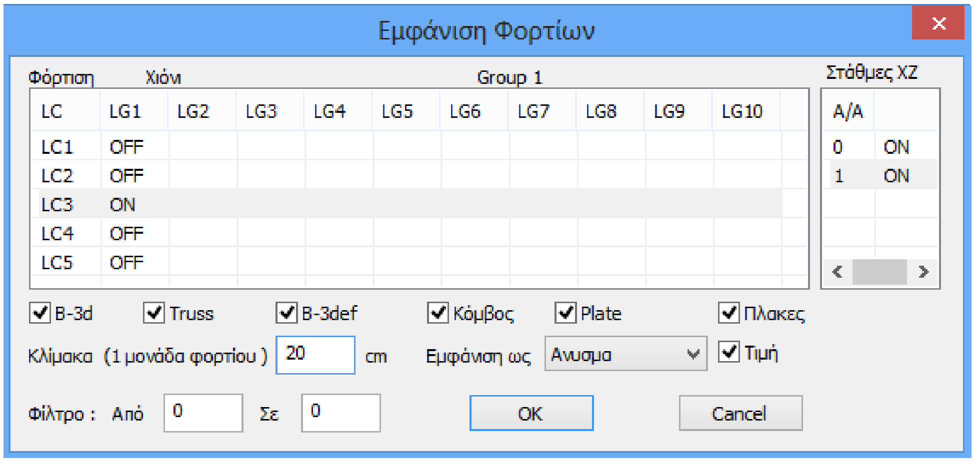
In the value field type -0.2 and in the application select the General y-axis.
You put a sign - because the permanents are applied in the downward direction, i.e. in the opposite direction to the universal y-axis. (You could of course also enter them in the local system of members with the appropriate sign). You then select the insert button to fill in the first row of the LC 1 loadings.
In the same way, click on the second line, select the mobile loads at the top of the window, type -0.10 and press enter.
Finally, repeat the same for the snow load. The final result that will be displayed is what you see in the picture above.
By selecting ok, the loads you have specified will be applied to the corresponding members you had selected.

You also have the possibility to view the loads in space through the "Showcommand 

After selecting the command, a new window opens through which you select the load and level for which you want to view the applied loads on the 3D vector.

For example, turn on the snow - LC3 ON, levels 0,1, change the scale of the load to 1:200cm and in the display select the bloom.

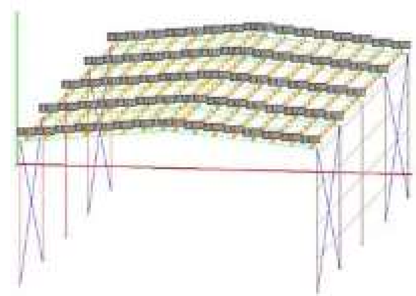
EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

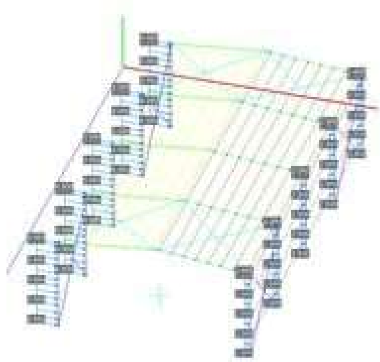


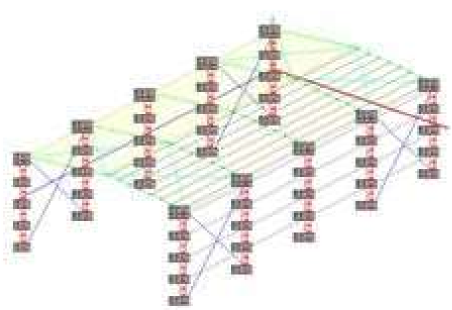
The result on your screen will be to display the snow loads on each roof purlin in yellow as vectors pointing downwards.
The color can be changed from the "**Load Groups"** field. Here we have changed the color to orange for clarity.

In this way you have complete control over not only the geometry of the structure but also the loads imposed by any load, individually or collectively.

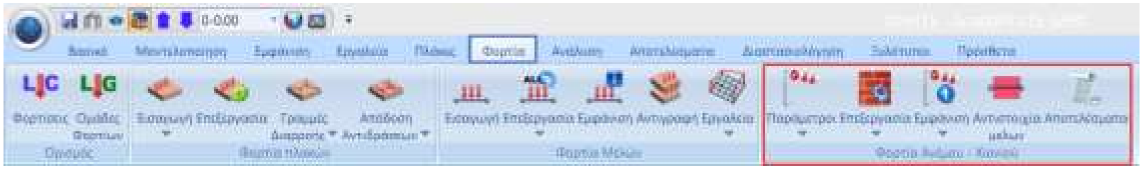
Then repeat the procedure to enter the wind pressure loads by x and z on the columns. After entering them the loads should appear as in the figures below:



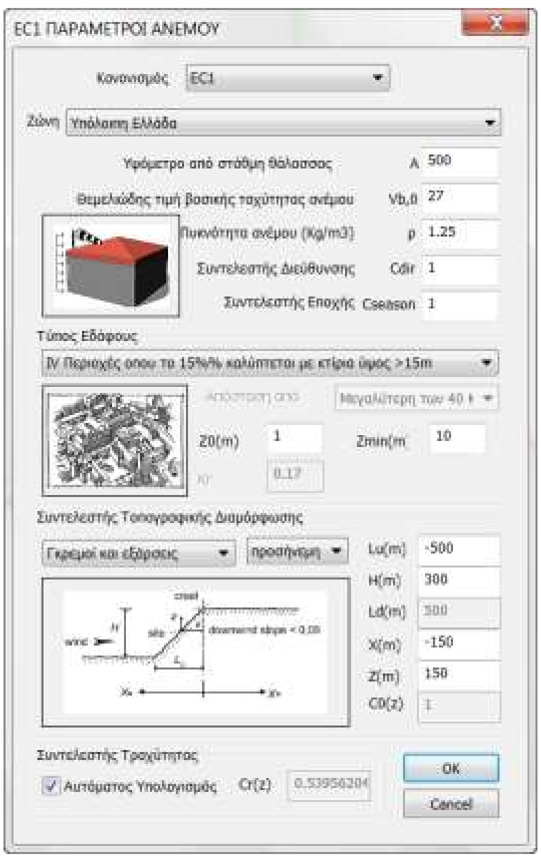


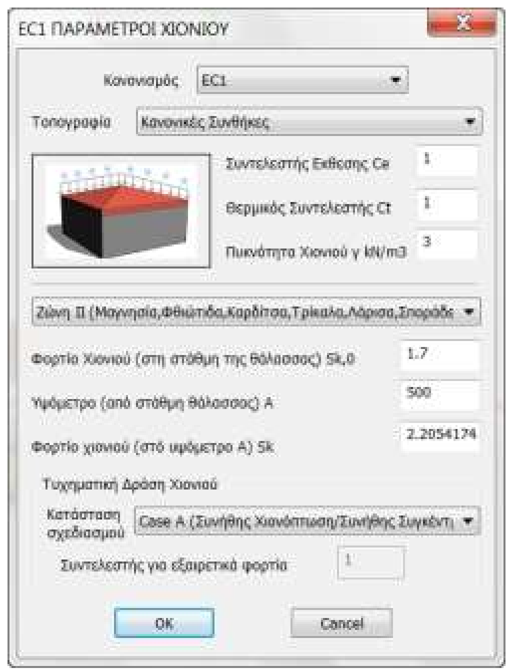


3.2 How to enter wind and snow loads in the automatic way based on Eurocode 1



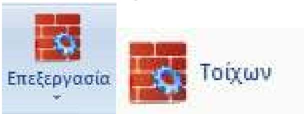
Within the "Loads" Module and the "Wind - Snow Loads" command group, you start by selecting the appropriate Wind and Snow Parameters according to the conditions of your study.

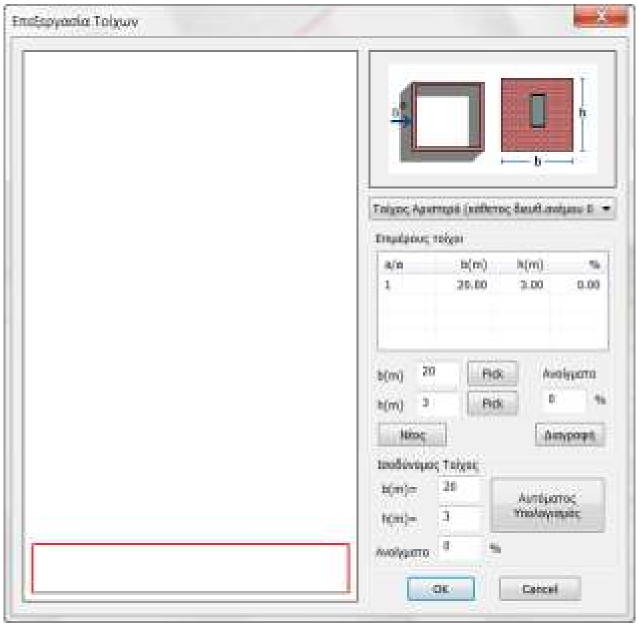




3.2.1 Wall Treatment

Then through the "Edit> "**Walls**",



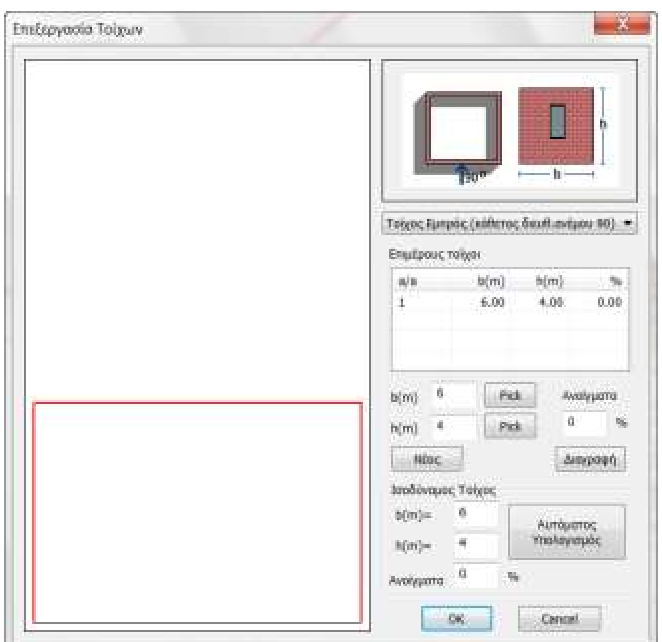


give the program the length (b) and height (h) for each wall (Left, Front, Right, Back), simply by clicking on 

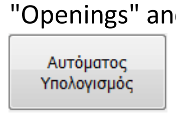
and selecting each time with the mouse the 2 edges of the wall in the corresponding direction, viewing the vector in 3D.

NOTE: The height h is always defined from level 0.

In the cases of importing metallic structures from Standard Structures, the geometry of the walls is automatically entered by the program.



Then, you set the percentage of openings in "Openings" and press the button

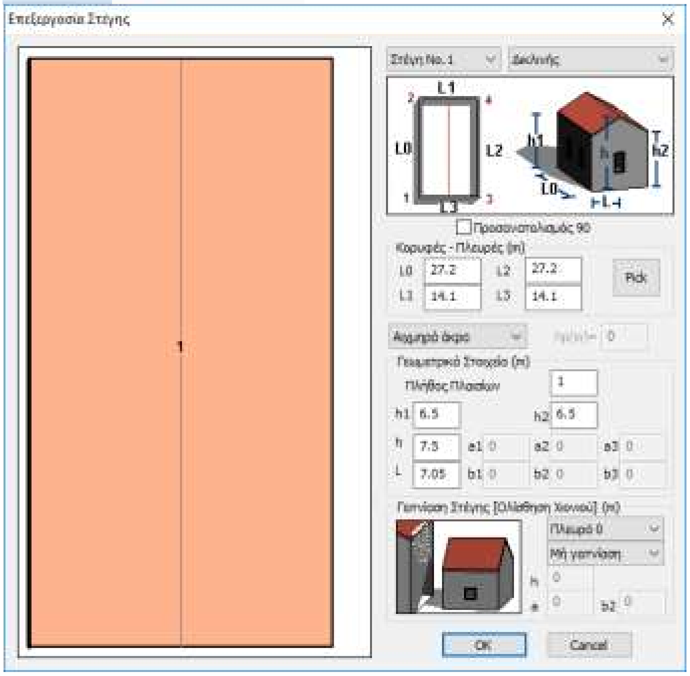


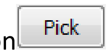
so that the program can calculate the dimensions of the equivalent wall.

Repeat for all four directions of the walls.

3.2.2 Roof treatment

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

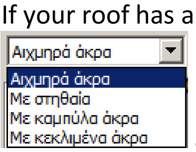


define the type of roof, its orientation and the dimensions Lo,L1,L2,L3, by clicking on 

and selecting each time with the mouse the 2 ends of the roof in the corresponding direction, viewing the vector in 3D.

In the cases of importing metallic structures from Standard Structures, the geometry of the roofs is automatically imported by the program.

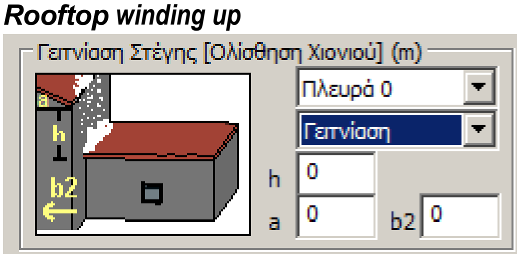
If your roof has an obstacle (snow accumulation point) select the type of obstacle from the corresponding list



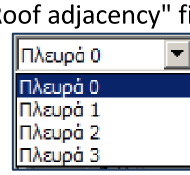
and enter its height in m.

In the "Geometric Elements" field, enter the number of frames and the remaining geometric elements in m.

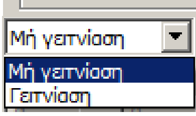
Rooftop winding up



If the structure in question is adjacent to another taller one, in the "Roof adjacency" field select the side that bordering



and from from list



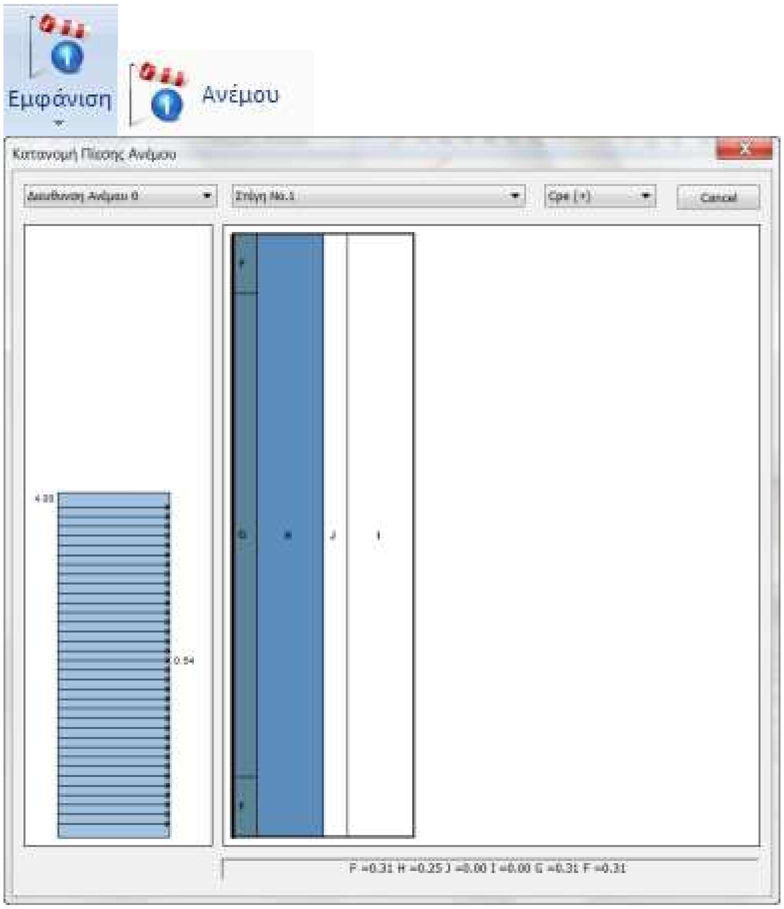
The field "Roof Coordination" is modified accordingly to enter the necessary geometric characteristics. "OK" to save the parameters.

Repeat the process for all four sides of the roof.

3.3 Show

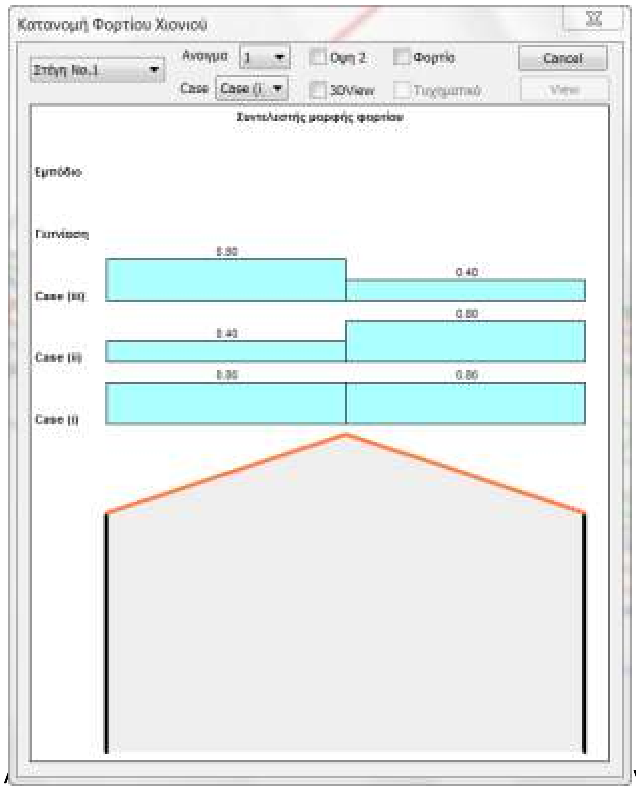
3.3.1 Wind Show

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

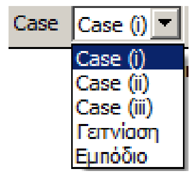


3.3.2 Snow Show

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

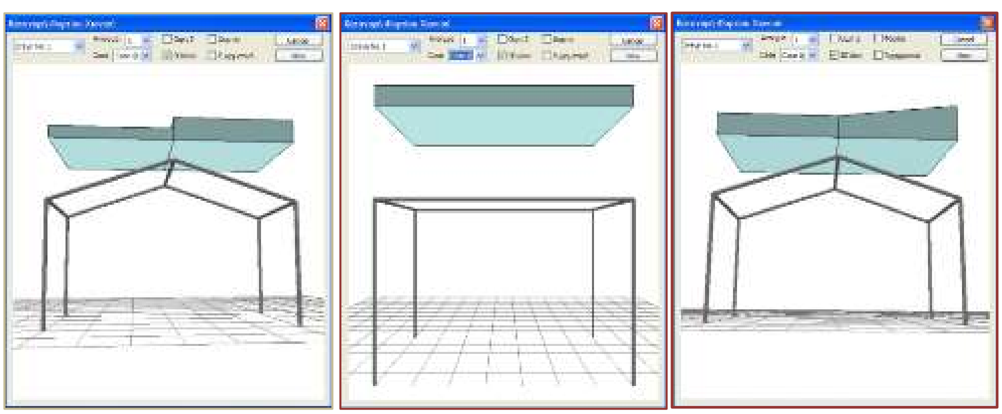


In the dialog box select from the lists on the top left the number of the "roof", the "opening" meaning the number of the box, in case you have more than one, and the "Case"



for snow load distribution.

values and next to "3DView" to get the snow distribution in the following illustration.



3.4 Matching members

to assign the calculated loads to the corresponding members, through the zones of influence.

Select the command and in the dialog box you select one by one the walls and/or roofs for the distribution.

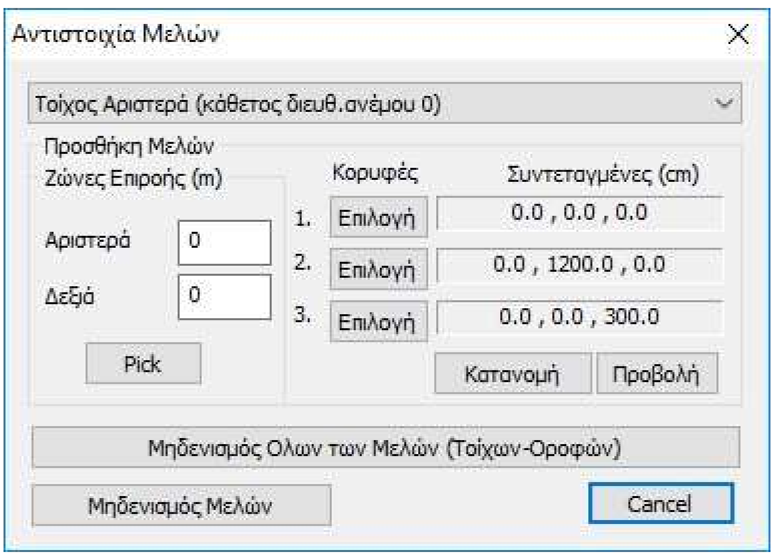
In the new version of SCADA Pro, the automatic calculation of the influence surfaces for the linear members was completed and integrated in order to distribute the wind and snow loads.

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

⚠ In the corresponding chapter of the User Manual (**See Chapter 6 "Loads"**) you will find a detailed description of the

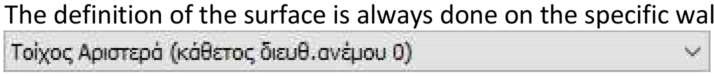
Manual, **Semi-Automatic** and **Automatic**:

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



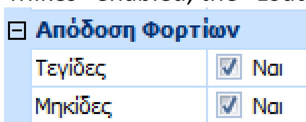
EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

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

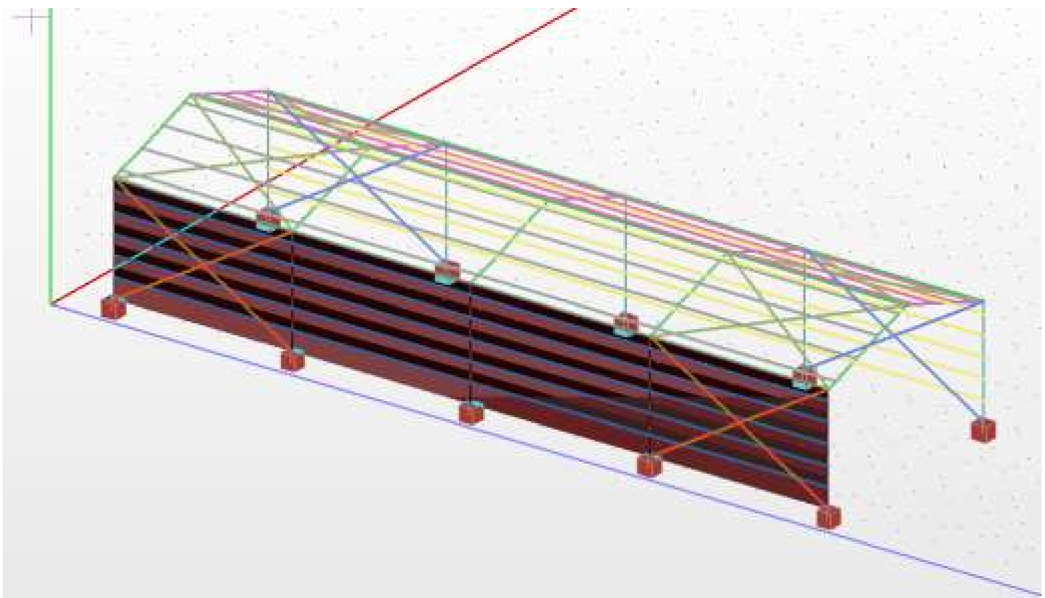
- **Automatic Process - <u>Using</u> the "Standard Constructions"**

Attention:

⚠ *In the **automatic** process coming from Standard Constructions DO NOT press the button "Member Zeroing", because the automatic allocation of loads to members will be deleted!!!!*

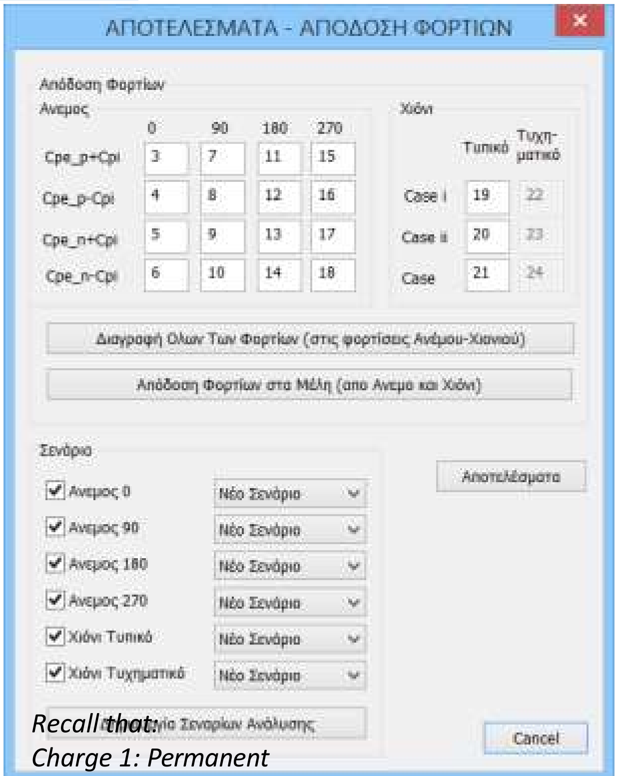
⚠ With "Tags" and "Mikes" enabled, the "Load Performance" field of the "Standard Construction" 

, just select "Pick" and the program automatically calculates the zones of influence by distributing the stresses to all the tectiforms and the mectiforms.



3.5 Results

Last command, the "Results" command.



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

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

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

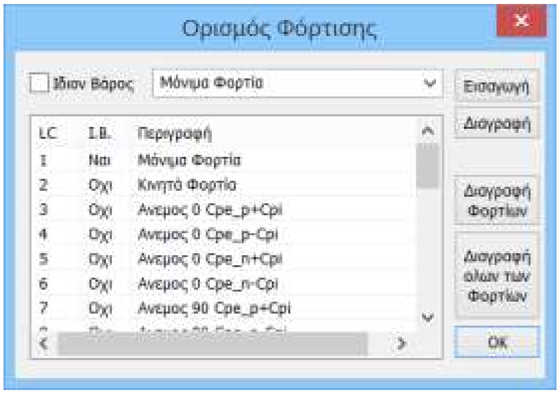
Recall that:
Charge 1: Permanent
Charge 2: Mobiles
and now another 16 loadings are added for wind (from 3 to 18) and 3 for snow (19, 20 and 21)

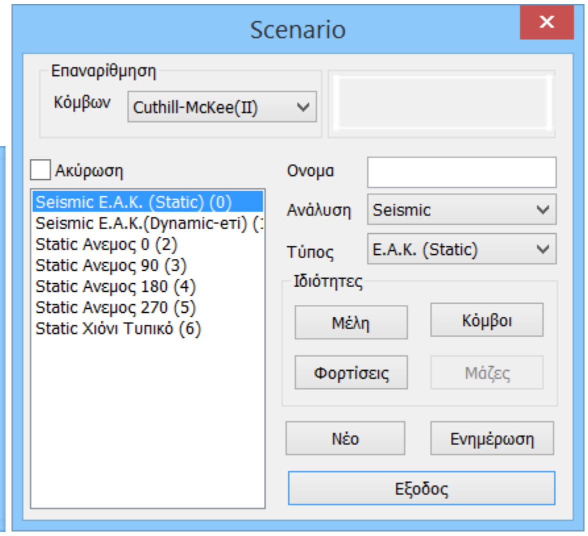
Select the command [Απόδοση Φορτίων στα Μέλη (απο Ανεμο και Χιόνι)] to attribute the Loads of wind and of snow at members the construction, or [Διαγραφή Όλων Των Φορτίων (στις φορτίσεις Ανέμου-Χιονιού)] delete them all.

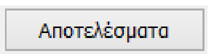
The "Scenarios" field contains a list of all possible analysis scenarios, which are automatically generated via the command ! [Δημιουργία Σεναρίων Ανάλυσης]

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

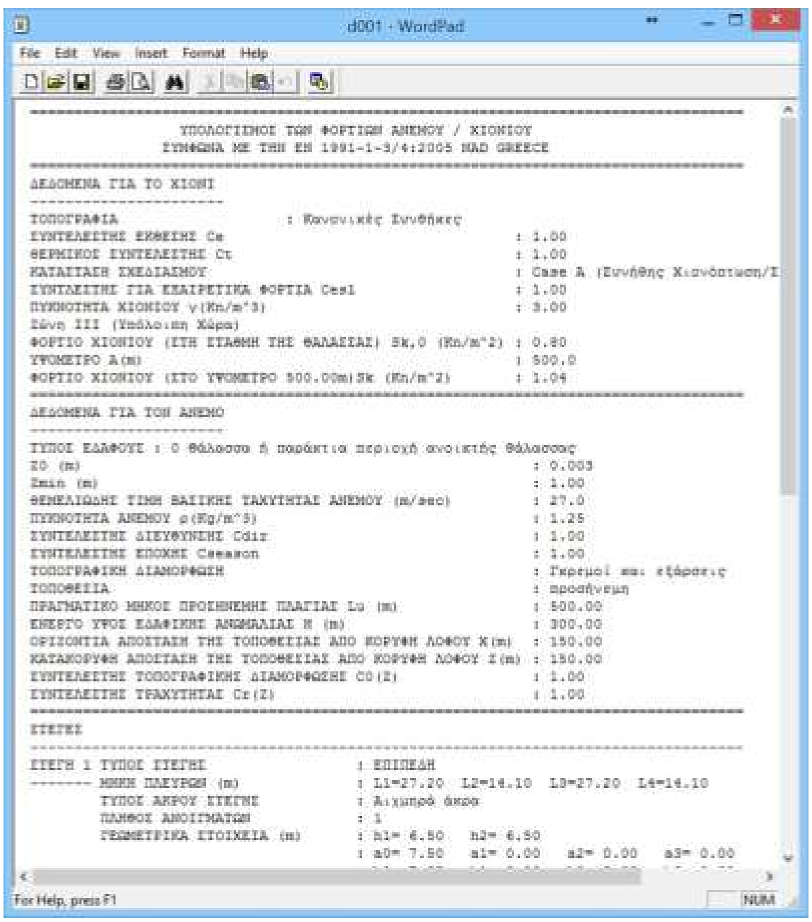
EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'





The command 

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

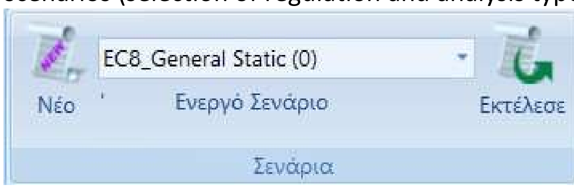


4. ANALYSIS

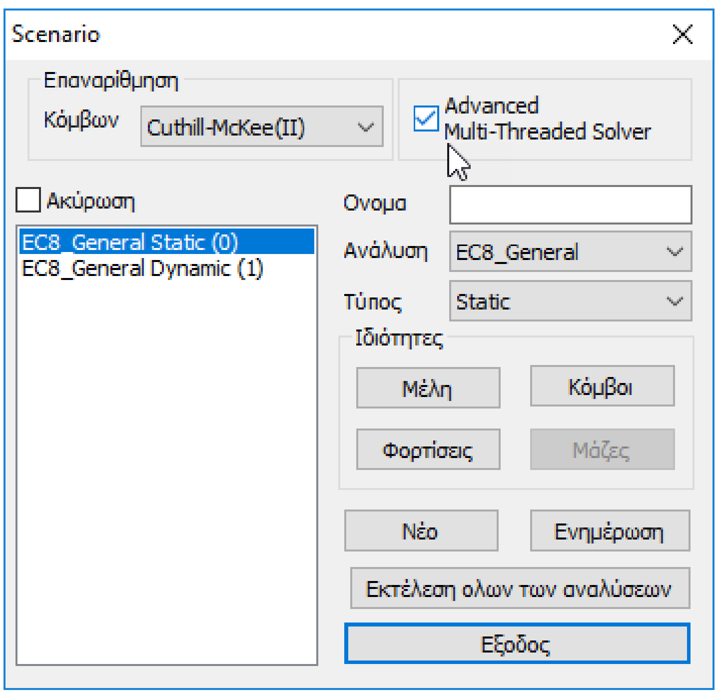
After the completion of the modeling of the structure and the input of the loads in the members, the analysis of the design based on the regulation you will define, the automatic creation of the load combinations and the results of the checks that will be obtained.

4.1 How to create an analysis script:

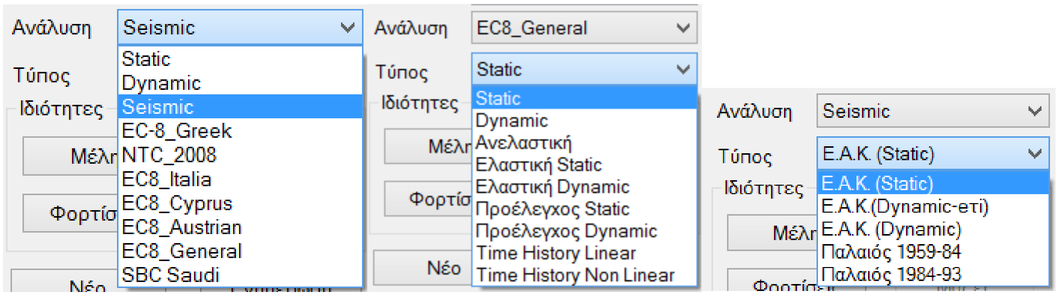
Within the "Analysis" Module, the commands of the "Scenarios" group allow the creation of the analysis scenarios (selection of regulation and analysis type) and their execution.



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 2: 'DESIGN OF A METAL STRUCTURE'

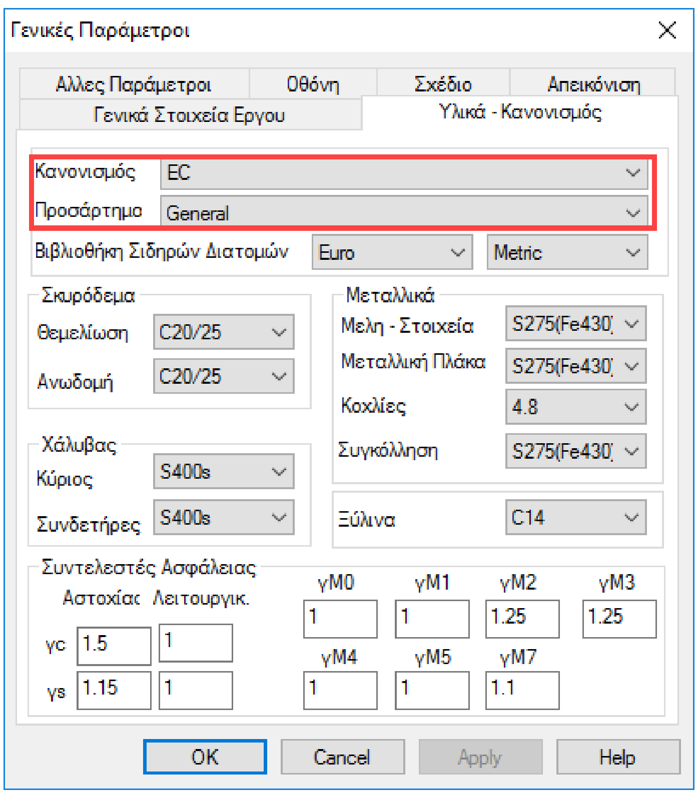


Select from the "Analysis" list and the corresponding "Type" list and click on 

to create a new script.

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

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



NOTE: Materials are automatically adjusted according to the selected regulation,

so that when entering data, all sections are given the correct grades and reinforced with the corresponding steel.
SCADA Pro allows you to choose between the following analysis scenarios:

For Greece:

ELASTIC - UNELASTIC

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

For overseas:

ELASTIC - UNELASTIC

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

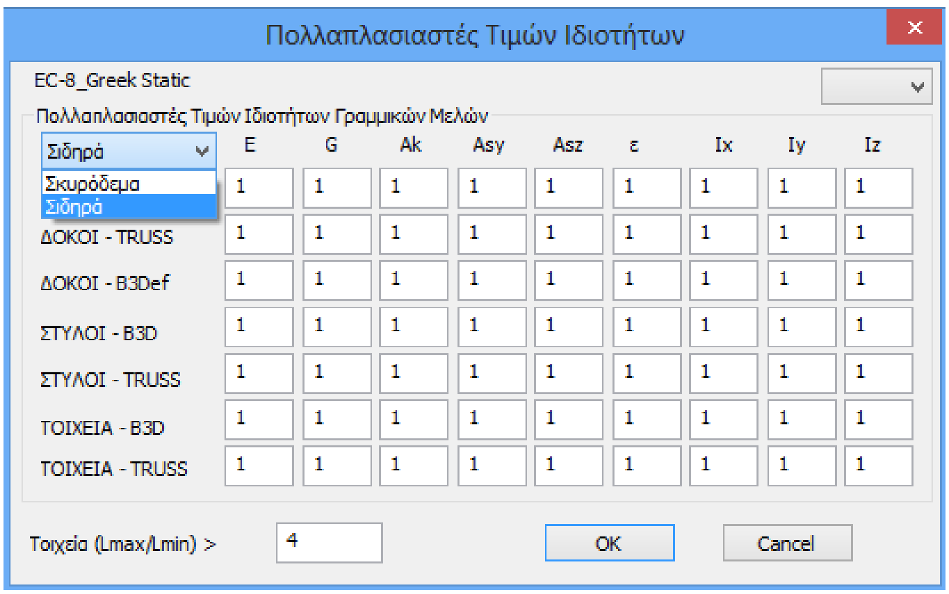
For this example you will select only the EC8 dynamic earthquake scenarios, as well as the Snow Typical, Wind 0 and Wind 90 scenarios, which were automatically created by the program in the previous step.

The Wind 180 and Wind 270 scenarios are not necessary, as this vector is symmetric.

In the usual carriers with linear members, "Renumbering" is not necessary, in large carriers it is recommended, while in carriers with surface members <u>it is necessary</u> to option "Cuthill-Mckee(II) recount".

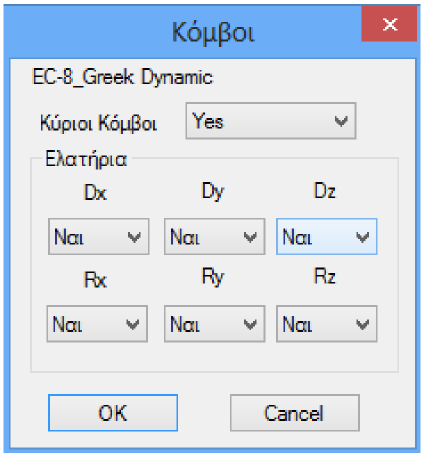
With the EC8 scenario selected Dynamic the **Properties - Members** command includes the property value multipliers of the linear members.

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



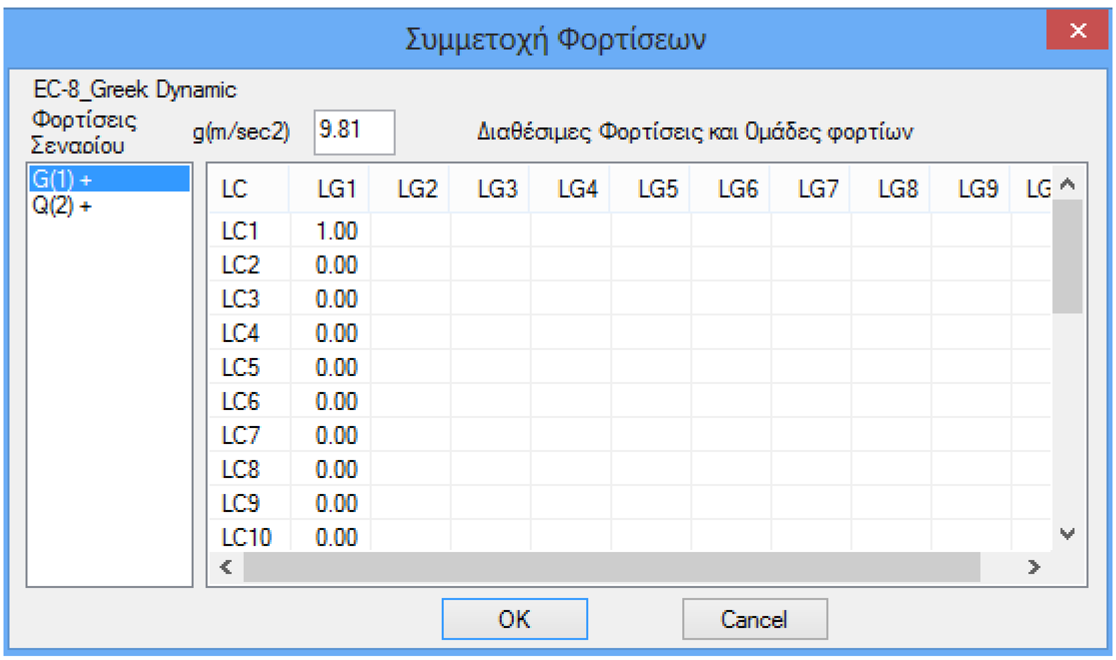
With the EC8 Dynamic script selected the "**Nodes**" command displays the following dialog box:

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'



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.

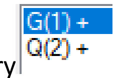
With the EC8 Dynamic script selected, the "**Load**" command opens the following dialog box:



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).

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

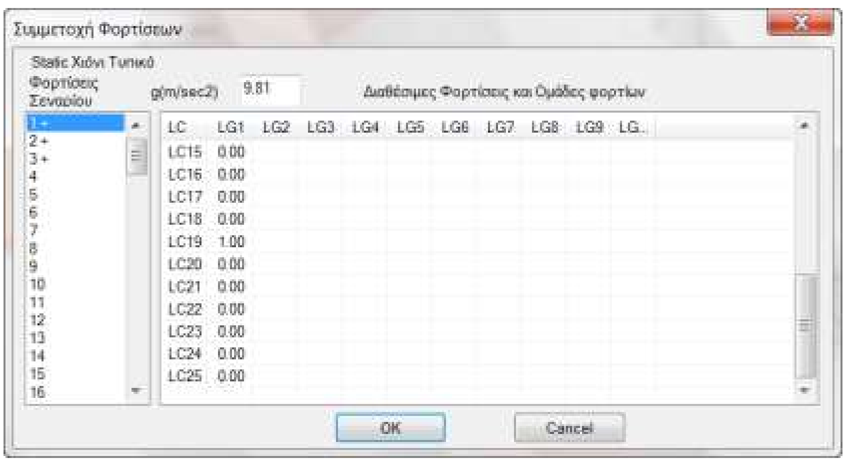
- The "+" next to the charge category 

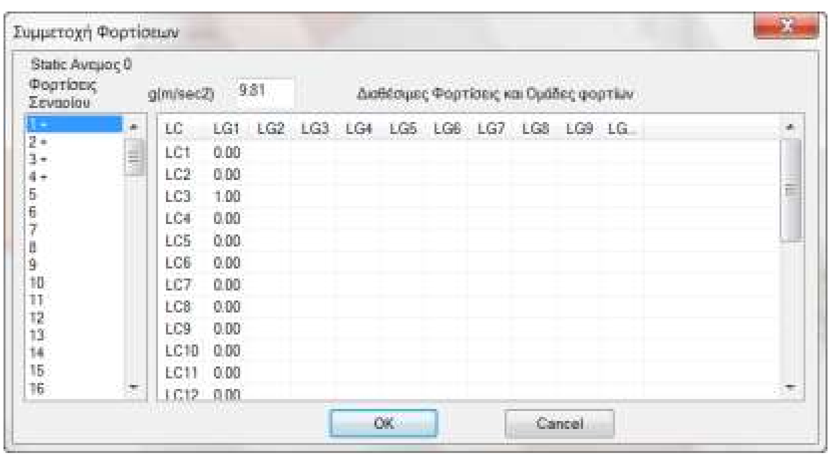
indicates that there is a charge participation for that particular charge.

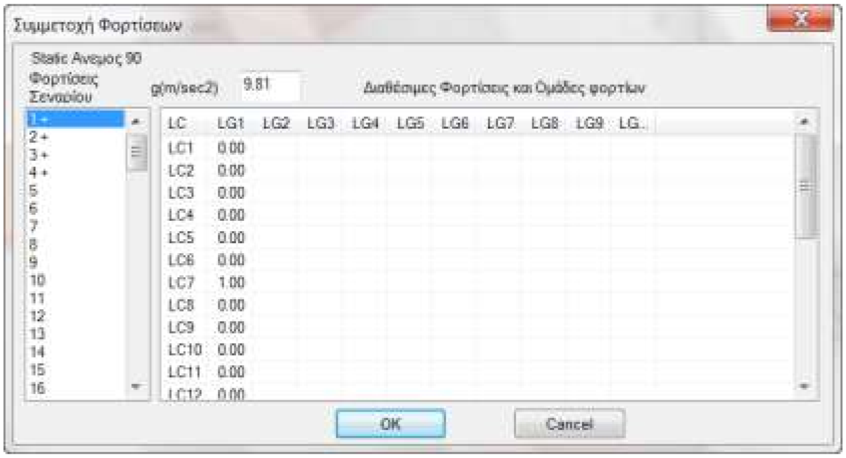
to update the script and register the changes.

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

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



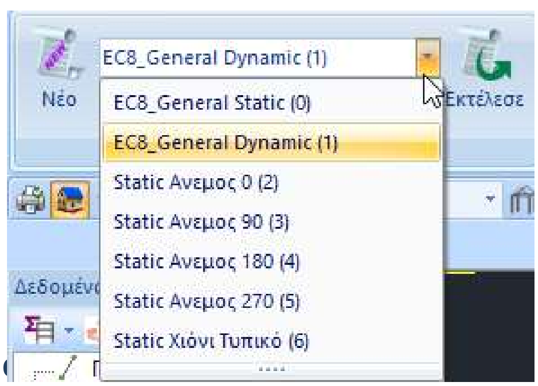




When each charge is activated, the+ symbol appears next to the charge number.

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

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

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

allows 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:

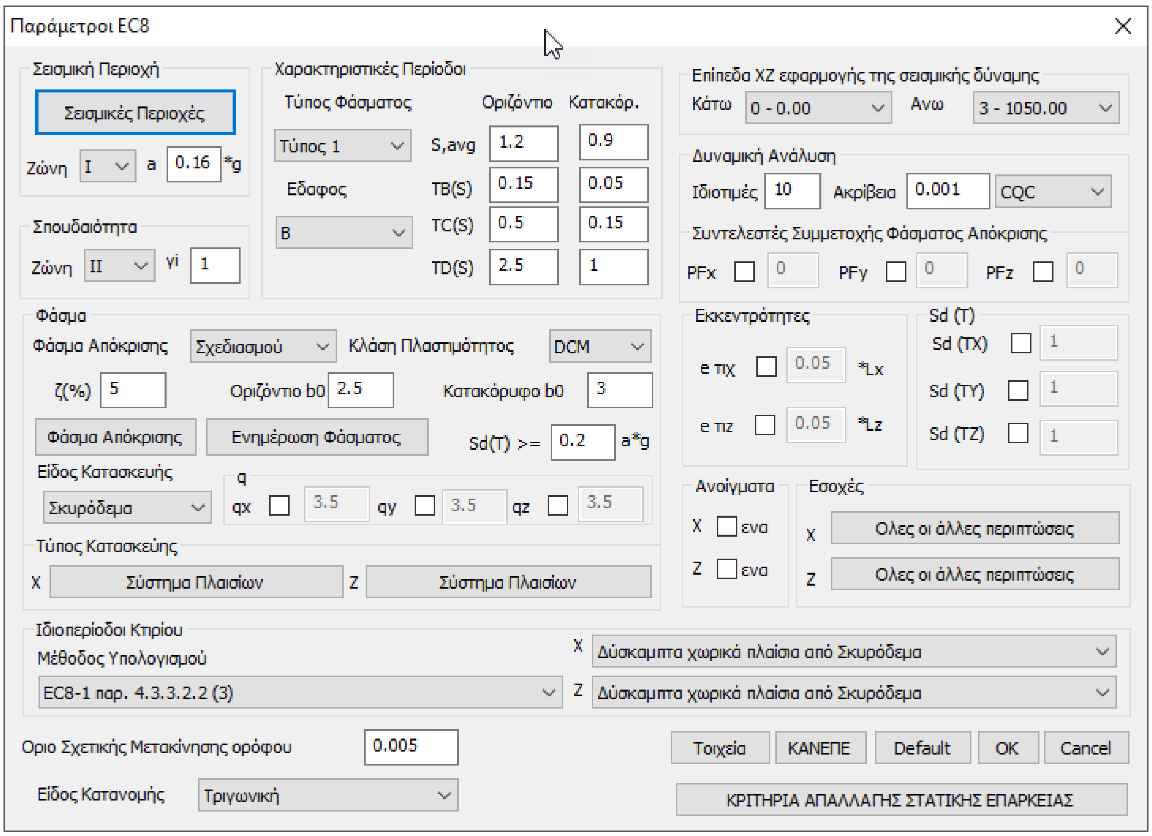
- √ the scenarios of the **Eurocodes** and
- √ the scenarios of **Statics**

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

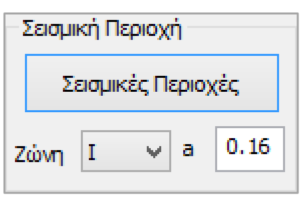
First of all, you select Ενημέρωση Δεδομένων to update the parameters of the active script and delete the data of the previous execution process.

Then, select Παράμετροι 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:*

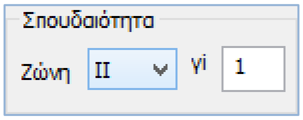


In this dialog box, enter the necessary information about the seismic area, the terrain and importance of the building, as well the earthquake application factors and levels.

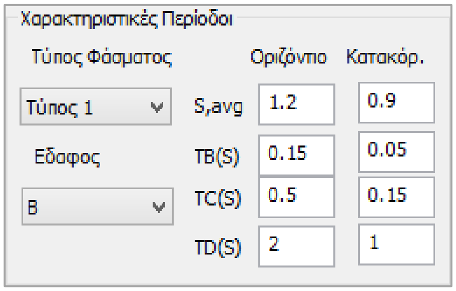


Select the seismic zone, after first checking the file that appears by clicking on "Seismic Areas" for the number of the zone corresponding to the municipality where your study belongs. Select the number from the "Zone" list and the factor "a" is automatically filled in.

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'



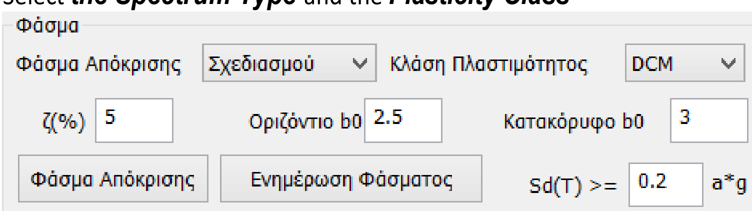
Accordingly, you select the "importance category" to automatically fill in the importance factor "y".



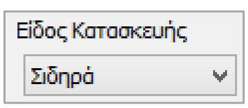
Then you define the type of spectrum (in Greece type 1 is used) and the soil category, so that the coefficients for the horizontal and vertical spectrum are automatically filled in.

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

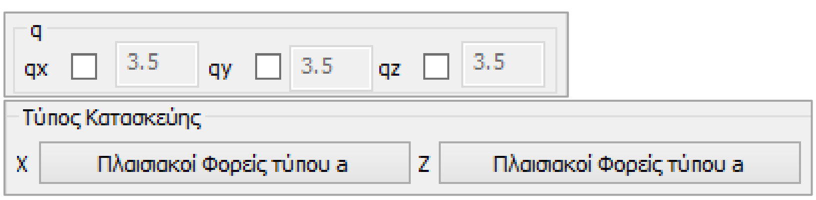
Select ***the Spectrum Type*** and the ***Plasticity Class***



Select the ***Type of Construction***

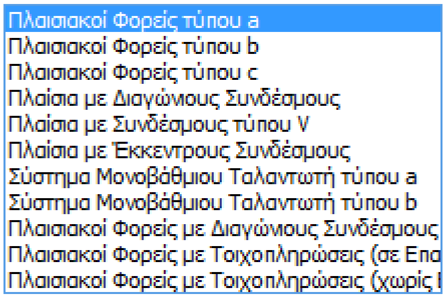


The choice of the ***Seismic Coefficient q*** and the ***type of construction*** requires complex calculations.



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

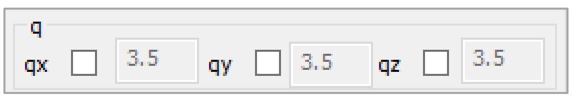
EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'



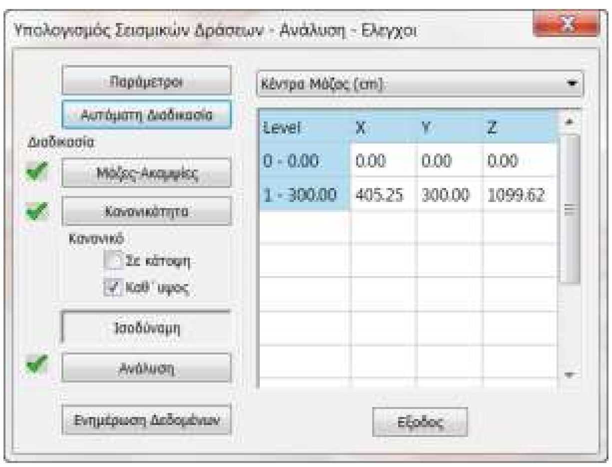
According to the Eurocode the **"Seismic behaviour coefficient q"** is derived from a calculation and the **"Type of Construction"** from specific criteria.

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:

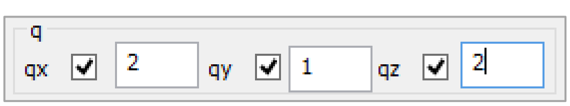


- Select "OK" and with "Automatic Process" perform a **first analysis**.



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

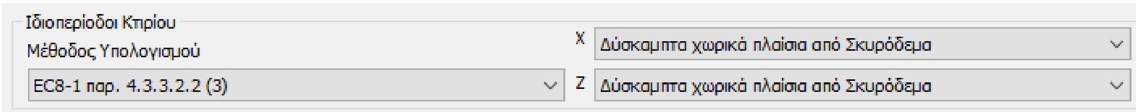
- You can proceed by keeping these values or modify them by checking the corresponding checkboxes and entering your own values (which you could done from the beginning, but then the program would receive your values without calculating the EC8-based values).



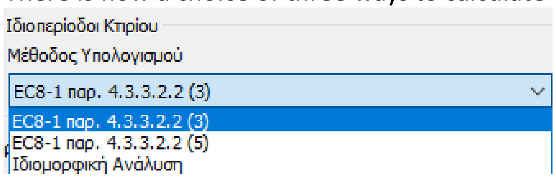
EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

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

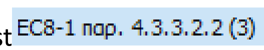
Where in previous versions there was the ***Building Type*** by X and Z field for the calculation of the basic eigenperiod, it has been replaced by the module:



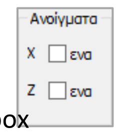
There is now a choice of three ways to calculate the eigenperiod everywhere.



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

1. In the first  is necessary:
   select, per direction, the type of building on the right

(in the case of X and/or Z where the structure consists of a single frame, the

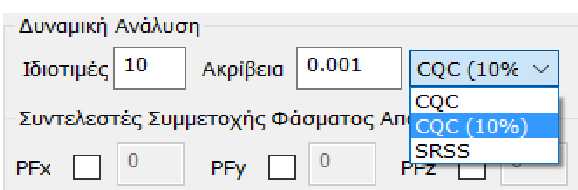
the corresponding checkbox in the "Openings" box )

2. The second approximate method , is sufficient to be selected and does not require any additional action.

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

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

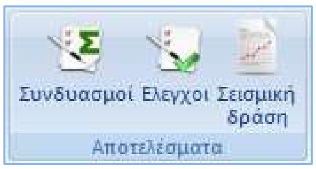
The user can increase or decrease the number of Idiosyncrasies, in case of dynamic analysis, <u>and Static, in case the user chooses to calculate the eigenvalues from Idiomorphic Analysis</u>, and the accuracy rate.



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

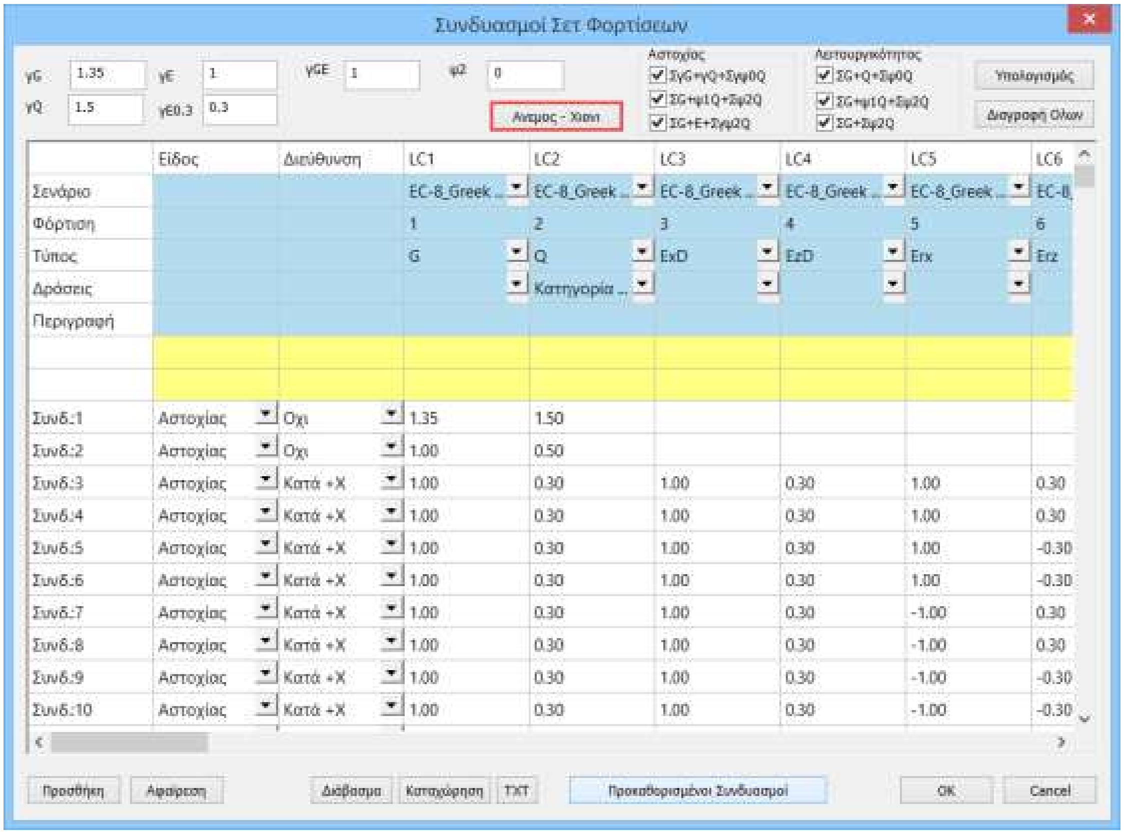
4.3 How to create combinations of charges:

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:



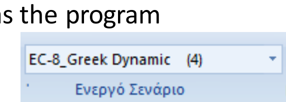
Selecting the "Combinations" command opens the "Load Set Combinations" dialog box where you can create your own combinations or call the predefined ones included in the program.

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'



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.

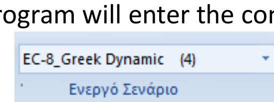
The same is achieved by selecting the "Predefined Combinations" command, as the program will introduce the combinations relevant to the active analysis scenario



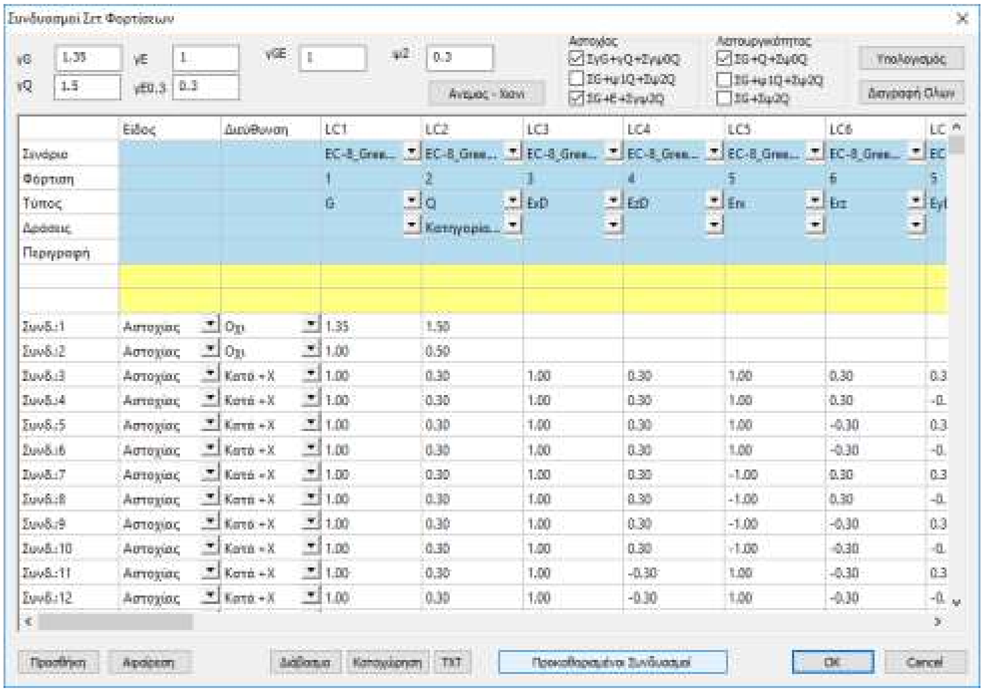
The predefined combinations of the "running" scenarios of the analysis are automatically entered by the program.

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

By selecting the "Predefined Combinations" command, the program will enter the combinations related to the active analysis scenario



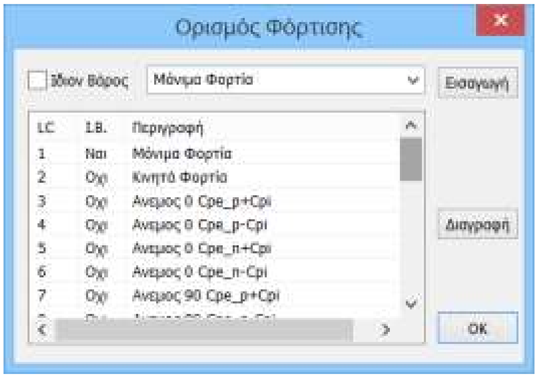
So, if you have dynamics enabled, clicking on "default combinations" will populate the table with the combinations provided by EC8 for dynamics as shown in the image above.

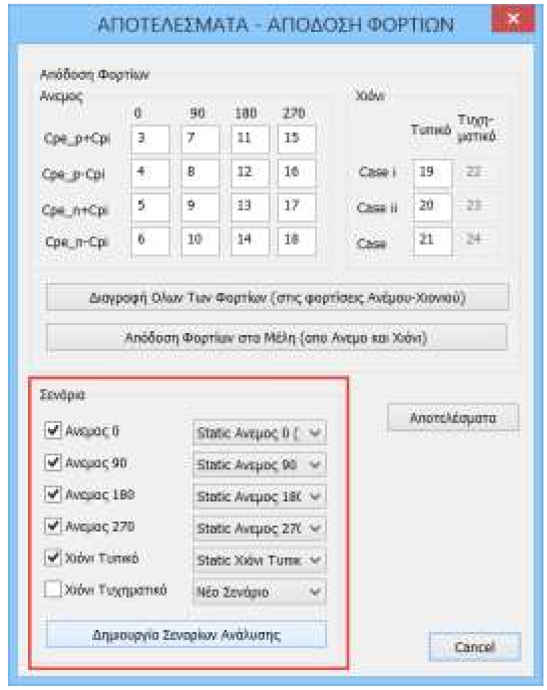


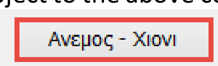
To automatically generate the program and the combinations for the additional loads (snow, wind x, wind z) you have to select from the 3 failure equations and the 3 functionality equations located in the top right part of the window.

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 generation of the loads and combinations (see Chapter 3.2) has been carried out beforehand.

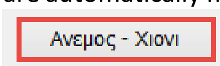
EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

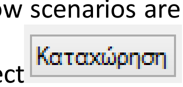




Subject to the above conditions, it is possible to create the wind and snow combinations automatically using the 

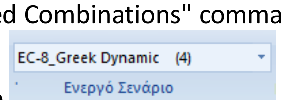
command.

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 

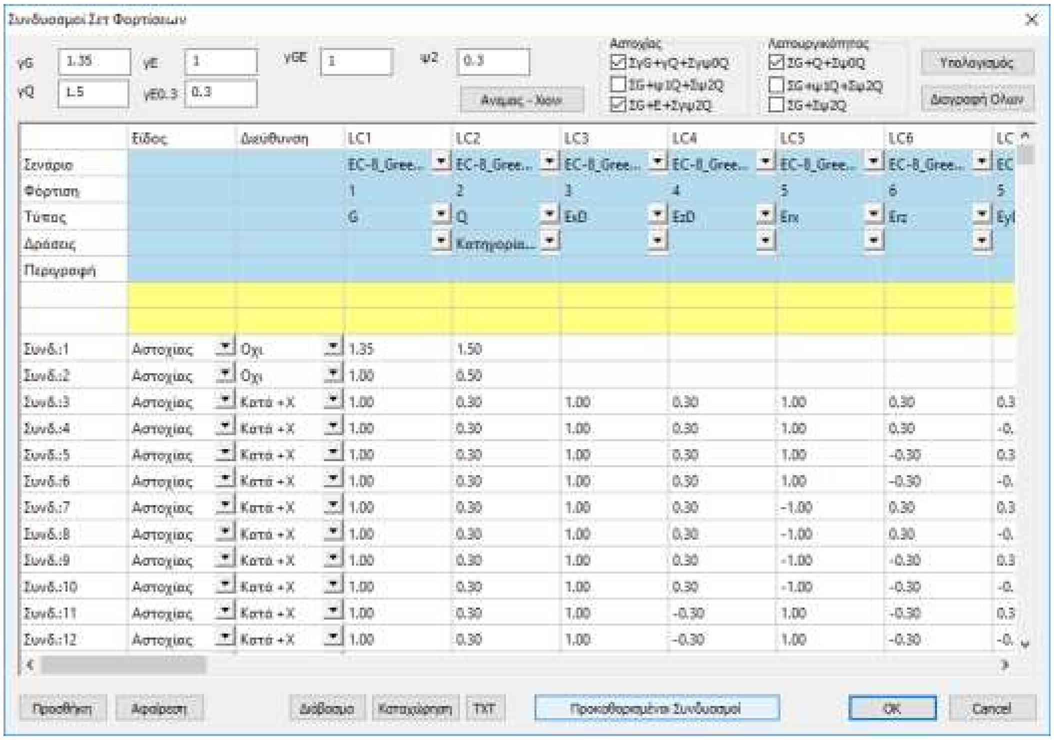
to save it to use it for sizing.

By following the **manual** way you can:

By selecting the "Predefined Combinations" command, the program will enter the combinations related to the active analysis scenario 

So, if you have dynamics enabled, clicking on "default combinations" will populate the table with the combinations provided by EC8 for dynamics as shown in the image above.

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'



To automatically generate the program and the combinations for the additional loads (snow, wind x, wind z) you have to select from the 3 failure equations and the 3 functionality equations located in the top right part of the window.

Select all equations, so that the combinations generated will be based on Eurocode 1. (If respectively only the 1h and 3h failure equation and the 1h functionality equation are selected, then the generated combinations will be based on the EAC).
In addition to the "Default Combinations" you can add others with loadings from other analysis scenarios.

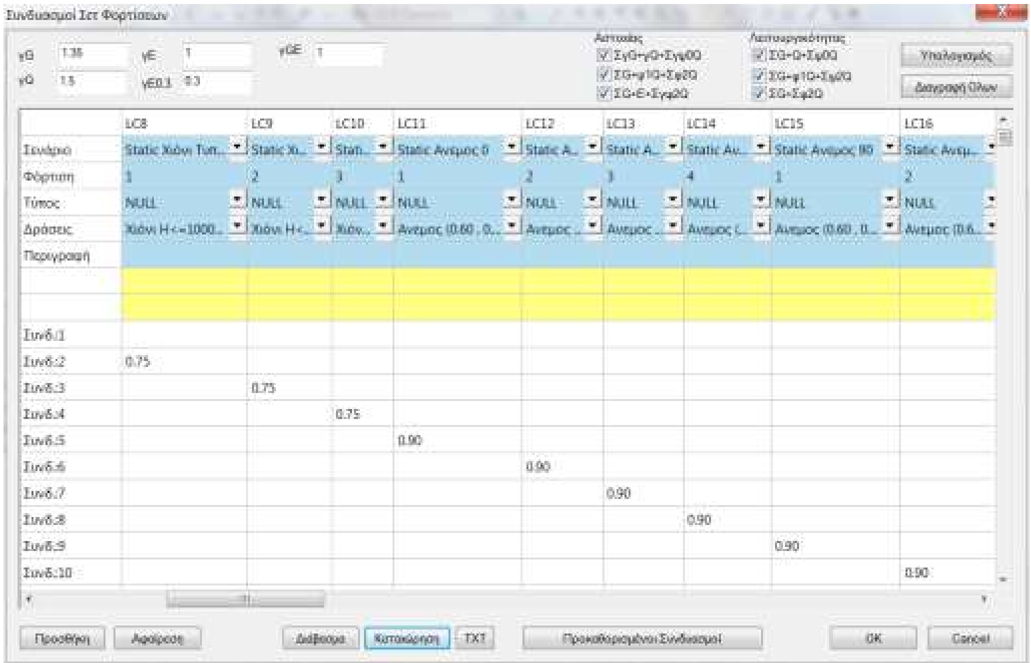
EXAMPLE

For this example you will select all 3(failure)+ 3(functionality) equations and click "Calculate". However, in order to take the additional loadings into account you will need to enter the following in columns LC8, LC9 and LC10:
In column LC8 you put:

- in place of the **Static Snow Typical** script
- in place of the charge **1** (the first charge of this scenario)
- in the type position **NULL** (since snow is not a permanent, mobile or seismic load)
- at the position of **snow actions<1000m**
- in the description field the word CHONI (optional, you can give any name you want)

Similarly for other 2 loadings of this scenario and for the loadings of the 2 wind scenarios , the values shown below:

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'



After clicking on "Calculate", you must select the **entry** command to save these combinations as a *.cmb file in your design folder, to be used later in the superstructure dimensioning and results.

⚠ If you had chosen EC8 **static** analysis and predefined combinations (having the scenario - EC8 Static - active) **9** loadings would have been created. So you would fill in LC10, LC11 etc...) and then calculate to create the 152 combinations.

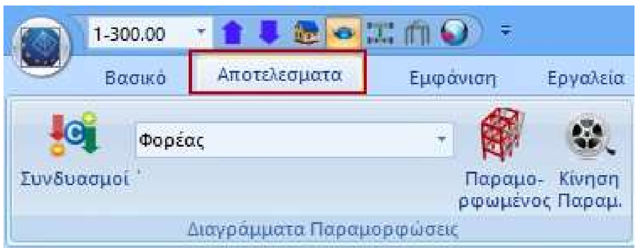
The [Προσθήκη] [Αφαίρεση] commands allow you to add or remove lines or columns after selecting them, as in an .excel file.

The [Διάβασμα] [Καταχώρηση] commands allow you to enter or open a combination file.

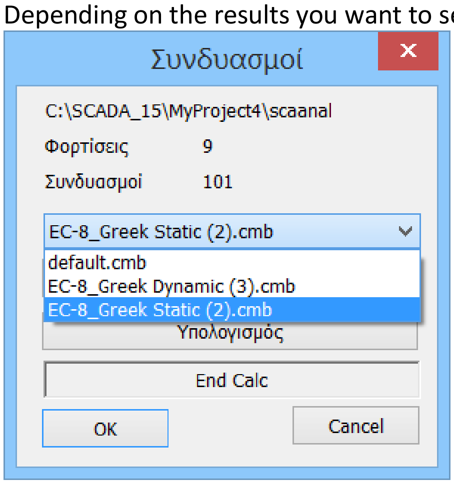
5. RESULTS

5.1 How to view diagrams and deformations:

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



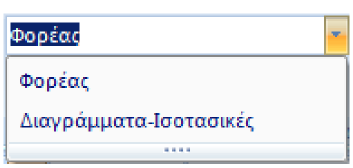
Depending on the results you want to see, from the "Combinations" command and within the dialog box:



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

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

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

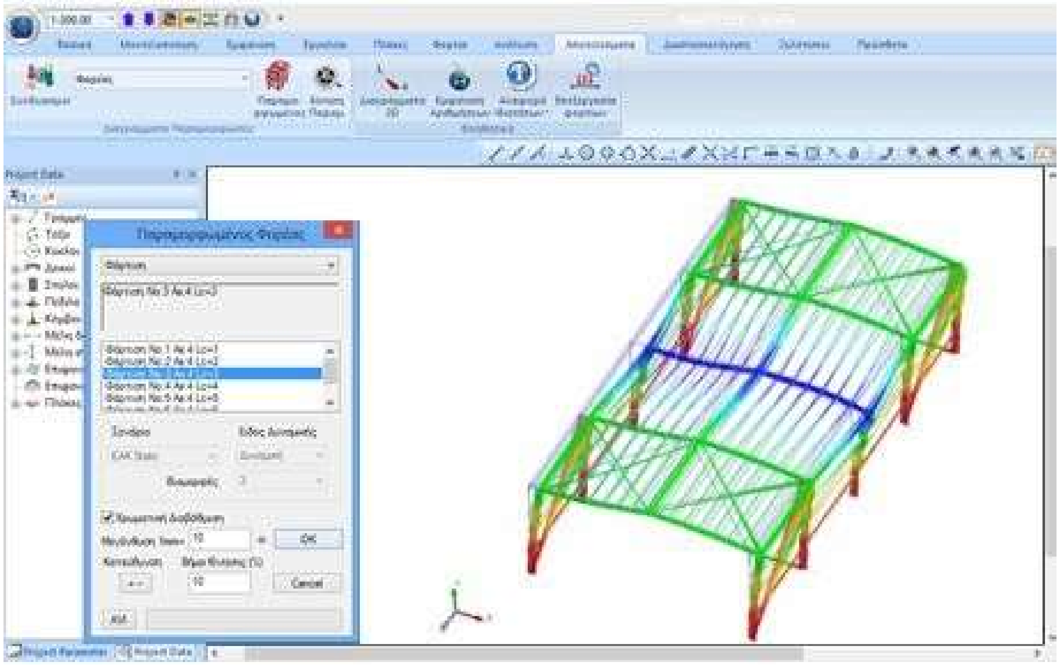


From the list on the right, depending on the results you want to see, select:

√ Institution or

√ Charts-Important

5.1.1 Body+ "Deformed Body"



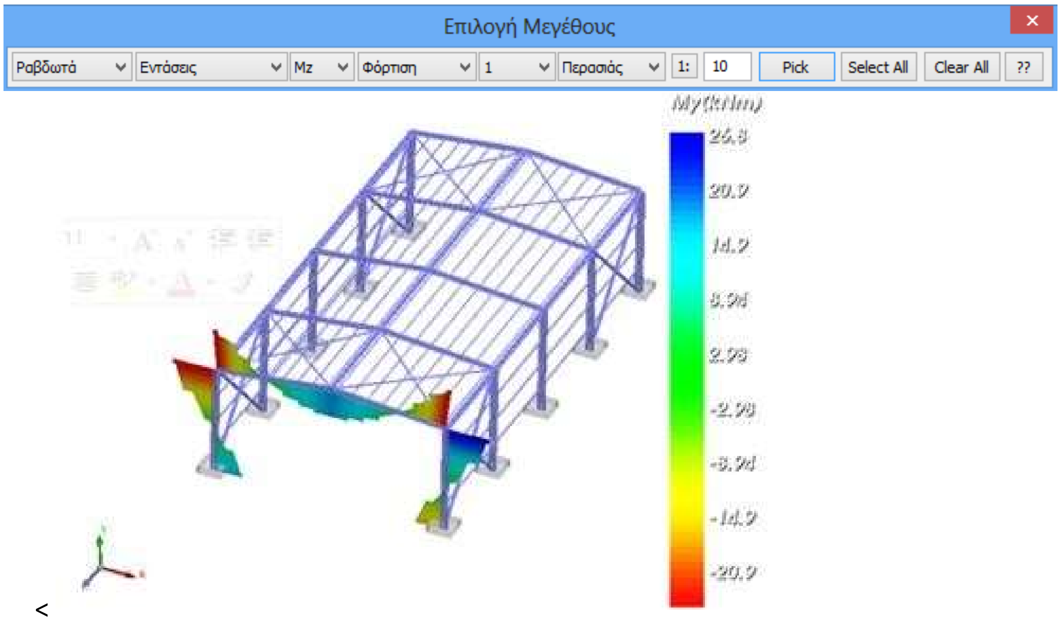
Select from the list Φόρτιση / Συνδυασμός / Ιδιομορφές / Pushover select the type of loading for which you want to see the deformation image of the carrier and from the next list specify its number.

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

ΓΕΩΜ ΠΑΡΑΜ ΦΥΣ-ΓΕΩ ΦΥΣ-ΠΑΡ ΔΙΑΦ.ΓΕΩΜ ΔΙΑΦ.ΠΑΡΑΜ

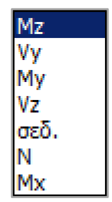
The "Motion" command is the switch that turns on and off the motion of the deformed vector, according to the choices you made in the dialog box of the previous command.

5.1.2 Charts - Equalisation



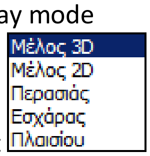
<

In this section you can see the diagrams of the stresses and strains for the linear members, and the isometric curves of stresses, strains and reinforcements for the finite surface elements. More specifically, to see for the Stick figures the diagrams you select the intensive quantity from the list



, in the then select the type of charge or combination or surrounding

and finally select the display mode of chart



member.

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

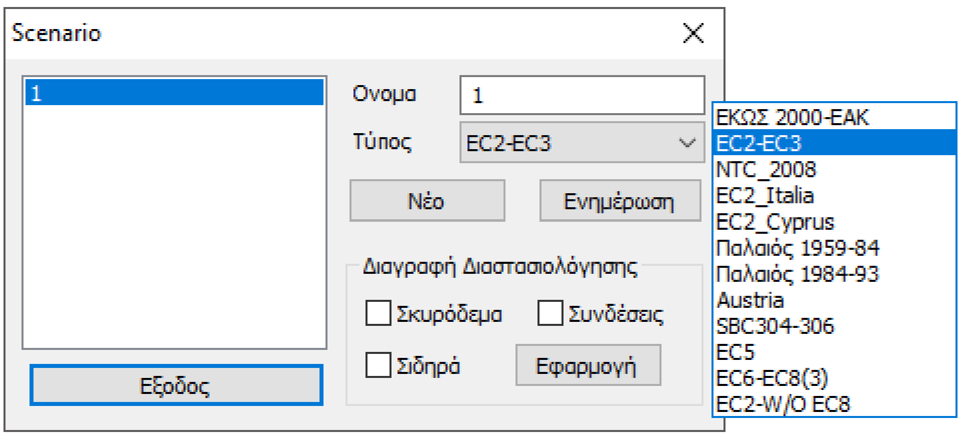
6. DIMENSIONING OF METALLIC SECTIONS

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

6.1 How to create dimensioning scripts :

Go to the "Sizing" section and select the "New" button 

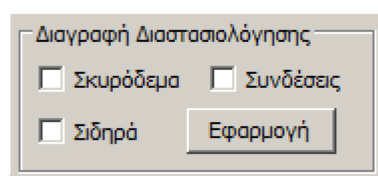
to create the scenario you wish by selecting the regulation (EKOS, EUROcode, Old regulations, for Greece).



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

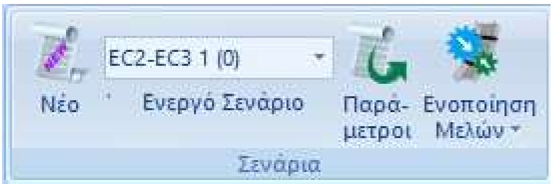
In this example, a Eurocode scenario was used.
Comment: *For metals, EC3 is applied through the program and is included in all scenarios regardless, since there is no corresponding Greek regulation. The EC2 designation refers to the method of analysis as well as the method of dimensioning concrete sections.*

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

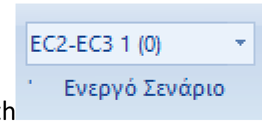


6.2 How to determine the parameters of the sizing of steel sections

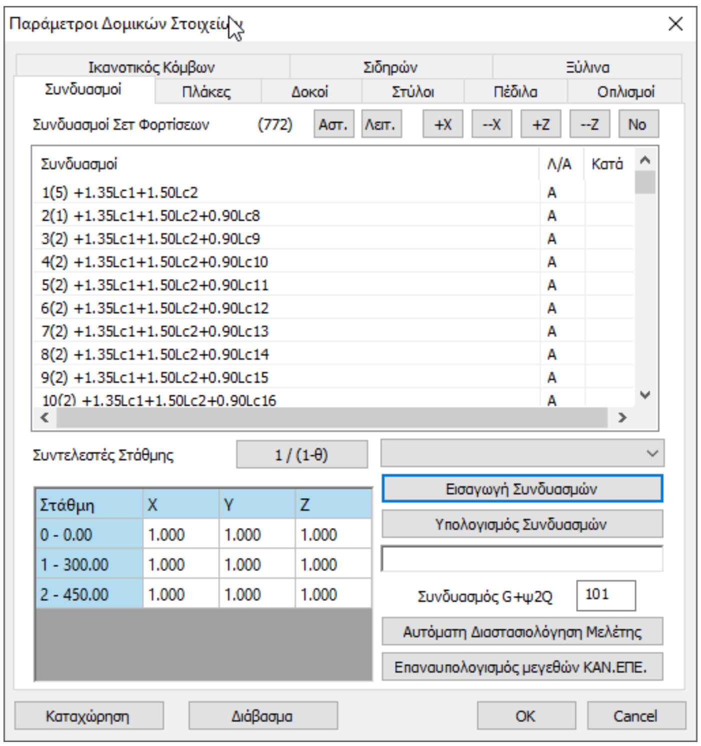
EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'



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

With 

active in the selected script, you display the Parameters 



A prerequisite for sizing is the calculation of combinations.

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

- from the list

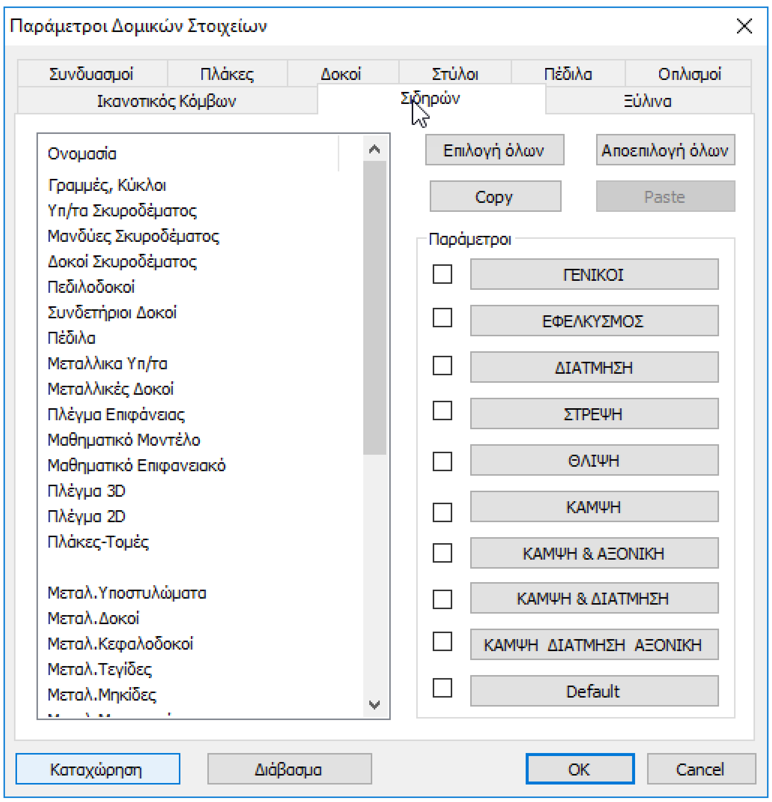
default.cmb
EC-8_Greek Dynamic (3).cmb
EC-8_Greek Static (2).cmb

with automatic calculation

- through the command **Εισαγωγή Συνδυασμών** 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 **Υπολογισμός Συνδυασμών** button to make the calculation.

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

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



The definition of the sizing parameters of the metallic sections is done layer by layer. You select the layer for which you want to define the parameters (e.g. Metallic Sub/Underground) and per control category (General, Tensile, Shear, etc.), you define 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.

EXAMPLE

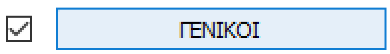
EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

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

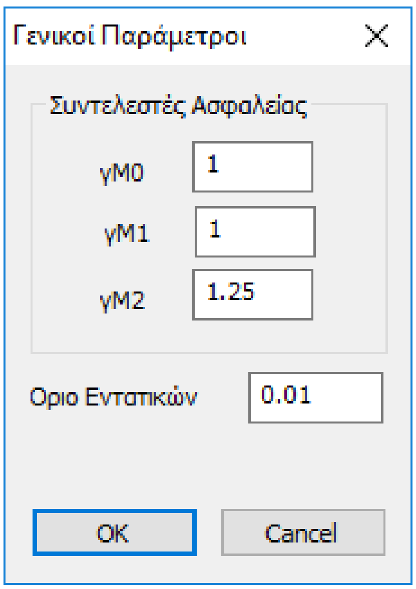
It should also be noted that to set parameters at least one (or more) layer must be selected.

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

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



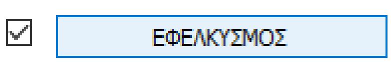
to set the safety factors γM:



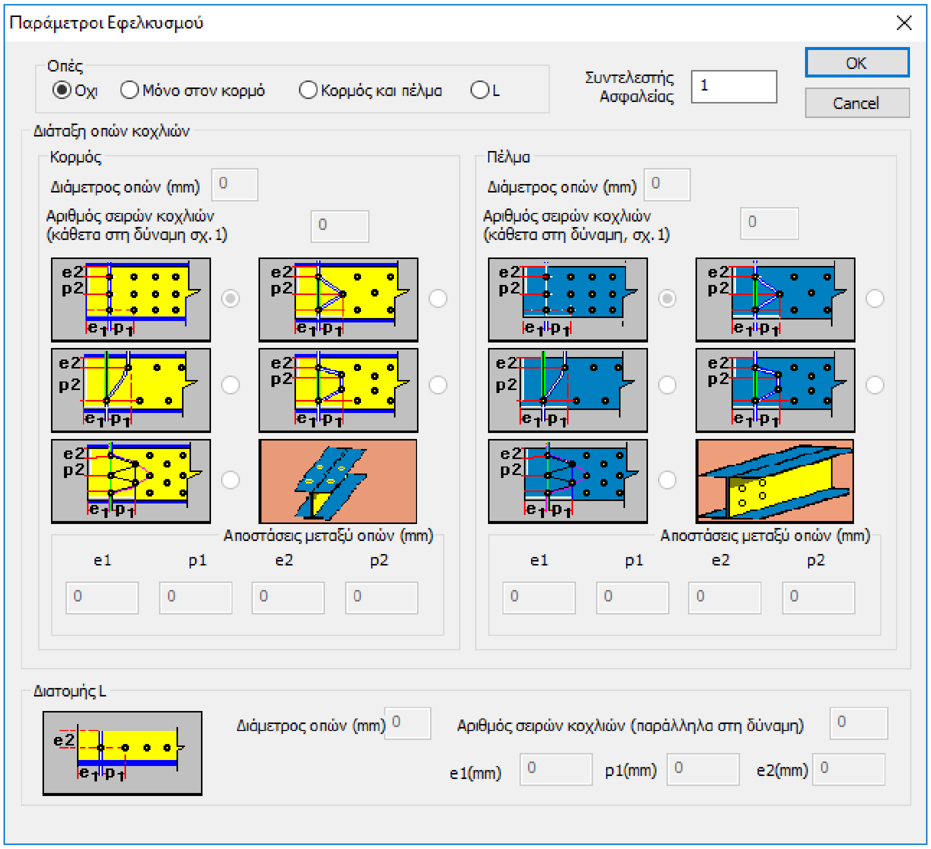
γ_{M0}= transverse stress resistance for each category of members γ_{M1} = resistance to buckling based on tests c_{M2}= tensile fracture strength of cross sections

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.

- **ΕΛΚΙΣΜΣ**"



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

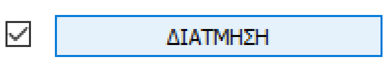


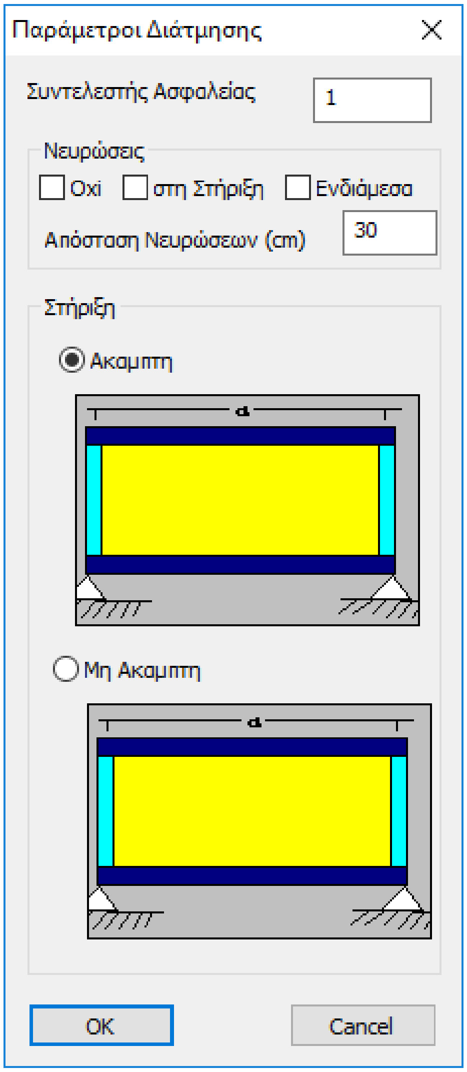
For the holes define the distances from the ends, the diameter and the number of rows on the torso and tread.
In the case of an L section, set the parameters at the bottom of the box.

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 already performed. So the connection check must have been done 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.

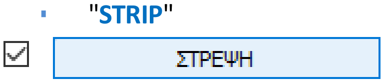
- "**DISTRICT**"



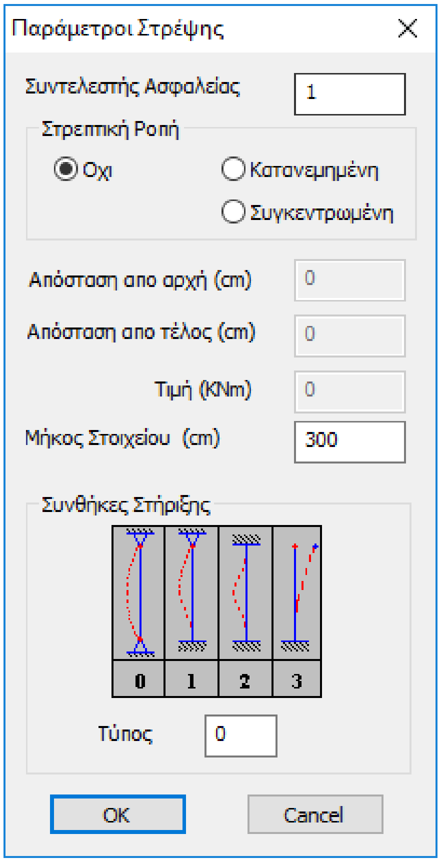


Here you define whether the elements of this Layer have ribs or not and if they do, where they are present (on the support and/or the trunk). You also define the spacing of the ribs as well as whether the support of an element is rigid or not.

- "**STRIP**"

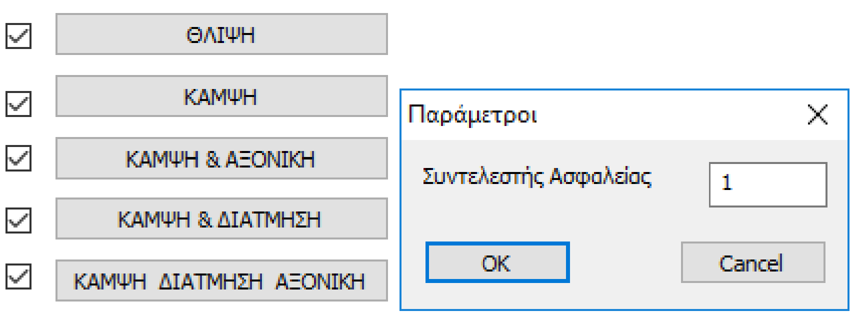


EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

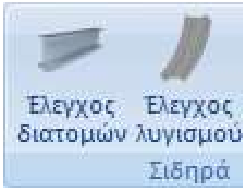


Here you specify whether the members of the layer are loaded by torsional moment (distributed or concentrated). If they are loaded, you define the elements of the loading. You also specify the support conditions of the members based on the support type shown in the graph.

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:

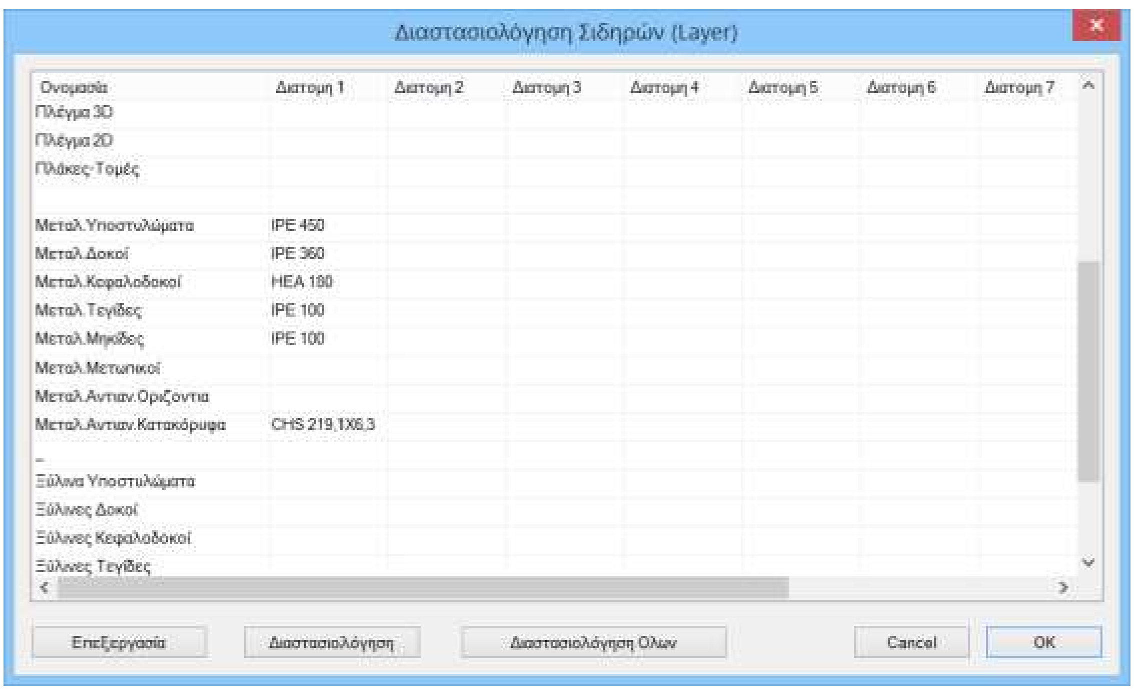


In the "Dimensioning" module, the "Iron" field includes the commands related to the solution of the metallic sections with the adequacy check, the buckling check and the check of the connections.

6.3.1 Control of metallic sections:

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

Using the command, the following dialog box appears.



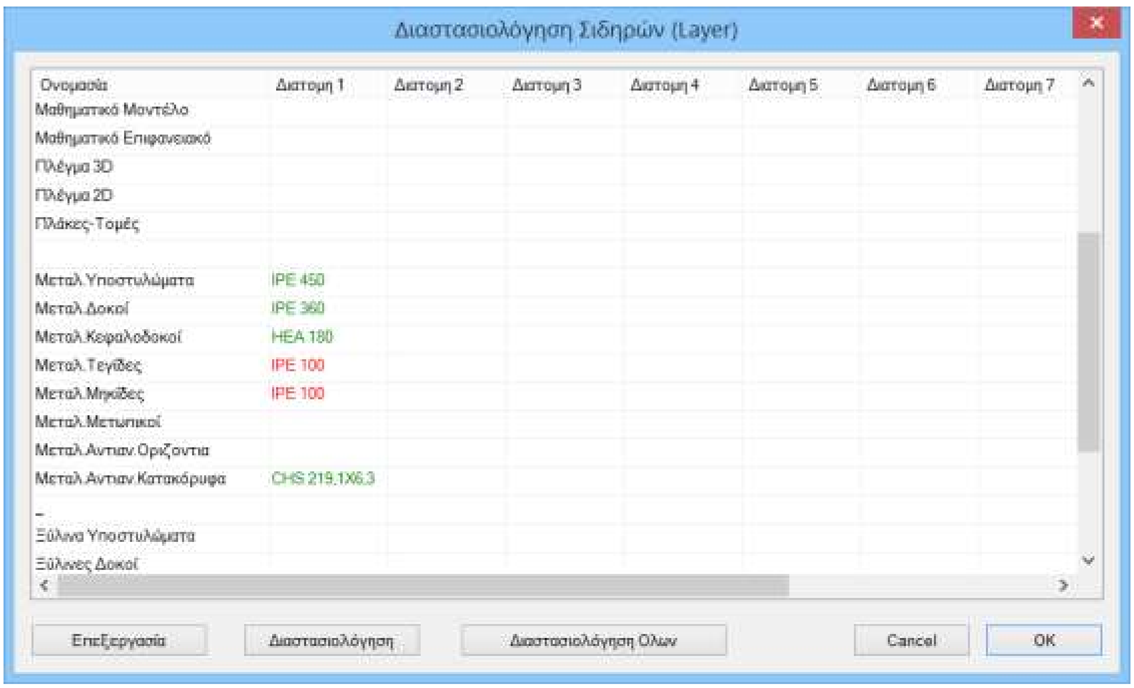
The cross-section check is done globally for all elements in a layer.
For each intensive quantity, the program identifies the element with the worst value for that quantity.
The first column is the layers that exist in this study and in the following columns are the types of metallic cross-sections that exist in these layers. In this particular

For example, in the layer "Metallic supports" the metallic supports of the structure with IPE450 cross-section have been placed. Similarly, the metal beams have been placed in the "Metal Beams" layer and the corresponding members in the other layers.

⚠ *In SCADA Pro you have the possibility to create your own layers by grouping the types of cross sections.*

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

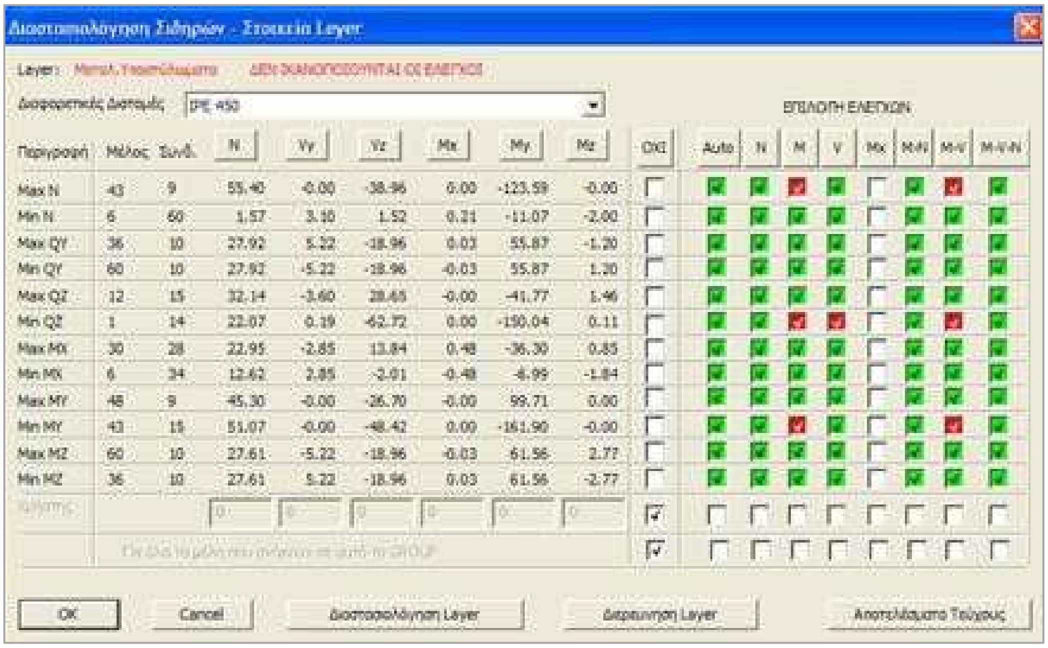
By selecting the command "**Dimension all**" the cross-sections for all combinations will be automatically checked and the groups - layers in which no cross-section fails will be displayed in green and the groups in which even one cross-section has exceeded the unit, i.e. has failed, will be displayed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



The fact that a layer is shown in red does not mean that all members of the layer have failed. For example, to see which columns have failed you should select the "metal columns" 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 this example, all cells in the "auto" column will initially be displayed in green.

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'



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

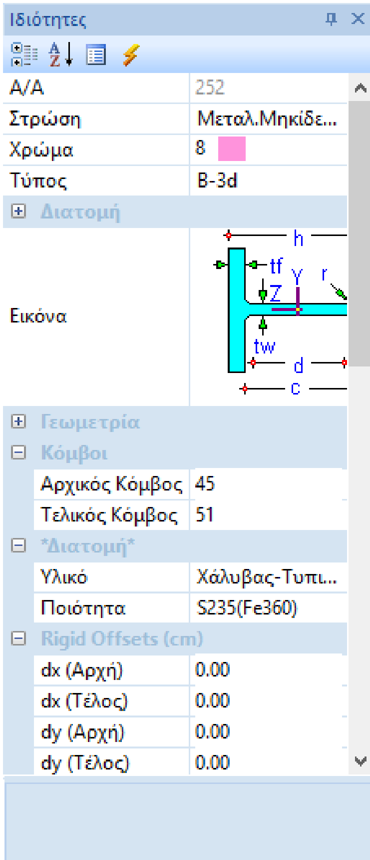
Suppose that member 43 for cod. 9 has the following:
N=55.40KN, VY=0KN, VZ=38.96KN, MX=0KNm, MY=123.59KNm and Mz=0KNm.

For this particular combination the automatic procedure (auto) calculates a ratio of 0.29 - green color in the corresponding cell). This ratio is the worst ratio obtained by testing in M-V-N (MY,VZ,N).
But if you choose to do the check yourself by clicking on the cells in the first row and then on the Layer dimensioning, you will notice that the check in M & M-V fails (red). This is because in this case the program only uses the values of MY,VZ and ignores N (worst case) which however in this example does not exist because the axial exists. In general, in addition to auto you can choose for which individual intensities or combinations of intensities you want the respective checks to be performed.

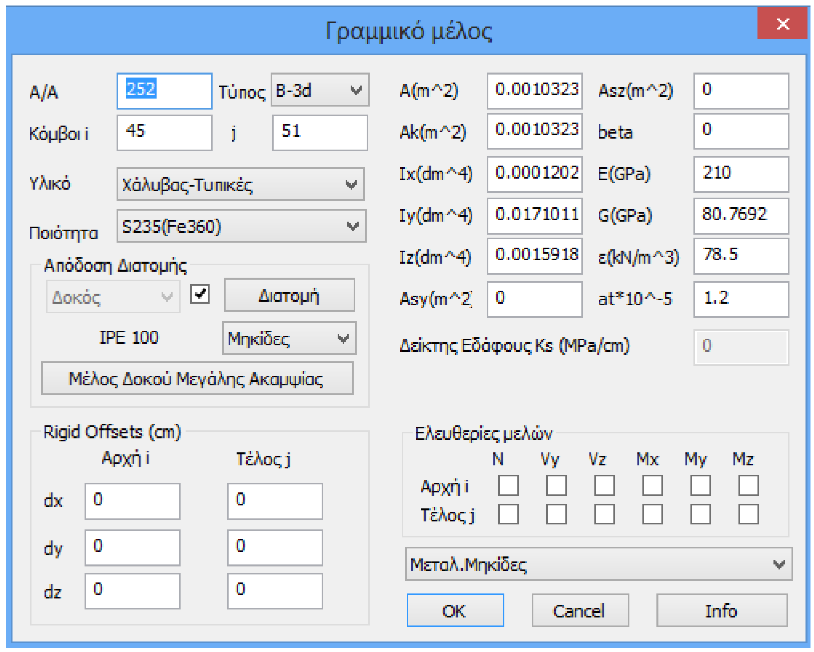
To summarize, when you put your mouse cursor over a cell and it is green, you will see that a value below one unit (sufficiency) is displayed. If instead you point to a red cell then the value displayed will be above unity (miss) .

You also have the possibility to do sizing by entering your own intensive sizes through the "user" option.
The results of the dimensioning in summary, (either from the automatic process or from the selection of the individual intensities, or by the user) can be viewed by clicking on the command "Issue results" or in detail by clicking on the command "Layer investigation". The

files that are displayed are also the ones that the program generates for creating the calculation book.



If you want to change the cross-section of a member, then click on the dimensioning table, go to the desktop and left-click on the member you want to change. Then the Properties list appears on the right of the screen, where the "more" command displays the window with the properties of the linear members. To change the cross-section of the column, it is sufficient to click "cross-section" and define a new one.



The cross-section check will be done based on the new cross-section but with the same intensive sizes if you do not run the analysis scenario again.

6.3.2 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:

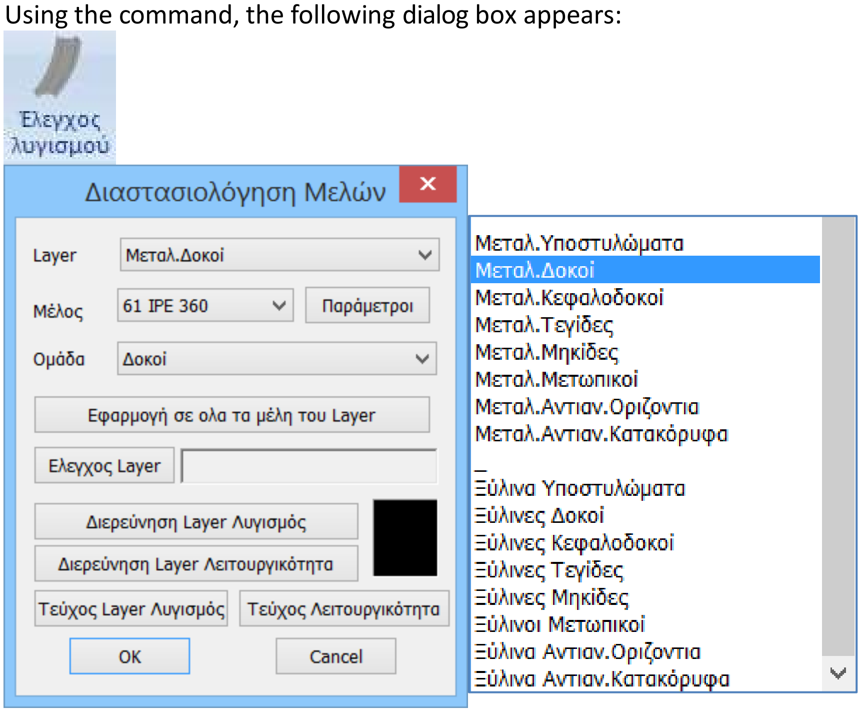
Limit State of Failure

- 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

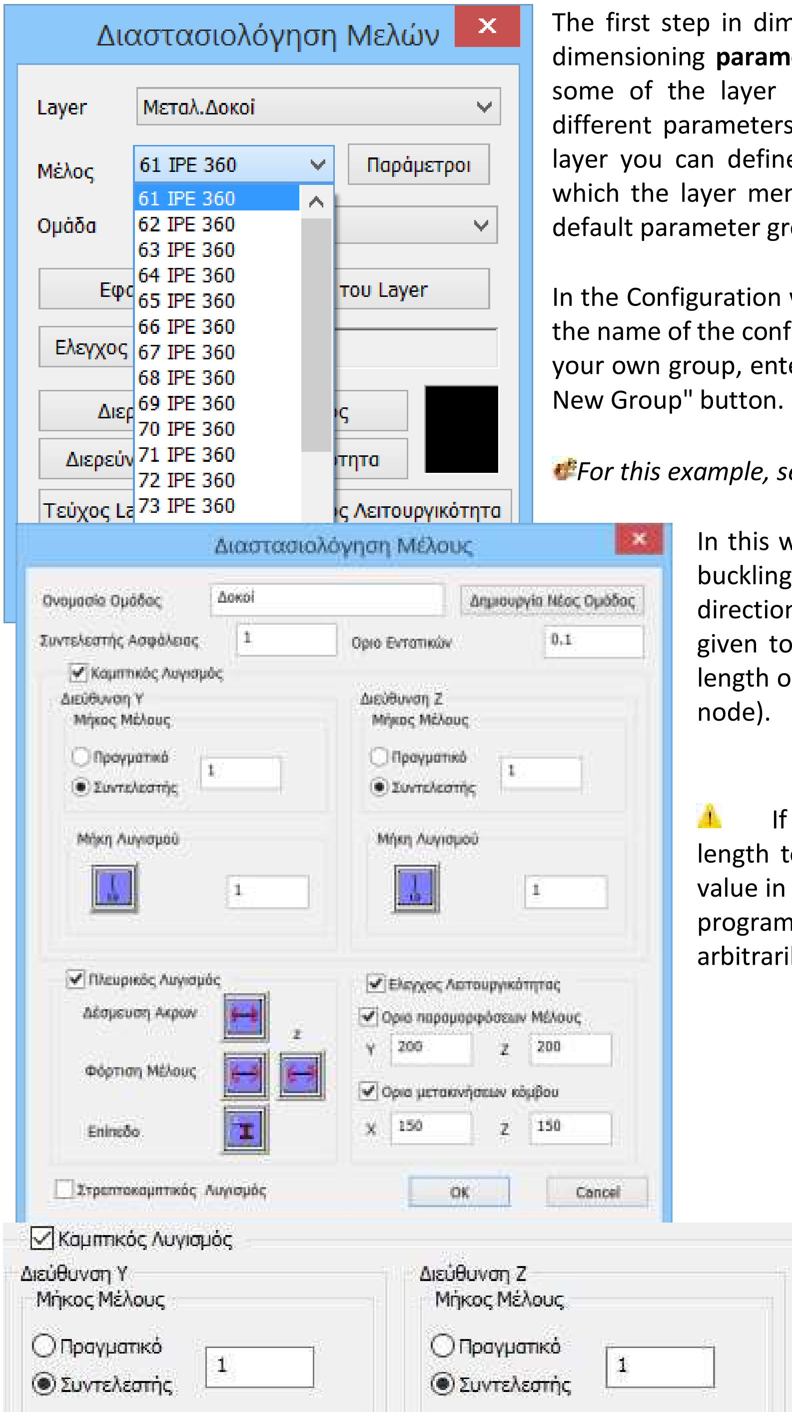
Using the command, the following dialog box appears:



The check is done per layer. So first select from the list

By selecting the layer, all the members of this layer and their cross-section are displayed in the "Member" list.

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'



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

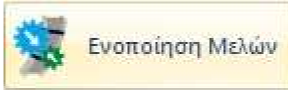
In the Configuration window, in the "Group Name" field is the name of the configuration group. If you want to create your own group, enter a new name and press the "Create New Group" button.

For this example, select the parameters shown below:

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

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

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



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

In "**Length of Membership**":

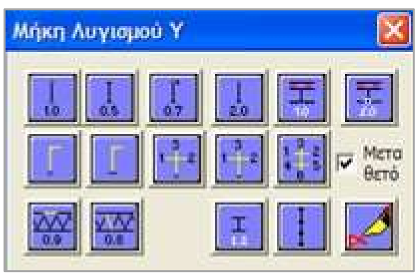
- if you select "*Actual*" you must enter in the field the actual length of the member in m.
- if you select "*Coefficient*" you must enter a coefficient by which the different lengths of the members belonging to this parameter group will be multiplied.

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

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

⚠ **OBSERVATION**

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



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

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

⚠ *For a detailed explanation of the icons please refer to the corresponding paragraph of the Manual Chapter 10c Sizing.*

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

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

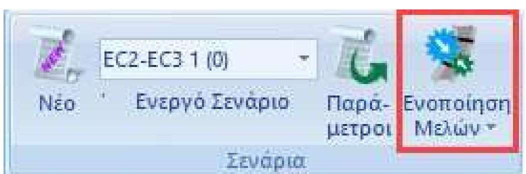
After you have entered all the parameters, you are returned to the previous window. Here if you select the command "**Apply to all members of the layer**" then the parameters you set before for member 61 will be applied to all members of the corresponding layer, i.e. the metal beams.

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

Then you select the "**Layer check**" command and the calculation of all the members for the combinations you have already defined starts and at the end a green or red square appears where 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 summarized by clicking on "Buckling Layer Investigation" or in detail by clicking on "Buckling Layer Investigation". The files displayed are also the ones generated by the program to create the calculation issue. The same applies to the functionality checks.

6.3.2.1 Consolidation of Members

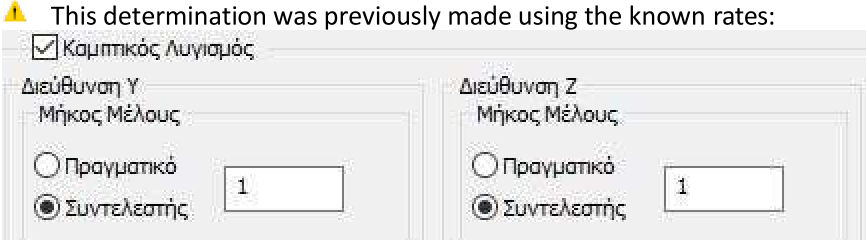


In the new version of the program a new group of commands has been added which concerns the consolidation of metallic members for the calculation and display of the bending and deformation checks based on EC3*. **See User Manual § 10.c Sizing).***

IMPORTANT NOTES:

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

- This determination was previously made using the known rates:

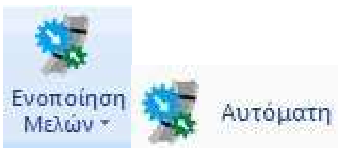


- Now with the use of the consolidation by direction, the process of the rates will not be needed, but the consolidation will be done, in most cases automatically.
- It should also be noted that with the consolidation process the bending length is correctly calculated and in the printing of the results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

NOTE

As a general rule, we could say that, 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 select the Automatic command:



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

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

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

Finally, in automatic consolidation, there is the definition of <u>stop levels.</u>

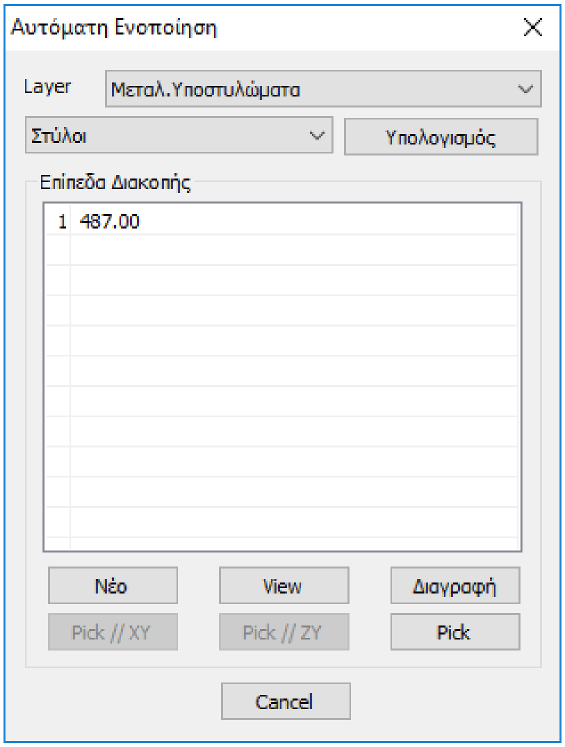
<u>Break planes</u> are horizontal or vertical planes that are used as breakpoints in the consolidation of a continuous element.
Thus, for vertical elements (Pillars) the stopping levels are horizontal levels which are defined, like the levels, with an altitude.

OBSERVATION

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

6.3.2.2 Automatic Consolidation

Using this command displays the following dialog box



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

members are displayed in colors on the screen.

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

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

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

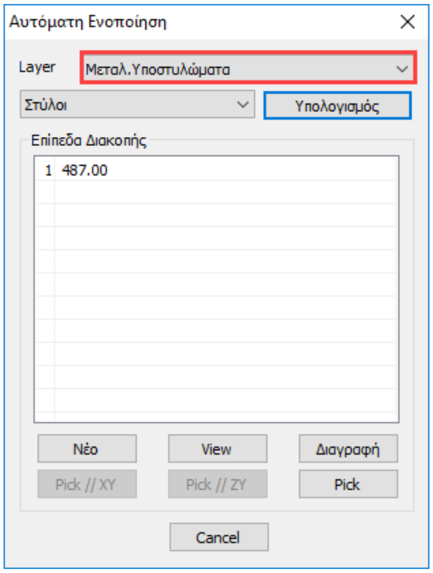
Interruption levels are levels that are boundaries of the beams and poles where you want the integration for either one or the other direction to be interrupted.

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

Predefined limits:

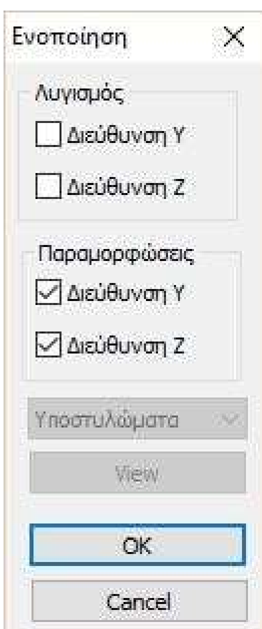
- for the horizontal planes the horizontal planes are the foundation level and the upper last level (the last level).
- and for the beams are the vertical limits of the girder.

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



6.3.2.3 User Consolidation

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



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

(See User Manual § 10.c Sizing).

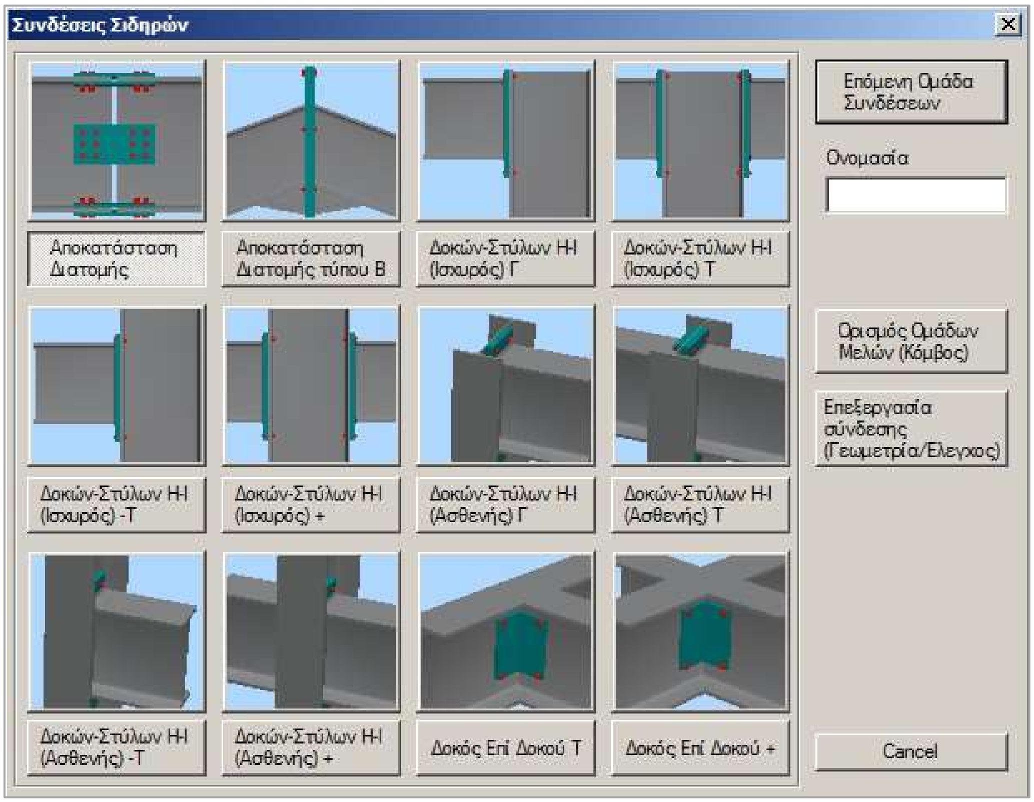
7. Connection sizing

7.1 How to size the connections of the metal members

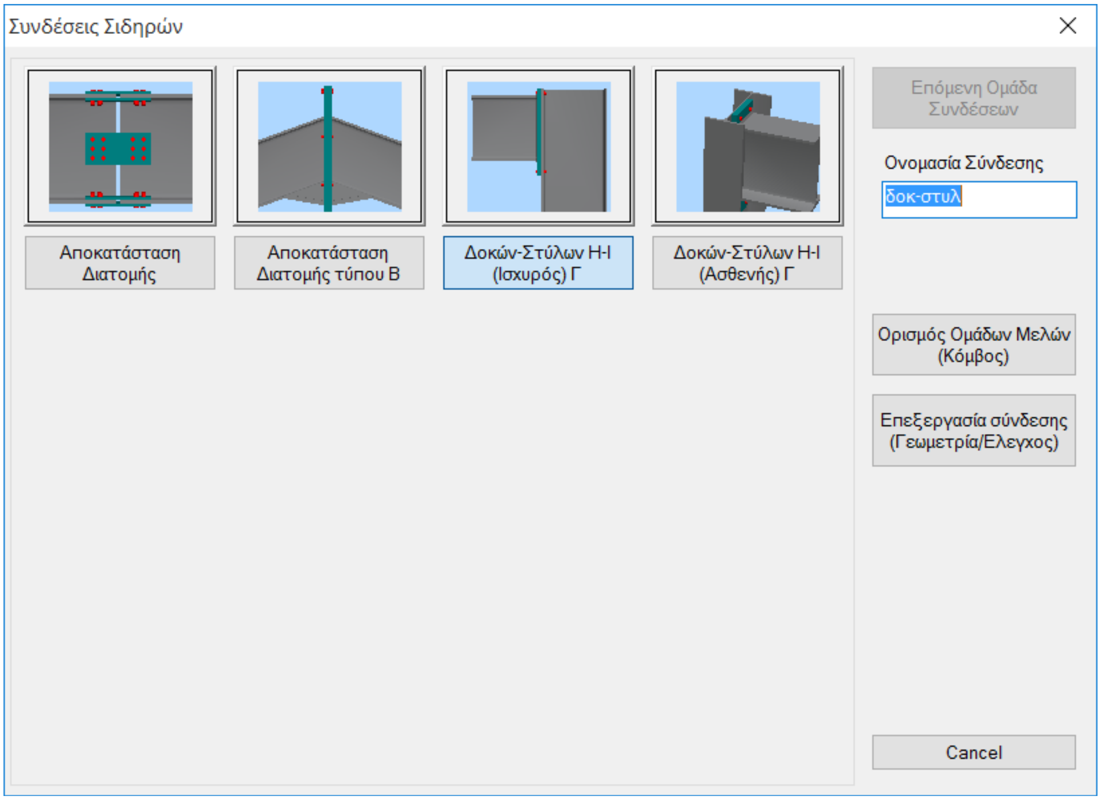
The last chapter of dimensioning for steel structures is the dimensioning of the connections of the structure. Select the command and you have two options to proceed with the sizing of the connections:

A) Click on "Connections 

and then right-clicking on the site (desktop) displays the library with all the available connections
from where you can choose the one you want.



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



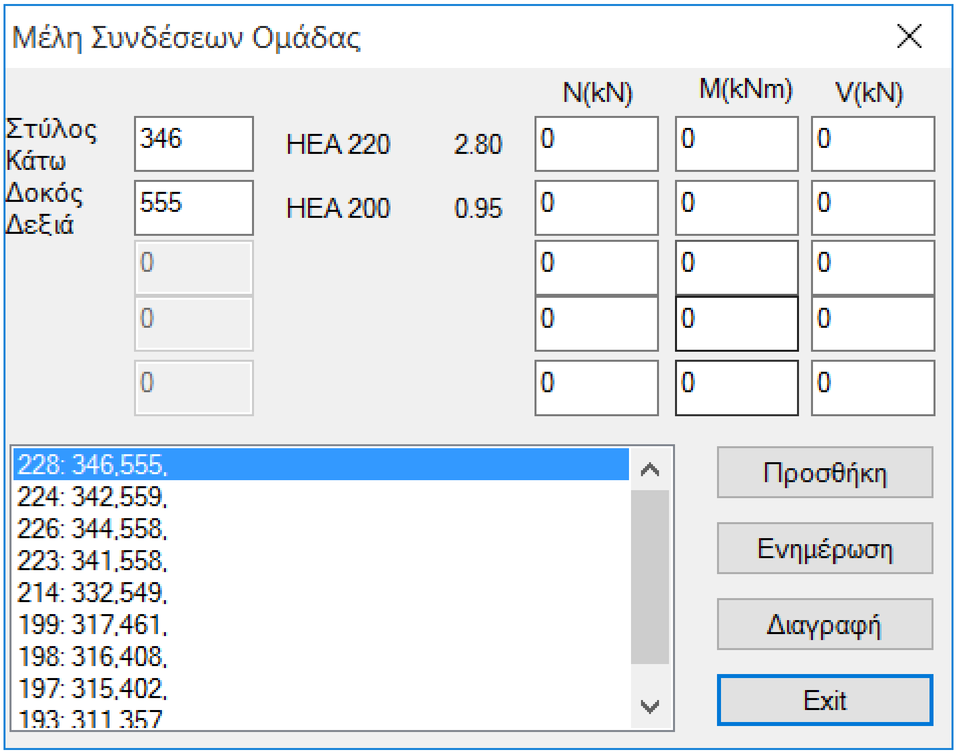
For example, select member 30 (column) and member 154 (beam) in sequence. Right-click to display the window with the 4 possible connection types. Select the last (to the right) connection which corresponds to a Beam - Column connection of cross-section type H or I on the weak axis. You will then enter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

Attention:

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

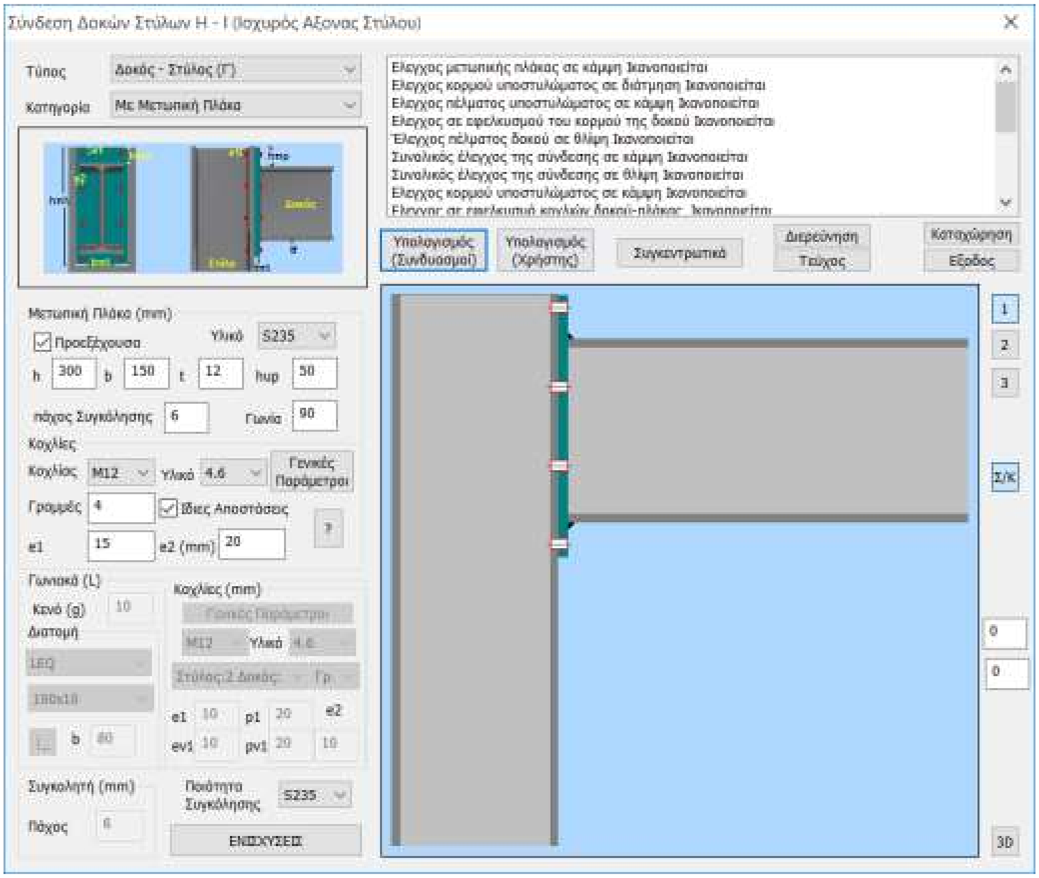
Then select the "Define member groups" command and in the dialog box you can add other similar pairs of cross-sections (column - beam) or add your own values for the intensive sizes N,M,V to the existing pair. To add other similar pairs, click on the "Column Bottom" field and then select Subcolumn 24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

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'



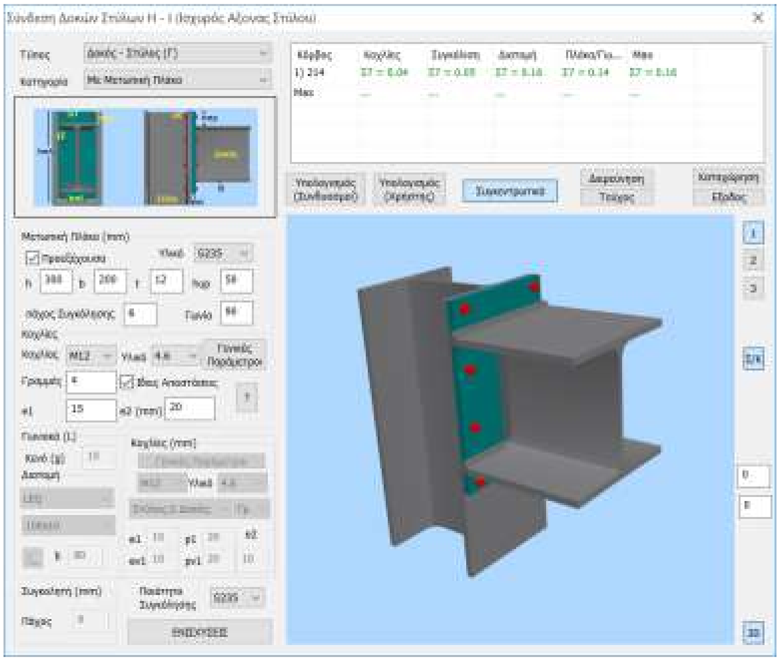
Essentially way, you can massively size all of the connections of the beam members of the girder which are connected to the weak axis in the same way (bolts or welds, plate geometry, etc.) and which have common cross-sections (IPE 450 - IPE 330 beam). The program will automatically calculate the intensive sizes of each pair and proceed to the dimensioning of the connection with based on the worst combination. This way you will not have to guess where in your structure the worst beam-post connection on the weak axis will be developed, while at the same time, if one connection is satisfiedall 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 set the number of the the type and geometry of the specific connection. Give the characteristic values shown in the figure or try to create your own connection.
To then check the adequacy of the connection with the combinations of 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 14 to 15 cm and click again on "Calculation (Combinations)".

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'



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

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'



When the checks of the geometry and topology of the connection are satisfied, the program will make all the required calculations and display all the checks performed according to Eurocode 3 for the specific connection. A summary of the results can be seen in the corresponding field. There, sufficient reasons will be displayed in green font and the connection failures in red font. If all the checks are satisfied, the program will be able to proceed with the registration of the connection as well as 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 perform a total sizing of the level pedestals. Select the command and all the level pedestals are sized.
The node of the pedestal, depending on the type of failure, shall be painted in the corresponding colour according to the following

 The skirt was sized and armed without any problems.

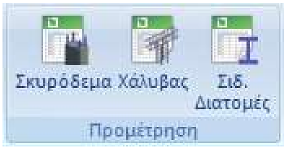
 The skirt missed. The type of failure is also indicated as a symbol above the indication of failure. The indications of failure are respectively the letter "Z" which means failure at limit load, the letter "e" which means failure due to load eccentricity and the letter "s" which means exceeding of the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

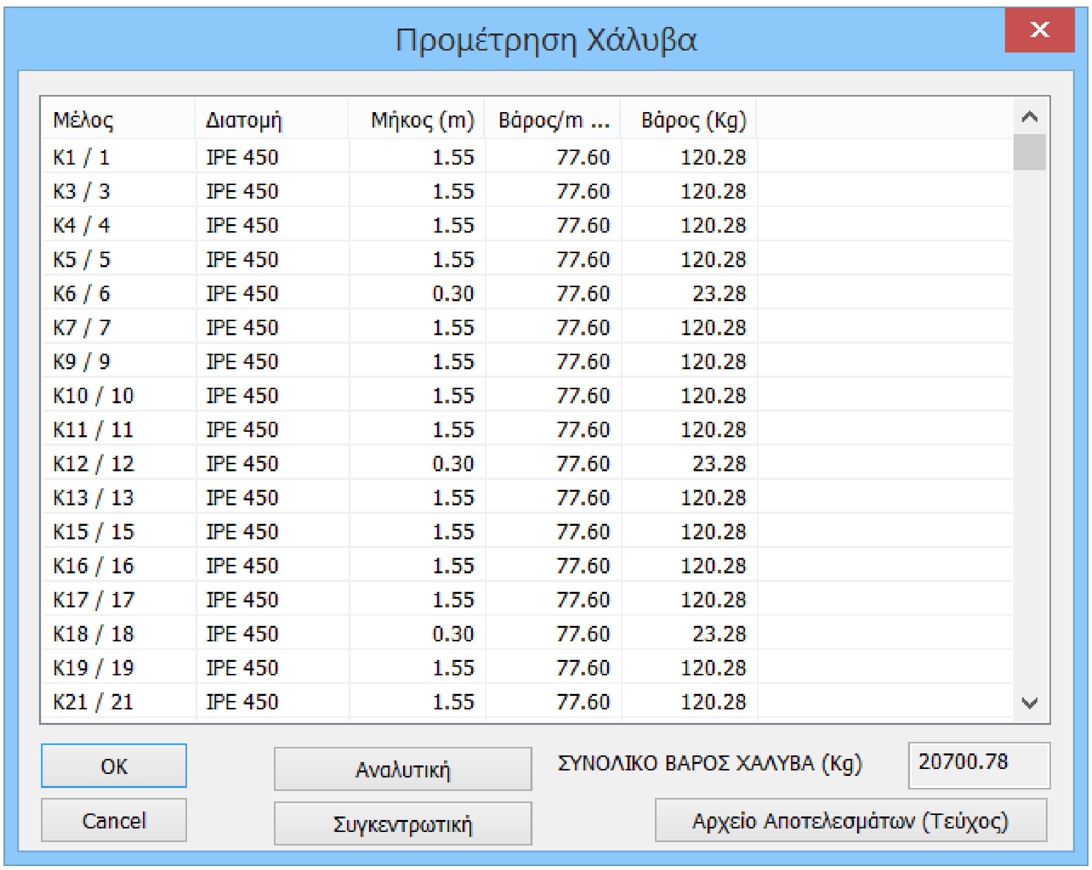
As is well known, seismic intensities derived from dynamic analysis are unlabeled because they result from the superposition of the eigenmodal responses. In the diagrams and wherever there is a necessity to superimpose them, they are always used with positive values. And for the dimensioning of the elements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



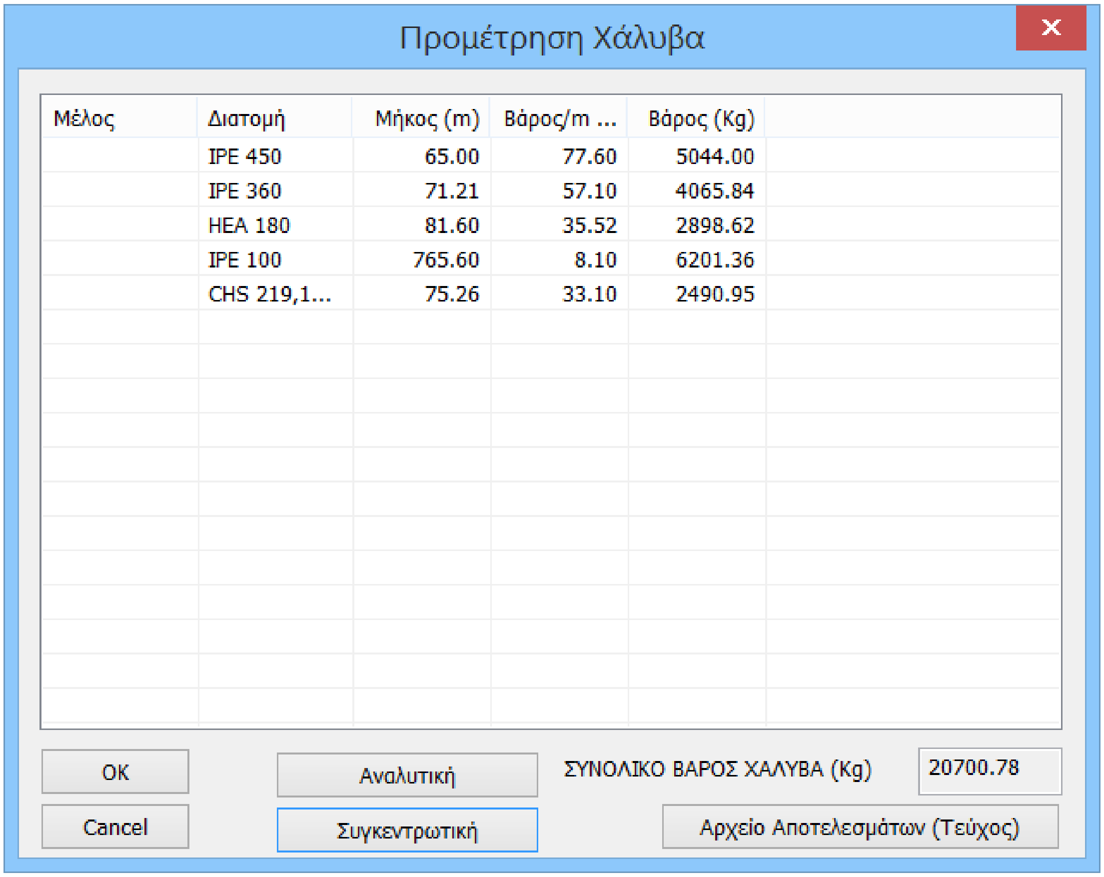
In the "Extras" section the "Premeasurement" group contains the commands for the premeasurement of the design materials.

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 ...	Βάρος (Kg)
K1 / 1	IPE 450	1.55	77.60	120.28
K3 / 3	IPE 450	1.55	77.60	120.28
K4 / 4	IPE 450	1.55	77.60	120.28
K5 / 5	IPE 450	1.55	77.60	120.28
K6 / 6	IPE 450	0.30	77.60	23.28
K7 / 7	IPE 450	1.55	77.60	120.28
K9 / 9	IPE 450	1.55	77.60	120.28
K10 / 10	IPE 450	1.55	77.60	120.28
K11 / 11	IPE 450	1.55	77.60	120.28
K12 / 12	IPE 450	0.30	77.60	23.28
K13 / 13	IPE 450	1.55	77.60	120.28
K15 / 15	IPE 450	1.55	77.60	120.28
K16 / 16	IPE 450	1.55	77.60	120.28
K17 / 17	IPE 450	1.55	77.60	120.28
K18 / 18	IPE 450	0.30	77.60	23.28
K19 / 19	IPE 450	1.55	77.60	120.28
K21 / 21	IPE 450	1.55	77.60	120.28

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'



Μέλος	Διατομή	Μήκος (m)	Βάρος/m ...	Βάρος (Kg)
	IPE 450	65.00	77.60	5044.00
	IPE 360	71.21	57.10	4065.84
	HEA 180	81.60	35.52	2898.62
	IPE 100	765.60	8.10	6201.36
	CHS 219,1...	75.26	33.10	2490.95

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

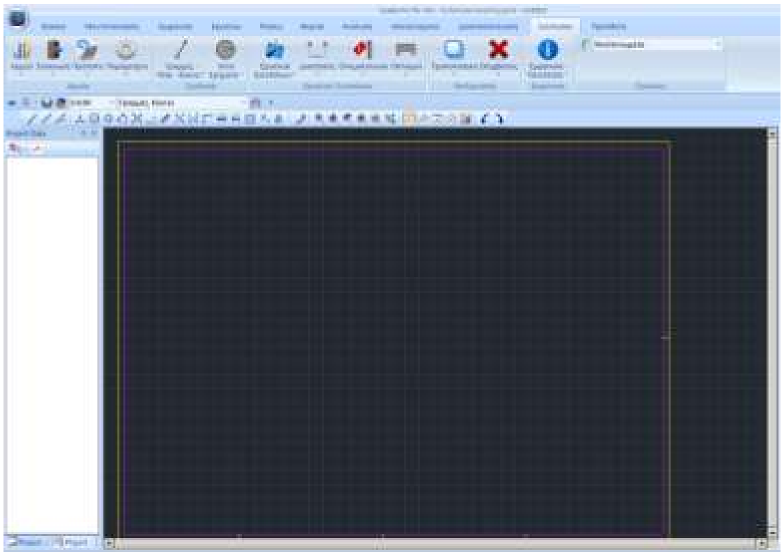
For example, as shown in the table, the total weight of IPE 450 cross-sections is 5044,00kg. Correspondingly the total steel weight of the whole structure is 20700,78kg.

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".

10. DESIGN

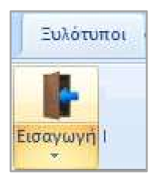
After completing the dimensioning of the carrier and the creation of the connections for the metallic ones, in the Wooden Formwork Module you enter, modify and finally create the drawings of the formwork and its details.

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

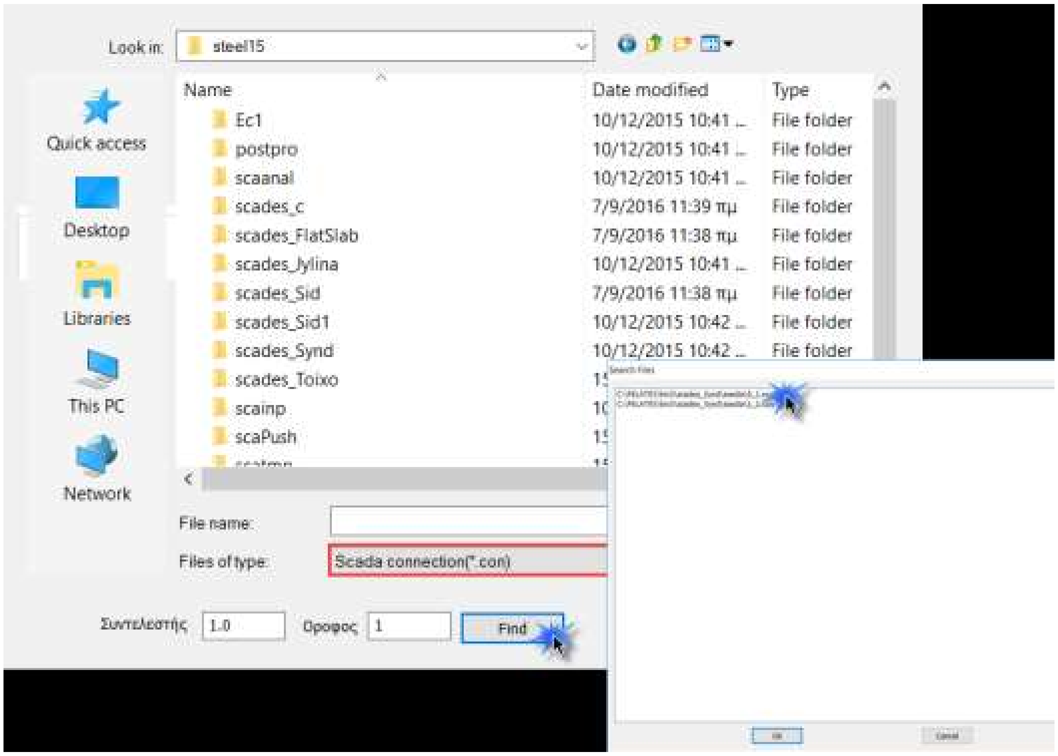


10.1 How to enter the plans of the links:

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 2: 'DESIGN OF A METAL STRUCTURE'

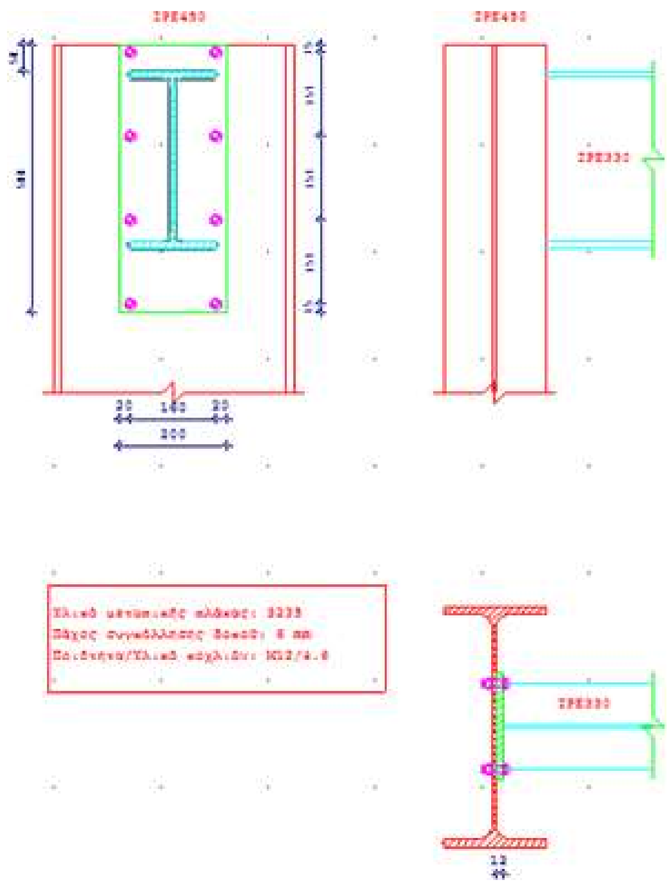


In the dialog box:

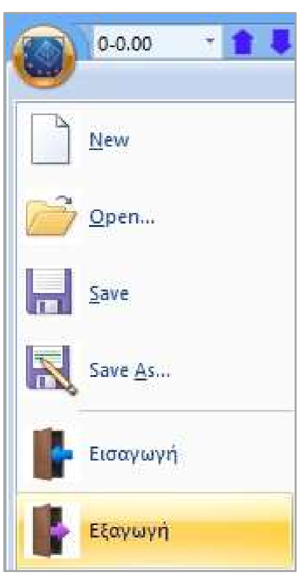
- in Files of Type select **Scada Connection(*.con)**

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

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'

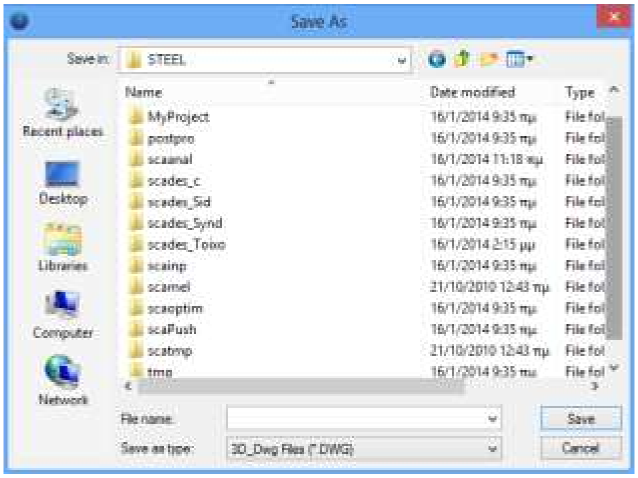


By following the above procedure you can produce over 120 different types of connections covered by the program.
To create corresponding faces, plan views and sections of the overall vector you will to follow a different way.



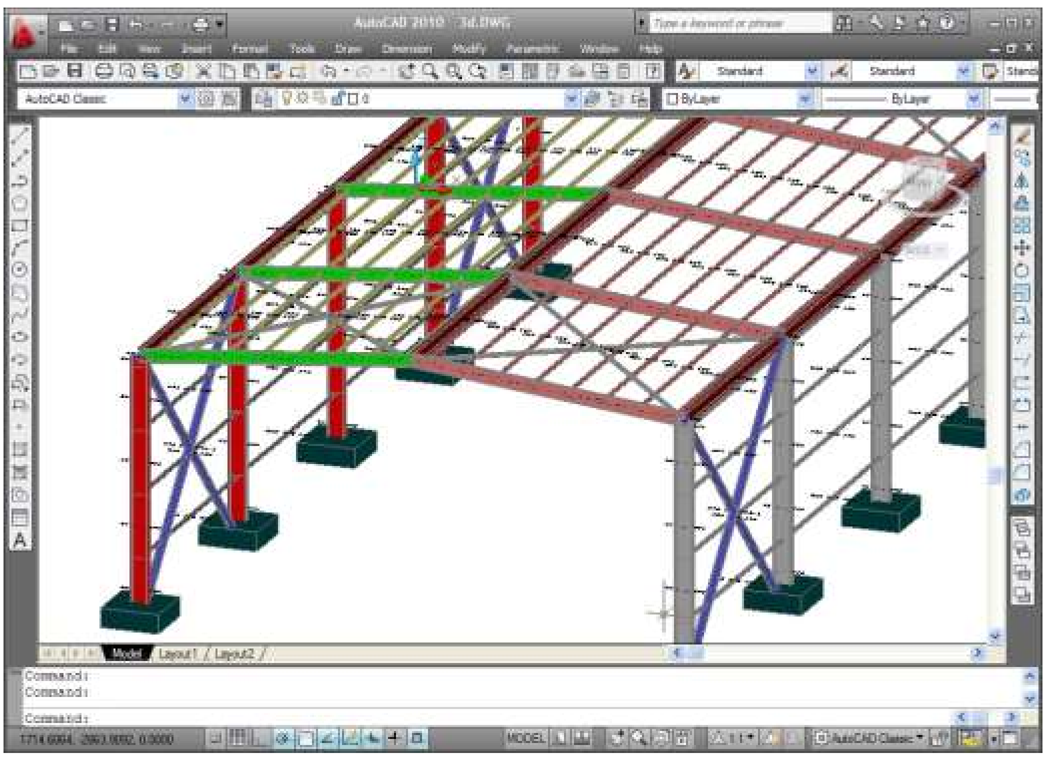
So, you should click on the command "Export" which opens a new window through which you can export the SCADA Pro file to an autocad *.dwg file. In the "Save As" field you select your design folder to export a 3D version your structure. To do this you type a name in the File name and then in the "save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

EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'



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

.



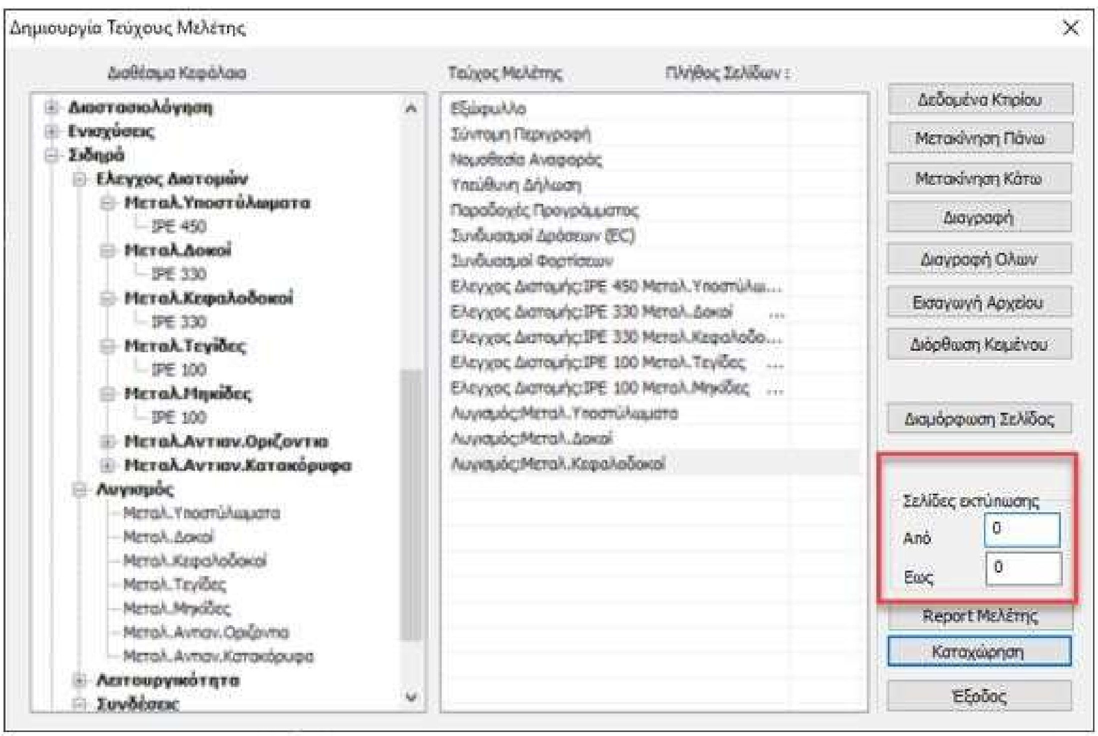
11. COPY

11.1 How to create the study issue:

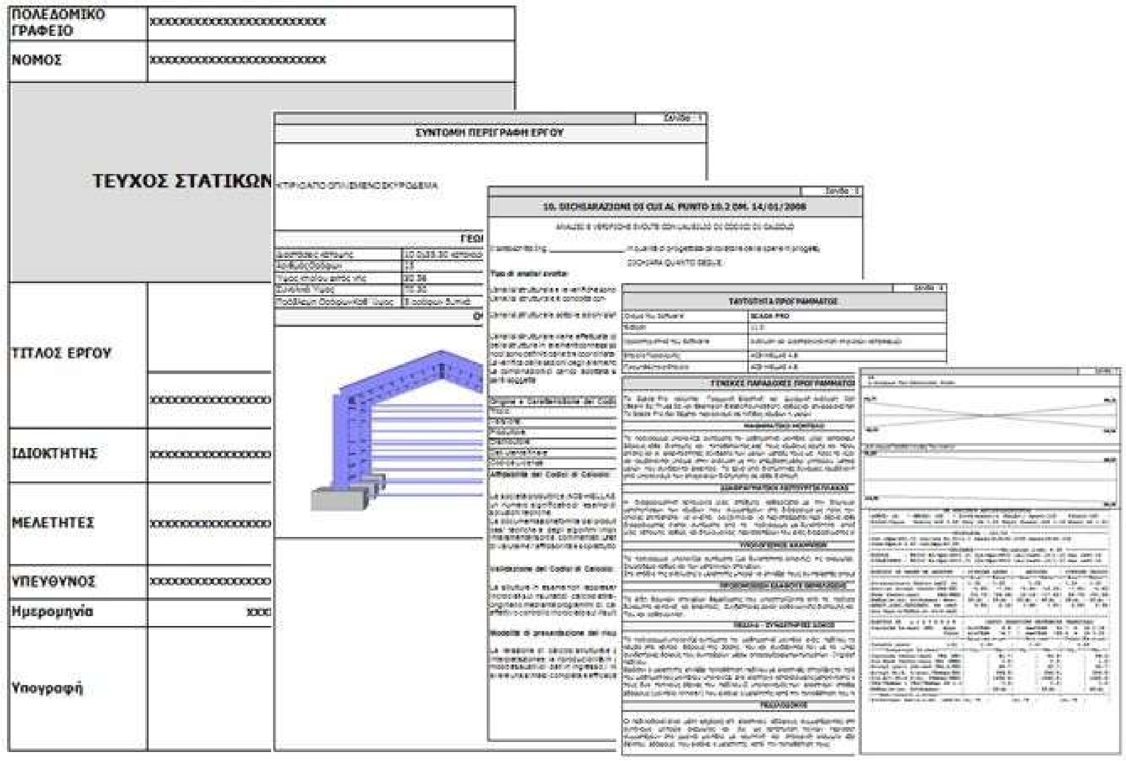
To create the study booklet, open the "Extras" section and select the Print command.
In the "Create Study Sheet" dialog box, the list of chapters available for printing appears on the left. The right list, with the chapters to be included in the booklet, is completed by selecting them from the left list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

For this example select the chapters you want to include and press "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 issue have been redesigned and implemented with modern tools to provide you with a new tabular, easy-to-read study issue with the addition of charts and images. You also now have a full preview of your issue as well as the ability to export and edit the file in more than ten different file formats including pdf, docx, rtf, xml, CSV, PowerPoint, etc.

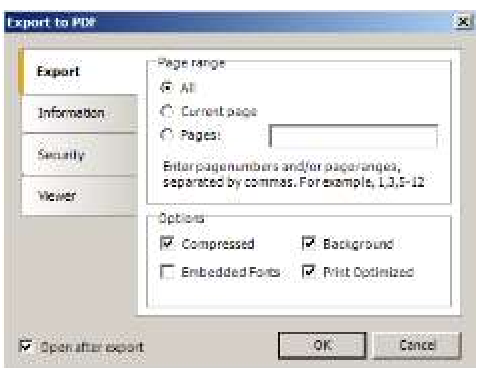
In addition, the ability to "break" the study book into individual sections has been added, a useful and practical feature especially for the easy management of multi-page studies.

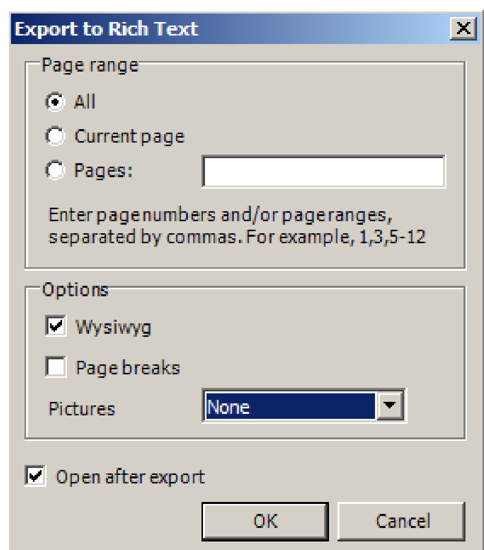


EXAMPLE 2: 'DESIGN OF A METAL STRUCTURE'



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





Through this simple example, you were able to experience just a few of 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 even the most complex metal structure.