SCADA Pro™
Structural Analysis & Design

Example 2
Steel Structure using Templates Analysis and Design

ACE HELLAS
INTEGRATED SOLUTIONS

Contents

- ## OVERVIEW

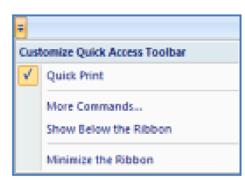
SCADA Pro new version is a result of more than 40 years of research and development while containing all the innovative capabilities and top-notch tools for the construction business.
SCADA Pro utilizes a compact and fully adequate platform for constructing new buildings (analysis and design) or existing ones (check, assessment, and retrofitting).
The software employs the Finite Element Method, combining line and plane finite elements in a smooth way. For design purposes, the user is offered all the Eurocodes as well as all the relevant Greek regulations (N.E.A.K, N.K.O.S., E.K.O.S. 2000, E.A.K. 2000, E.A.K. 2003, Old Antiseismic, Method of permissible stresses, KAN.EPE).
There are numerous possibilities offered for the modeling of various kind of structures. Structures made of reinforced concrete, steel, timber, masonry, or composite structures are now fully feasible.
Several smart operations add on to the practicality and usability of the software. The user can produce the model of a structure no matter how complicated it is, work at ease with the 3D model, process through the steps of analysis and design in a convenient way, up to the conclusion of what initially may seem the most demanding project.
SCADA Pro is presented to you as a powerful tool to meet the highest needs of modern civil engineering!

- ## INTRODUCTION

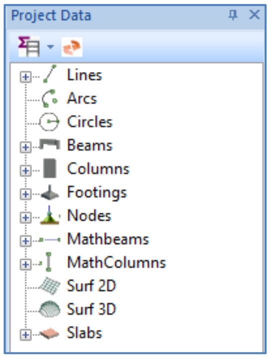
The current manual comes as an aid for a new user of SCADA Pro, making the interface of the software as familiar as possible. It consists of several chapters, where one after the other, describes the consecutive steps of a simple example of a loadbearing masonry project.
The most useful information is presented, in regards to the best possible understanding of the software commands and logic, as well as the process that has to be followed.

- ## THE NEW INTERFACE

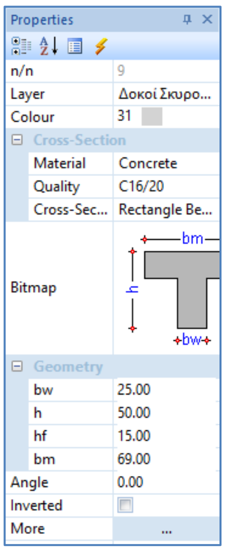
The new interface of the SCADA Pro software is based on the RIBBON structure, thus, the several commands and tools are reached neatly. The main idea of the RIBBON structure is the grouping of commands that have small differences and work in the same context, in a prominent position different to each group. This converts the use of a command, from a tedious searching procedure through menus and toolbars, into an easy to remember the chain of two or three clicks of the mouse button.



The user can collect his/her most popular commands into a new group, for an even faster access. This group remains as it is for future analyses after the program ends. Different commands can be added to it or removed from it, and its placing in the workspace may be altered through the "Customize Quick Access Toolbar" utility.



Apart from the RIBBON structure, all the entities that a structure consists of are presented in a tree structure, at the left side of the SCADA Pro main window, either for the whole structure or at each level of the structure. This categorization enhances the use of each entity. When an entity is being chosen by the tree structure, it is highlighted at the graphical interface and the level of the structure that contains this entity is isolated. At the same time, at the right side of the window, the entity's properties appear. The user can check or modify any of these properties at once. Conversely, the entity can also be chosen at the graphical interface, and automatically it is presented, at the left side in the tree structure and at the right side with its properties. The right-click mouse button can be very helpful here, since several commands and features, distinct for each entity, can be activated with it.

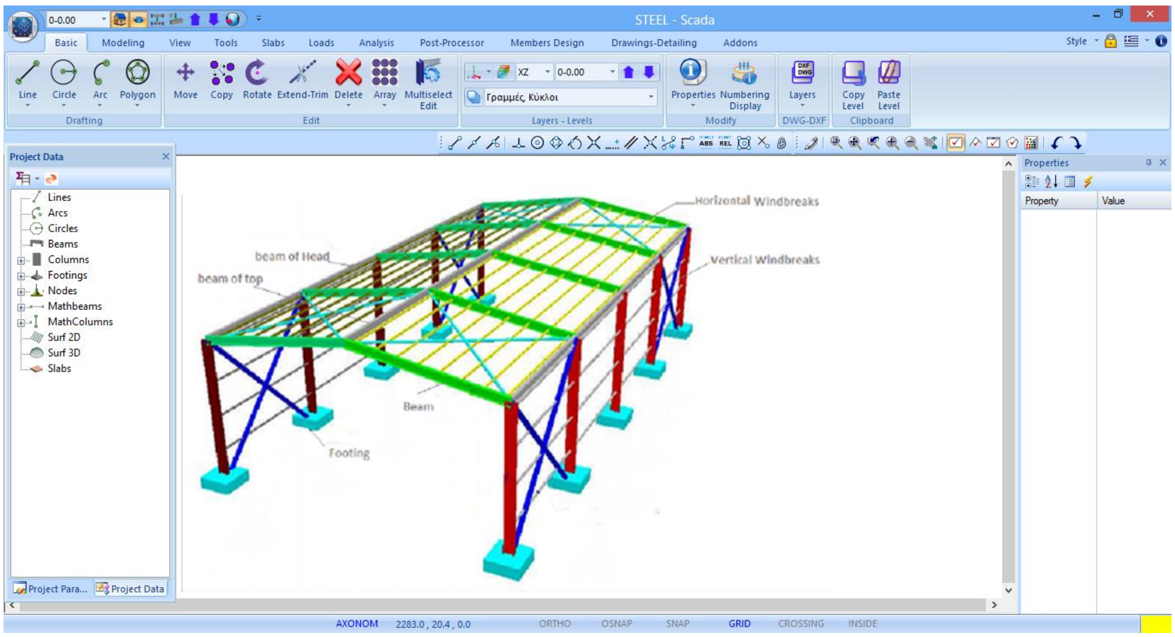


The "Properties" list that shows up at the right side of the window, not only shows all the properties of the entity shown but can be used for any quick and easy changes, the user wants to make, too.

1. GENERAL DESCRIPTION

1.1. Geometry

This steel structure example contains five frames whit one opening (7,05 +7,05 = 14.10m). The frames distance 6.80m. The roof has two gabled with slopes 5.33°. The height of ridge is 7.50m. The height of the columns is 6.50m. The foundation is formed by individual footings of concrete in both directions. For the full geometry, see the picture below.



1.2. Materials

All steel elements use steel quality S275(Fe430). The elastic modulus is E =21000kN/cm2 and Poison ratio is v=0.30. The steel density is 78,5 kN/m3.

1.3. Regulations

Eurocode EC0, ENV 1990 for the loadings.
Eurocode 3 EC3 ENV 1993 for steel elements design.
Eurocode 8 (EC8, EN1998) for earthquake loads.
Eurocode 2 (EC2, EN1992) for foundation design.

1.4. Profiles

Columns :	IPE 450
Beams:	IPE 360
Vertical windbreak:	CHS219.1/6.3
Horizontal windbreak:	CHS114.3/5.0
Main Beams:	HEA180
Purlin:	IPE100
Girders:	IPE 100

1.5. Acceptances Loading-Analysis.

The loadings according to the Dynamic method are :
G (Fixed loadings)
Q (movable loadings)
Ex (Loadings of Earthquake on nodes about XI axis from Dynamic Analysis)
Ez (Loadings of Earthquake on nodes about ZII axis from Dynamic Analysis)
Erx -(Loadings of Earthquake on nodes from Ex displaced about accidental eccentricity –ετzi)
Erx +(Loadings of Earthquake on nodes from Ex displaced about accidental eccentricity ετzi)
Erz -(Loadings of Earthquake on nodes from Ez displaced about accidental eccentricity –ετxi)
Erz +(Loadings of Earthquake on nodes from Ez displaced about accidental eccentricity ετxi)
EY (Vertical Earthquake loading from dynamic analysis)
In this case, we add and three loadings:
S (Snow)
W0 (wind about X axis)
W90 (wind about Y-axis)

In seismic analysis involved only dead and live loads. Snow and wind loads are considered in separate "simple" static analysis scenario (see Analysis).
The values of snow and wind loads in this example will be taken arbitrarily without accurate calculation according to the Eurocode 1, for simplicity.
ψ0, ψ1, ψ2 action factors, will be according to EC0.

1.6. Information

All the commands that will be used in this example (in fact the whole group of the software commands), are analytically described and explained in the User's Manual of the software.

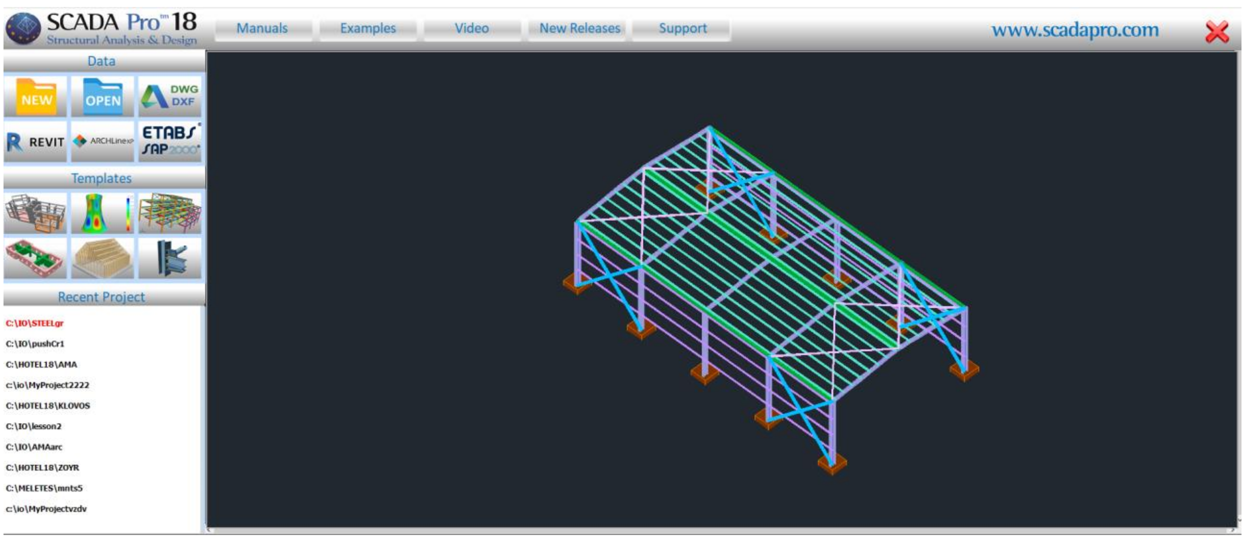
2. DATA INPUT – MODEL PRODUCTION

2.1. How to start a new project

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

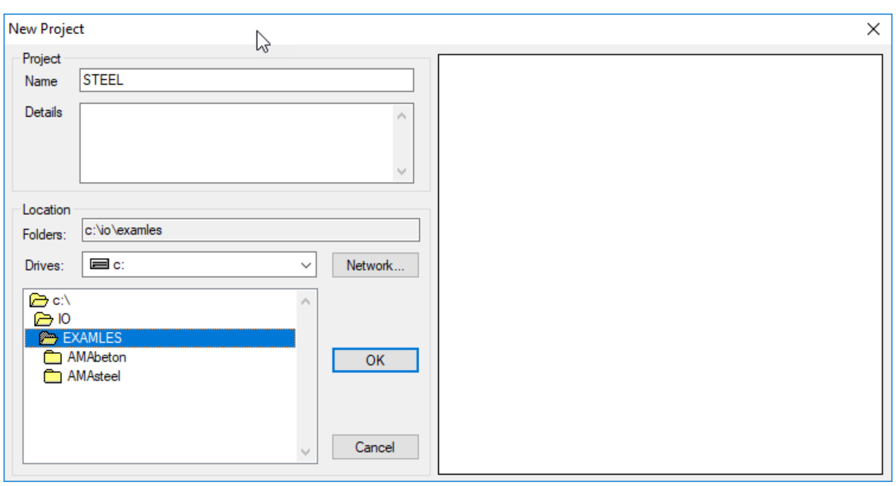
> ⚠ In this example, we will see in detail the use of the typical structures for modeling a steel structure.

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



By left clicking on the related icons, one of the following ways, to initialize a project, can be performed:

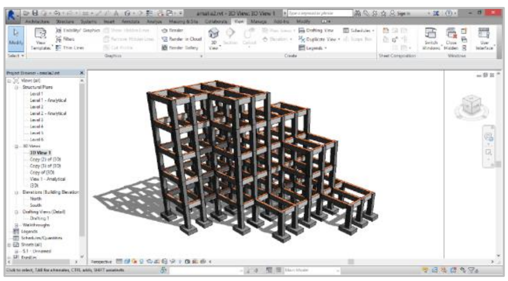
> ⚠ No matter which way you choose to start a new project, the same form always opens to set the project name and the path of the file, a necessary procedure so that the program commands can work.

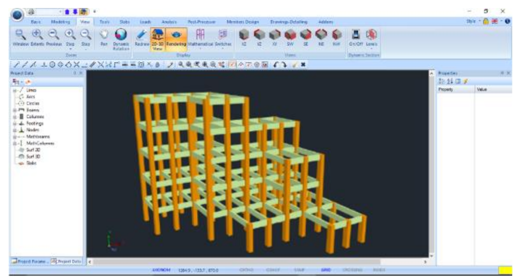


NOTE:
The name of the file can contain up to 8 characters of the Latin alphabet and/or numbers, without any symbols (/, -, _) or spaces.
You can add a description or add some information related to the structure, in the "Info" field.

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

"**REVIT**": Reading ifc files created by the Autodesk Revit.
By using appropriate libraries, SCADA Pro automatically recognizes all the structural elements (columns, beams, slabs, etc.) with their respective properties, generating in this way the ready for the analysis model.



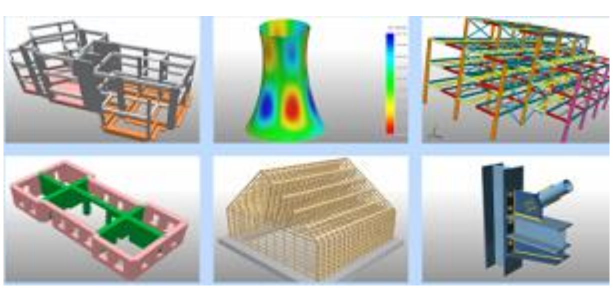


: Reading .xml files Read an .xml file from ARCHLine.XP architectural software.

: Import a cad file and use it as an auxiliary file into the interface or base for Automatic Level Creation and Automatic Section Identification.

- *A detailed description of the automatic procedure based on the .acad files is given in the concrete structure example.*

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

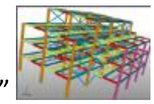


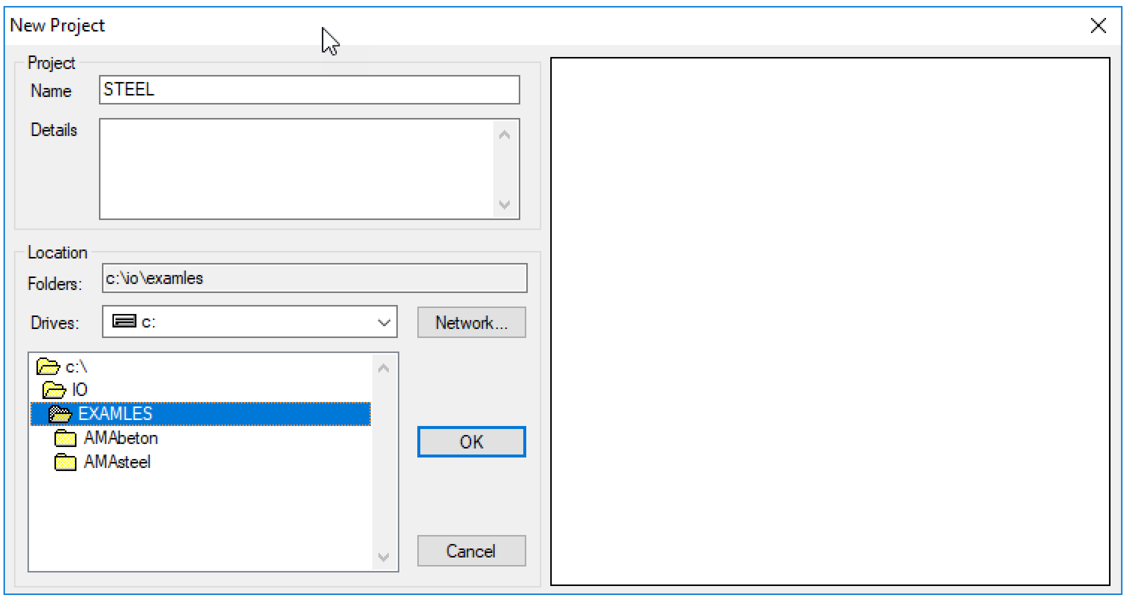
concrete   shell elements   steel

masonry   *timber*   *connections (IDEA statiCa)*

The most common steel structures contain continuous frames in one or both directions with duo pitch roof. Stringers, purlins, windbreakers, and front columns may be included. In case of using a template structure, you can perform the entire modeling with one single command! However, in case of more models that are complicated as well, the template command can set the bases to complete the entire modeling faster, just by modifying some of the automatically generated characteristics.

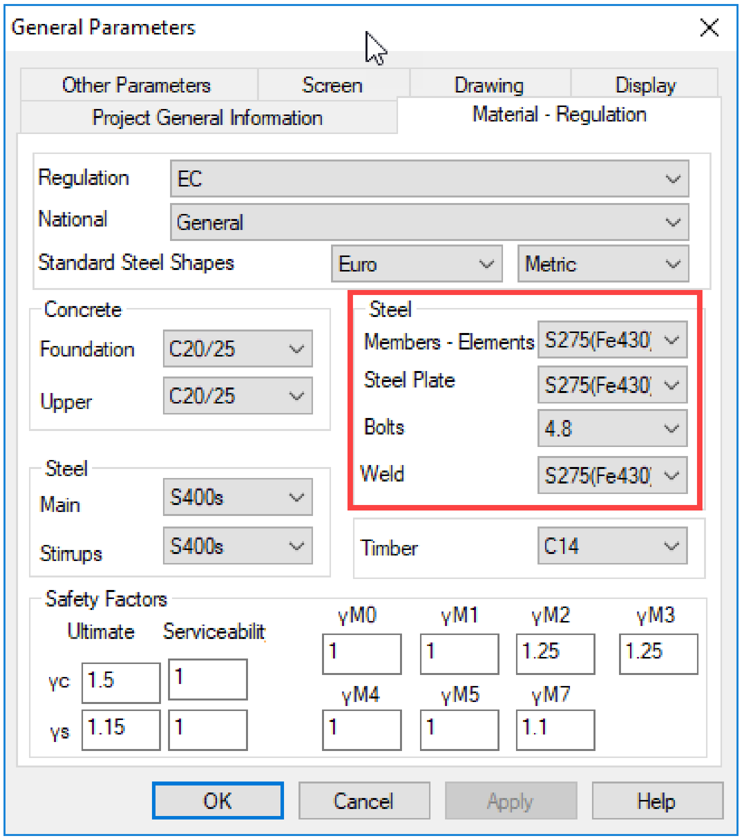
2.2. Templates – Steel Structures:

To start the example, press the startup icon, “Templates” 

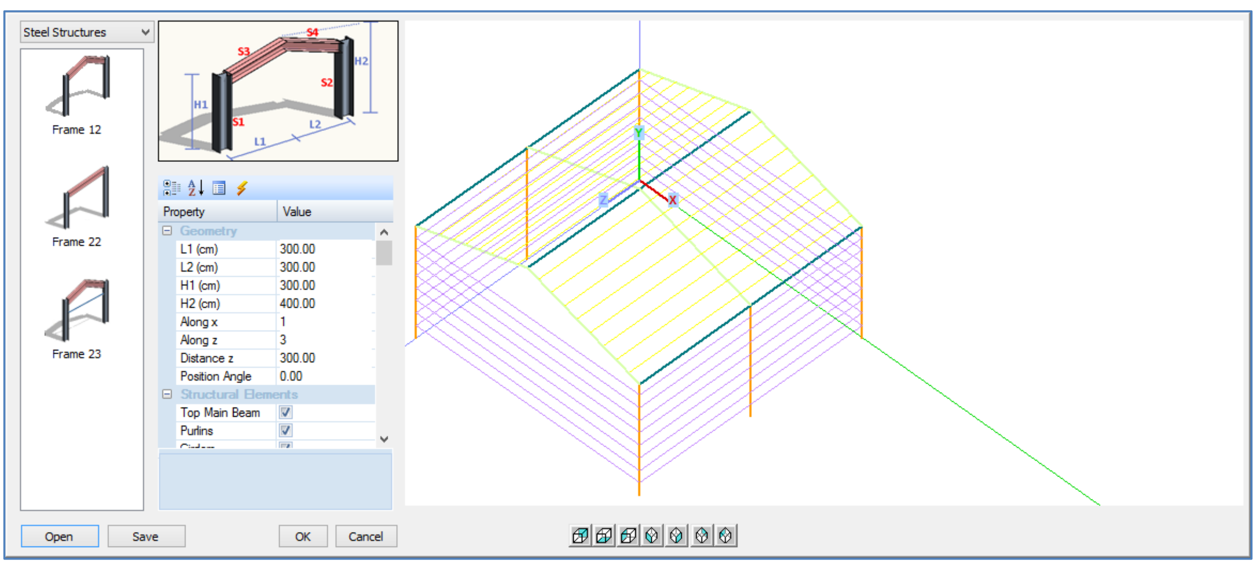


Set the “**Project**” name. If you wish, write in the “**Info**” field, some information related to the project and define the path that your project will be stored to, inside the local disk.

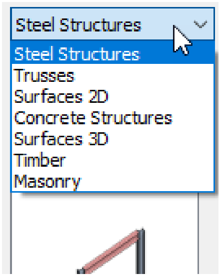
Automatically opens the General Parameters window, to set the parameters of the project, such as Material and Regulation, and other general parameters:



Press OK and automatically opens a new window containing all the geometrical properties of a steel structure, where you can set, geometry, cross sections and foundation in a single step.

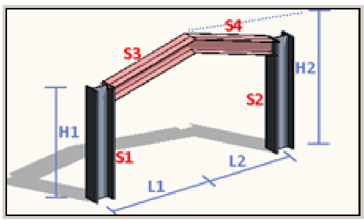


First, select the steel frame on the left. There are three different types of steel frames.



You could also select another type of structure: Trusses, Surfaces 2D-3D, Concrete Structure, Timber, Masonry.

Choose the frame and start typing the property details. Check the geometry in the Figure on the right.

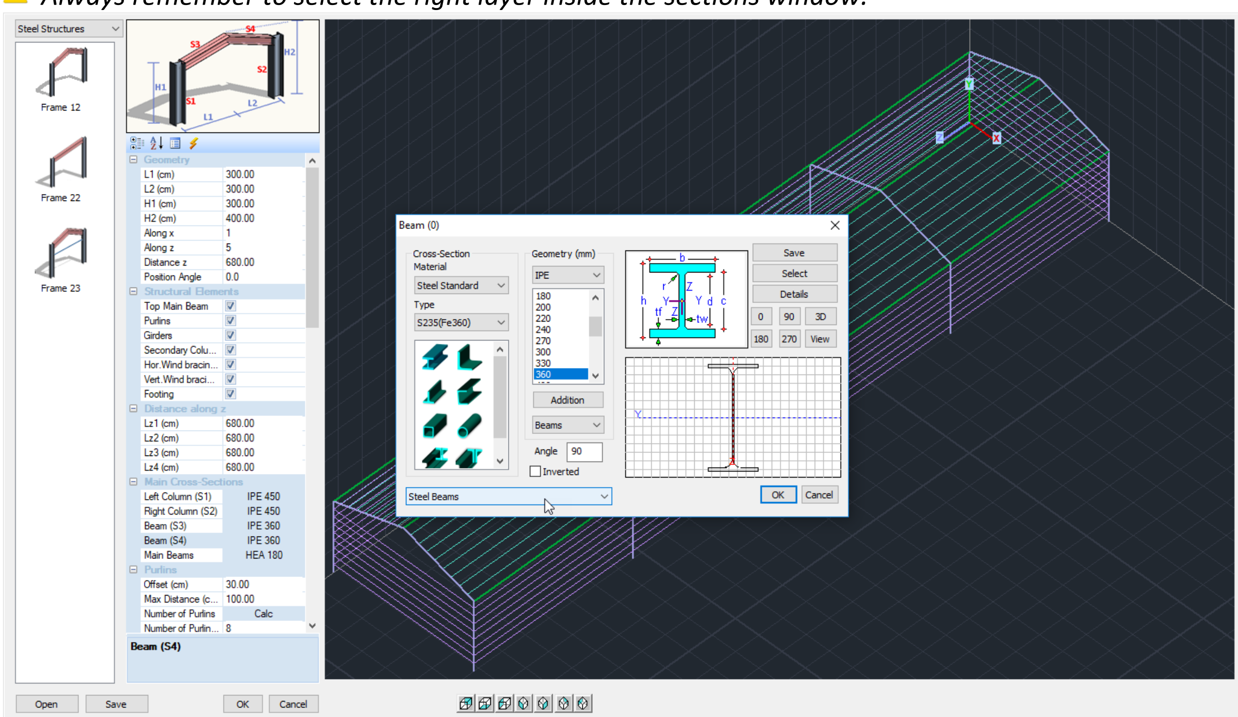


Set the parameters in the “Geometry” field according to the drawing and the repetitions in x and z-direction.

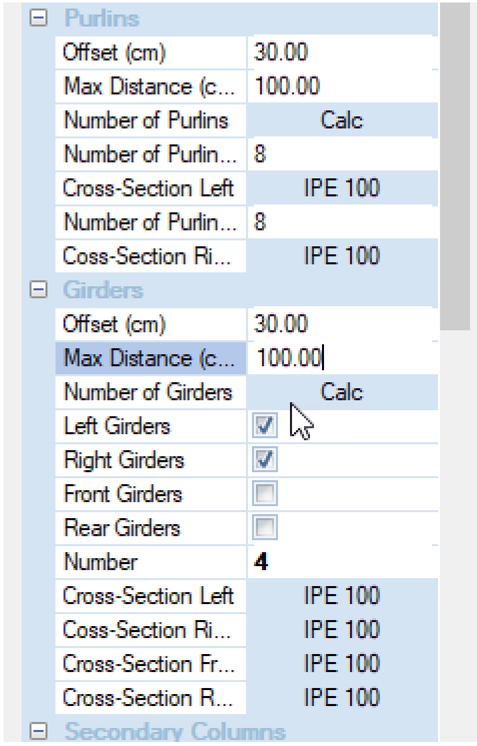
Type 1 repetition in x and 5 in the z-direction.
Type 680 cm distance, between the main frame structure.

Set the following sections:
Columns S1-S2: IPE450 with an angle of 90 degrees
Beams S3-S4: IPE360 with an angle of 90 degrees
Main Beams: IEA180 with an angle of 90 degrees

⚠ *Always remember to select the right layer inside the sections window.*



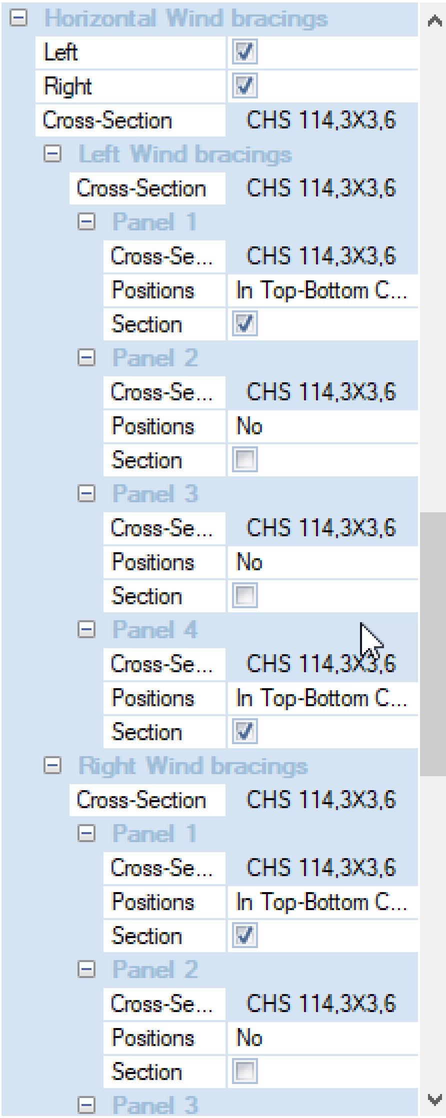
Then select the number, the position and the profiles of **purlins** and **girders**.



The offset indicates the distance will be the first purlin or girder from the beam of the head.
You can type the maximum distance between adjacent purlins or girders and press Calc. The program automatically calculates how many purlins or girders fit the beam or column.

In this example, select as offset 30 cm and type 8 purlins (IPE 100 with 90 °angle left and right) and four girders (IPE 100 with 0 °angle-left and right).

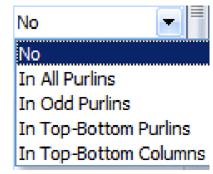
Notice that, in the Templates, the program automatically calculate the angle of the purlins, by calculating the angle of the frame. So, even if you type 90-degree angle to purlins, the program will calculate the exact angle and places it in 3d properly oriented.



For the horizontal Wind Bracing. So, the Wind Bracing will start from the node of columns and end nodes of ridge traversing the intermediate purlins without connecting each other.

Activate the horizontal wind bracings of the left or/and the right slope. The list expands further and for each panel, define the position and the intersection of the bracings.

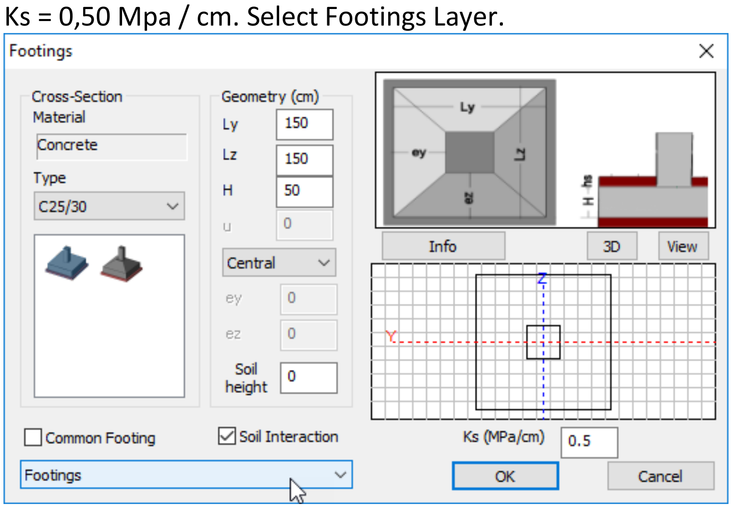
Choose the CHS114.3/5.0 section and place them in the first and last panel.
Choose the "Top-Bottom columns"



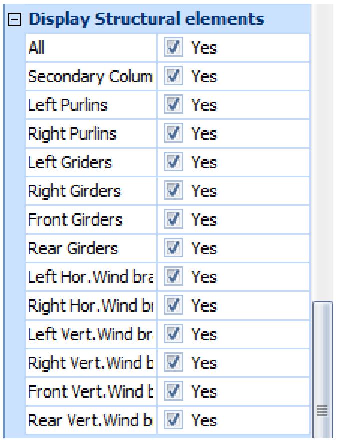
If you uncheck the "cut" the Wind Bracing will not split at the middle and you will have two cross elements. In the opposite, you will have four elements connected to a single node.

Follow the same procedure for the vertical wind bracings.
Choose CHS219.1/6.3 profile, left and right, on "Top-Bottom columns".

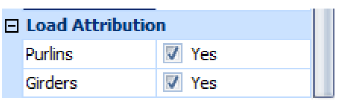
Finally, select the footing profiles (150cmx150cm rectangular footings). Check Soil Interaction and type Ks = 0,50 Mpa / cm. Select Footings Layer.



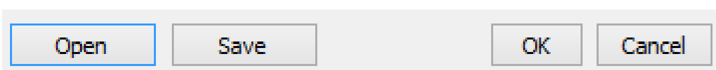
Select the elements for display.



On the right, you can see the model. Using the bar below you can change the view. You can also use left click to move and rotate the image.



“Load Attribution” regards the wind and snow loads according to Eurocode 1 (“Loads>>Wind-Snow Loads”). When the fields “Purlins” and “Girders” are active, the program automatically allocates the wind and snow loads on them.



The command “Save” is used to save the template. You can create a folder and save all your templates in it and make your template library to use them as they are or perform changes and use them in other projects.

The “OK” button close the Templates window and import the template model in SCADA Pro’s environment, in the virtual view.

ATTENTION:

When the template model enters in SCADA Pro’s environment cannot go back in the templates window.

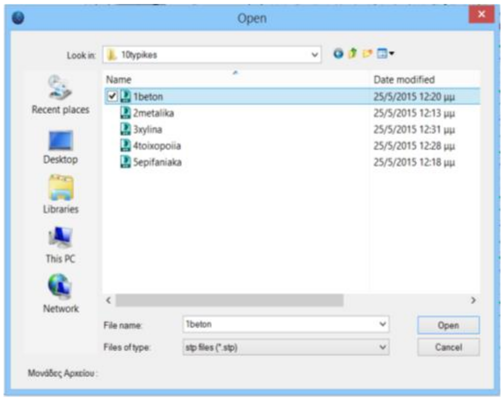
Switch off the virtual view to receive the 3D physical and mathematical model. Now you can work on the model using the appropriate tools (Manual “Basic”) and make all the changes you need to create the real model of your project.

You can also use more templates for the same project, with the same material or not. Just click on the input point, define the template, click “Save”, “Ok” and repeat for the second template.

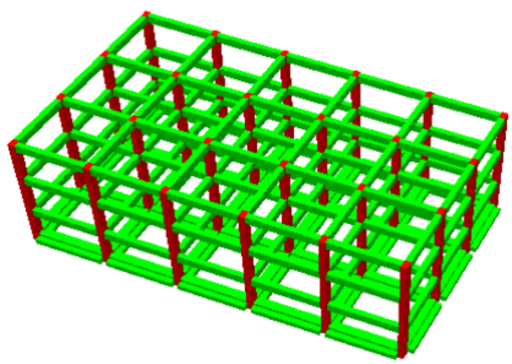
EXAMPLE

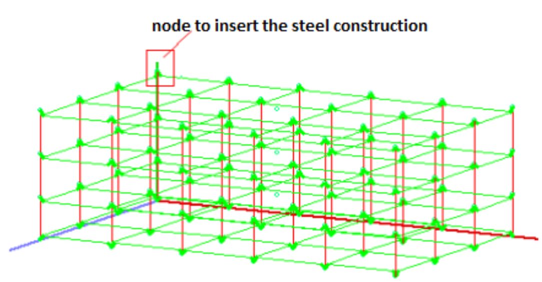
To insert a steel template over a concrete structure (already existing in SCADA Pro's environment):

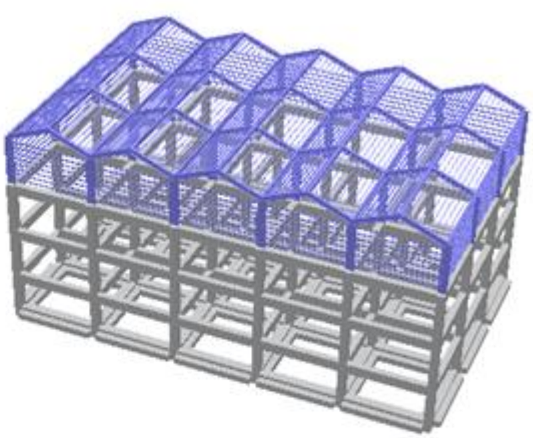
- Select templates command
- Indicate the insert point
- Inside the templates window, that opens automatically, press Open and select the template from the list



Open and the Ok.





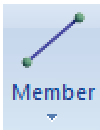


The above option is a very useful tool especially for Mixed structures, since the time required to describe the model is minimized, increasing the productivity of the designer.

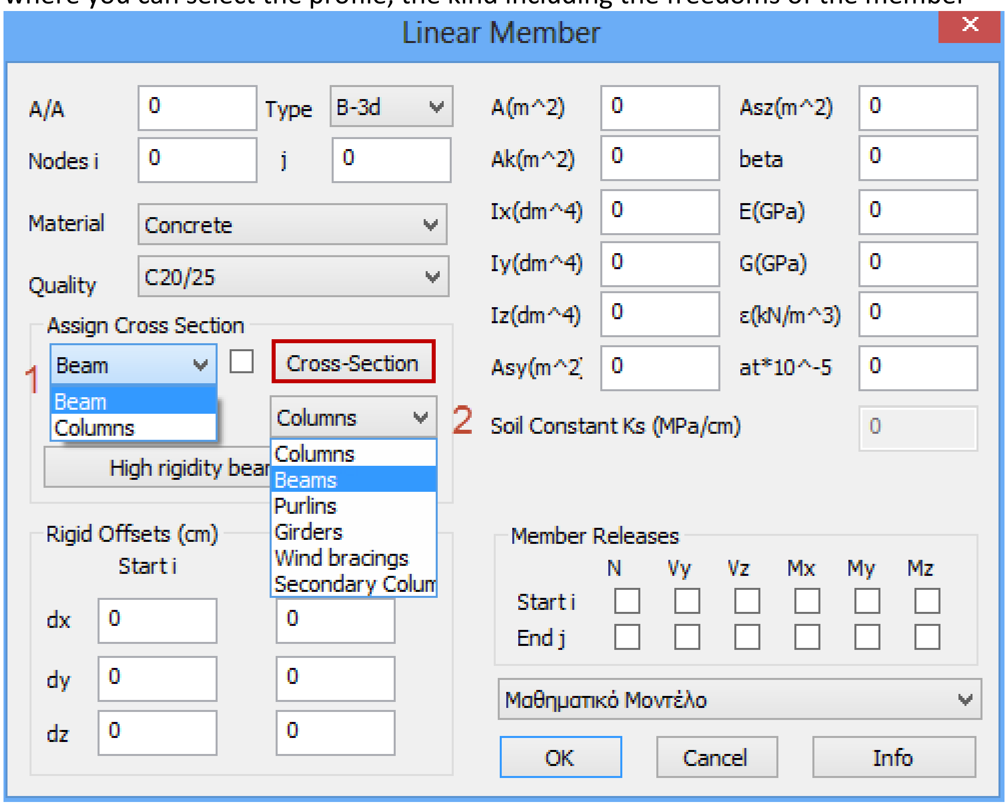
2.3 How to modify a structure created using templates command :

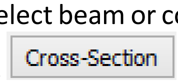
When the template enters in SCADA Pro’s environment, it is possible to make modifications or additions and create even the most complex structures.

You can delete members and nodes using deleting command from tab “Basic’, or use the right mouse button.

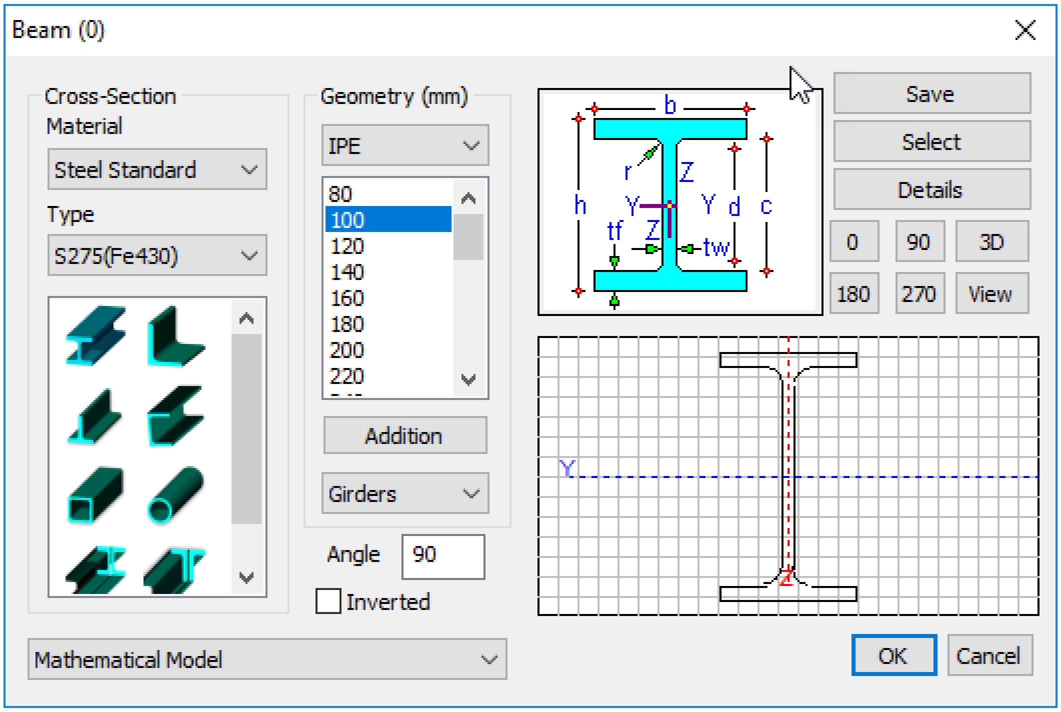


You can add elements. Just select “Linear Member” from Modeling, which opens a window where you can select the profile, the kind including the freedoms of the member



In Assign Cross Section field select beam or column from the first list, and from the second list, the specific type of the section, and press 

. In the new dialog box type the characteristics of the section



Press "ok" for inserting and, in the 3d view, pick the start node and the end node.

3. IMPORT LOADS

3.1. Manual import loads :

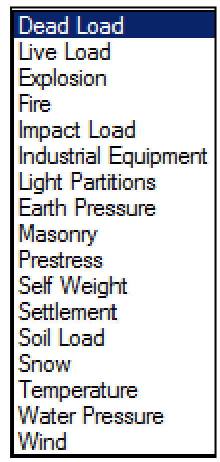
Described here, for educational reasons, the methodology of import loads manually.
Normally we use the "Wind and Snow Loads" tool for automatic importing of wind and snow loads. Therefore, you can skip following and go directly to pag.25.

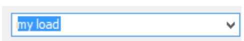
To define the Load cases use "Load Cases" command.
There are two default load cases:

3 Dead Loads (L.C.=1)

4 Live Loads (L.C.=2)

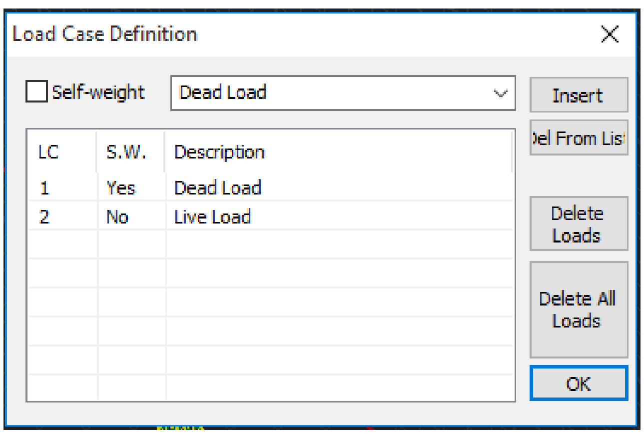
The S.W. column indicates the participation of the self-weight in the specific load case.



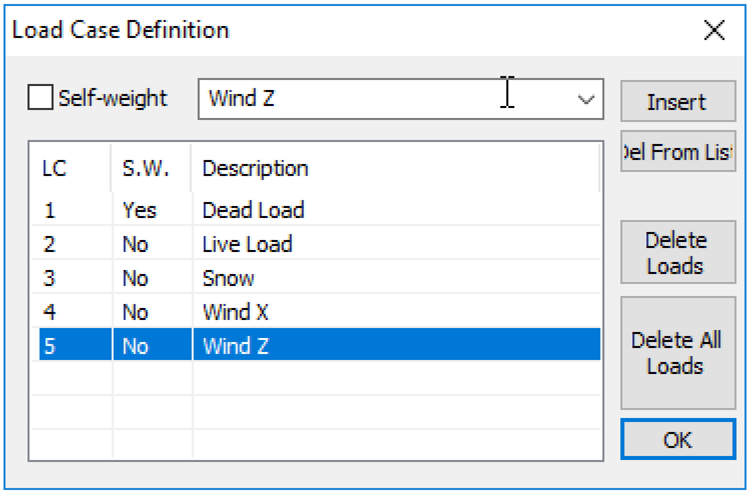
In addition to dead and live default loads, you can also define other loads. You can choose loads from the list, or you can set your load by typing a name 

and then click the "Insert" button.

The imported load, takes a serial number, in LC (Load Case) column and a "Yes" or "No" indication, depending on, whether or not, the self-weight is including.

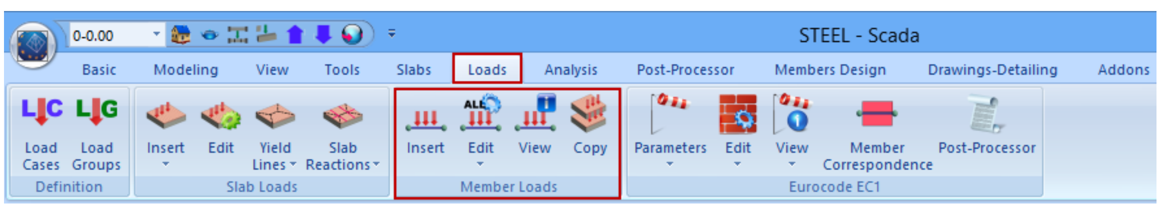


For this example click on the third line and choose Snow from the list (Load case 3). Click on the fourth row, choose Wind and type X (Load case 4) and respectively Wind Z in the fifth row (Load case 5), like in the figure below:



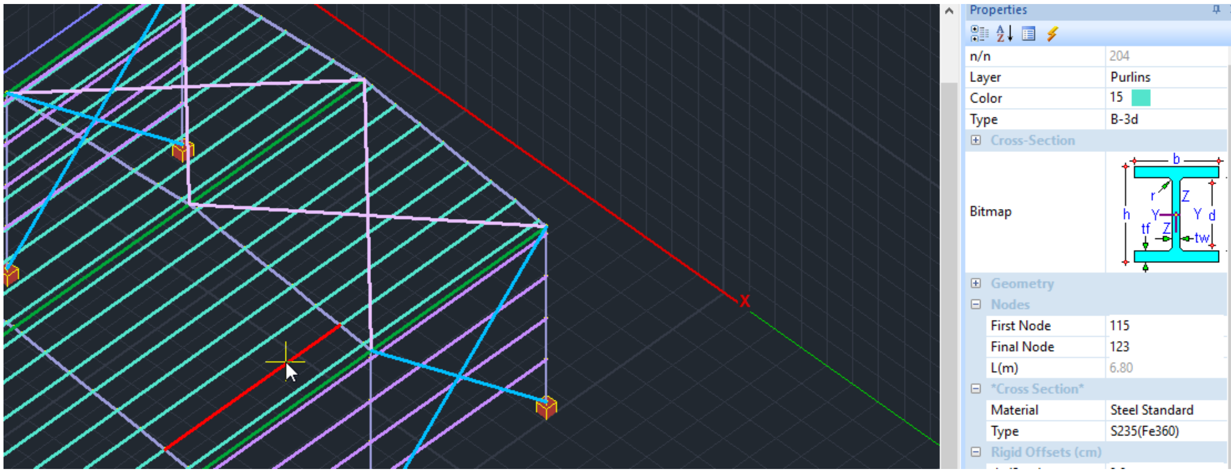
In this example, consider:
- Dead load: 0,20 kN / m in all purlins
- Live load: 0,10 kN / m in all purlins
- Snow load: 0,15 kN / m in all purlins
* Alternatively you could put the snow loads on the beams
- Wind load in X: 8KN / m to the columns with red light color and 2 KN / m to the columns deep red color.
- Wind load in Z: 3KN / m to all columns.
* Alternatively you could put the wind load on the girders.

Further information:



In “Load” tab and “Member Load” command group, select “**Insert**” and left click to define the elements (member, node, and surface) to insert the loads.

You can choose the elements either one by one or group elements through filters. Here, we use the second method.
First of all, select a purlin to see the Properties on the right side.

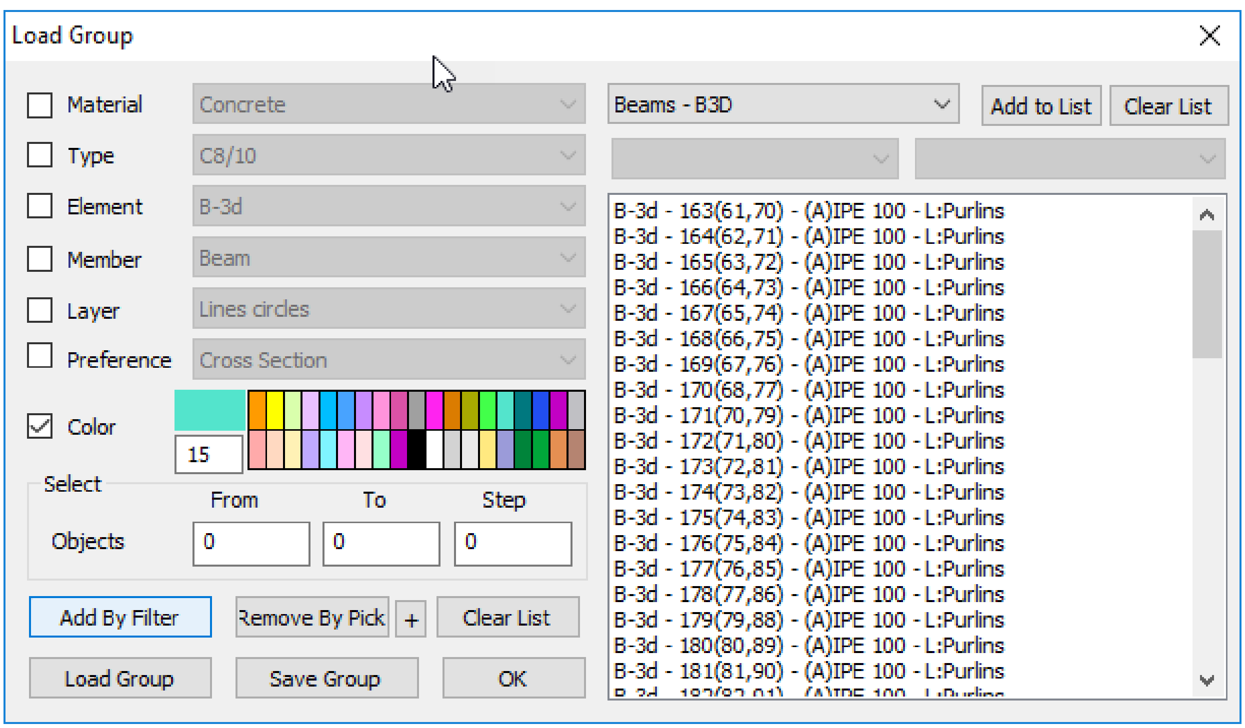


Here you can read the number of the color of the Layer: Purlins

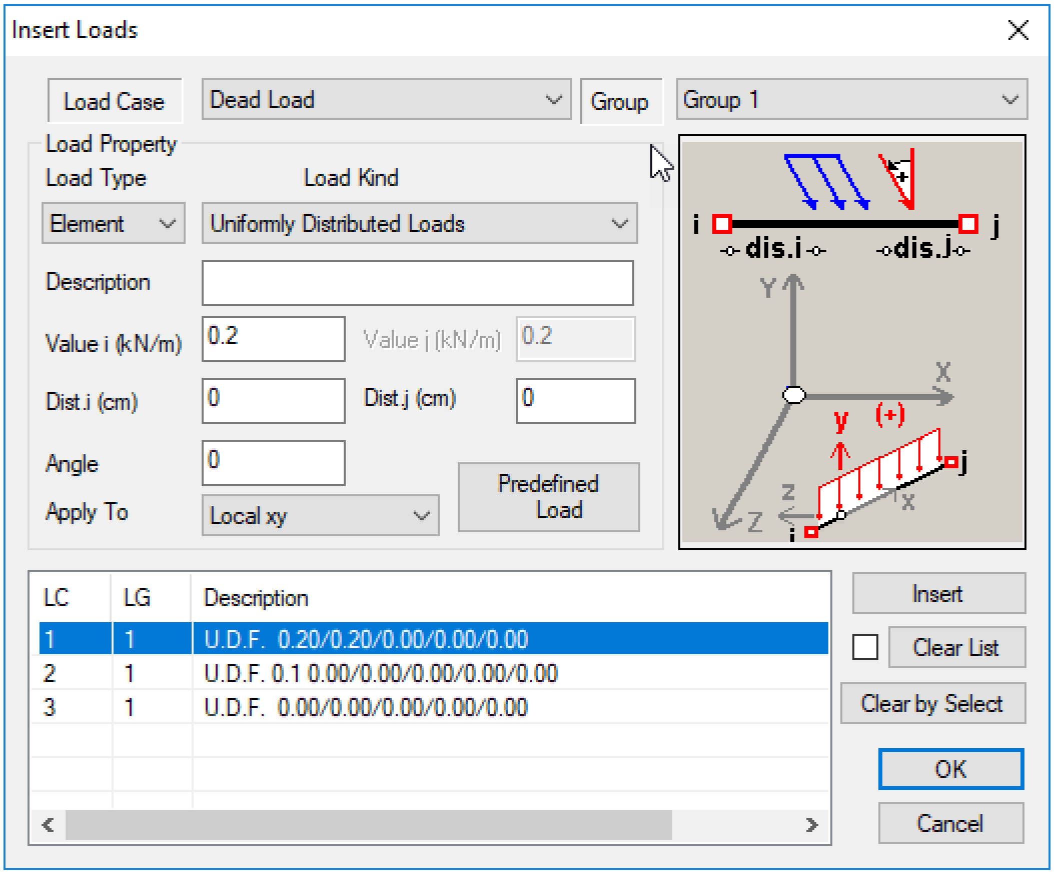
Select "Insert " and then "Select Group" button

that opens a new window.

5 Activate Color, and type the number of the color of the purlins.
6 Press "Add by Filter".
Automatically, in the right area of the window, display the list with the purlins.



Click OK and right click and opens the Insert Load window.
Select:
- Dead Load and type 0,2 and press Insert.
- Live Loads and type 0,1 and press Insert.
- Snow and type 0,15 and press Insert.

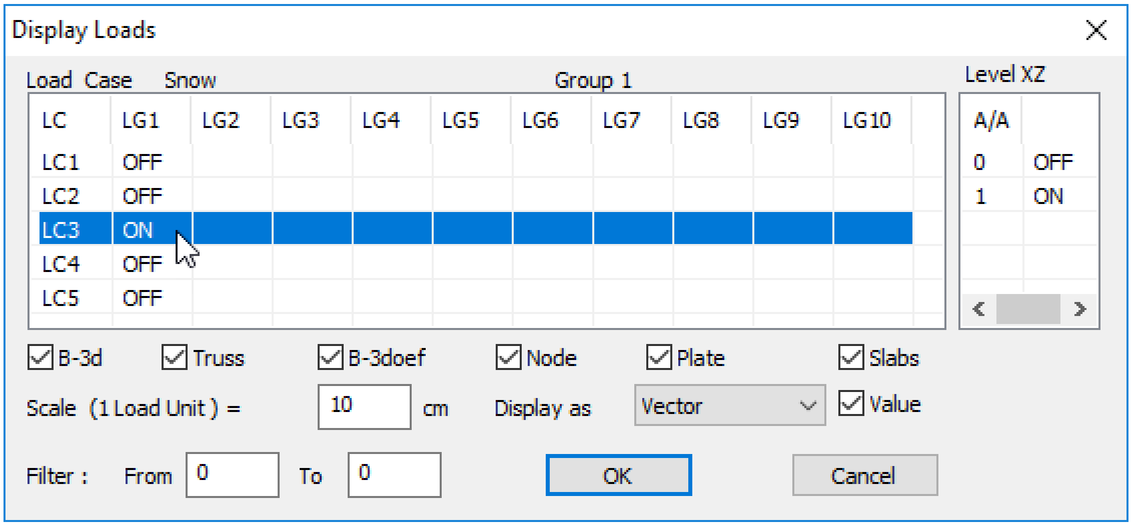


Click OK to apply the loads and close the window.

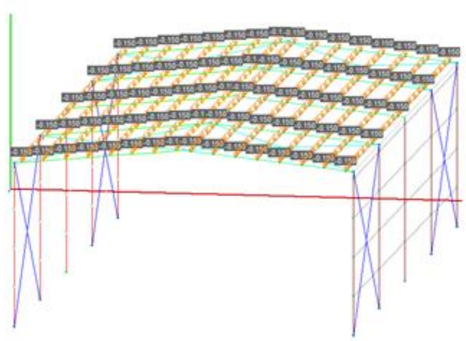
Repeat the same process for loading the Columns.

Furthermore, pressing/command, you can see all imported loads in 3d representation. In the dialog box, select load case and level.

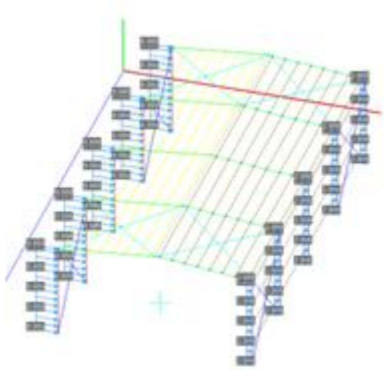
For example, turn on the snow - LC3 ON:

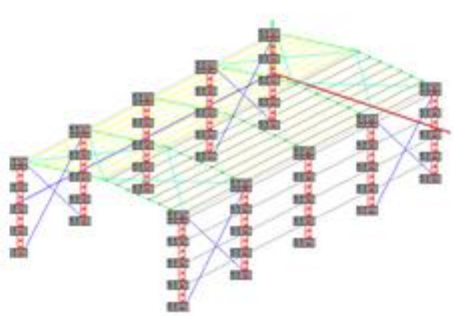


On your screen will display the snow loads on each purlin of the roof as yellow vectors pointing down.

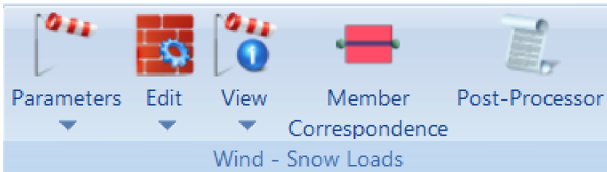


Repeat the process to enter the wind loads in the X and Z columns axis. View command will display the loads as shown in the figures below:





3.1 Automatic import loads. How to insert wind and snow loads automatically by EC 1:



"Wind-Snow Loads" commands group contains tools for the automatic calculation of wind and snow loads and the distribution to members by Eurocode 1.

It is an extraordinary tool that includes:

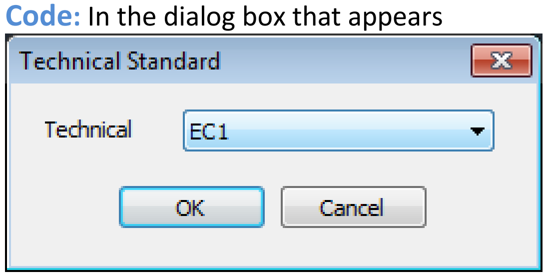
- Automatic calculation of characteristic values of snow load on the ground and the roofs determined by EN 1990 for all types of the roof: flat, single, double, quadruple, vaulted, with proximity roof tallest building drift in protrusions and obstacles of the Roof shape coefficients automatic calculation.
- 2D and 3D display of snow load distribution.
- Basic wind velocity automatic calculation.
- Automatic calculation of average wind speed VM (z) at height z (according to soil roughness and orography)
- Categories and soil parameters
- Wind turbulence
- Max velocity
- Wind pressure distribution on surfaces
- Wind forces
- Pressure coefficients for buildings (vertical walls or roofs)

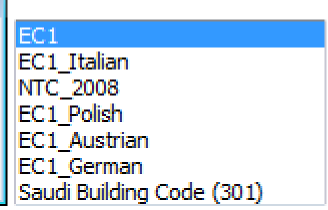
The procedure for calculating wind and snow loads and their distribution to members includes five groups of commands:

1. **Parameters**: Code selection, Wind-snow general parameters
2. **Edit**: wall-roof
3. **View**: wind-snow
4. **Member correspondence**
5. **Post-Processor**

3.1.1 Parameters

Code: In the dialog box that appears

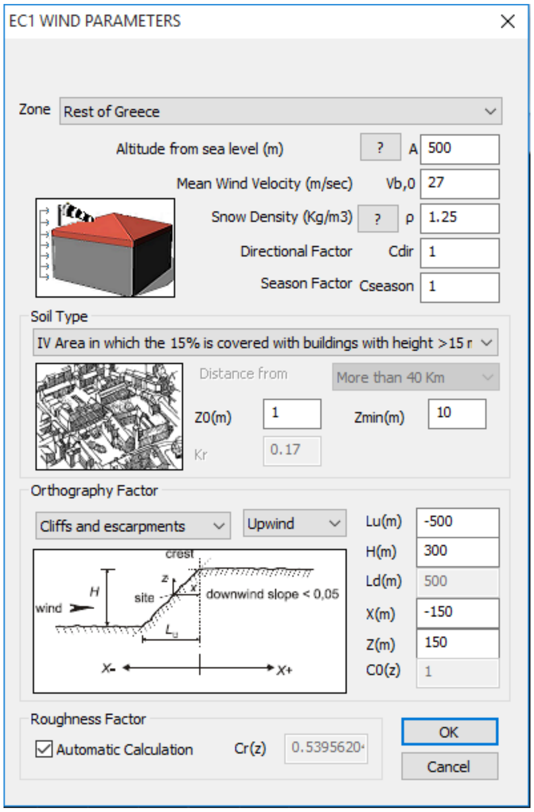




Select the code for wind and snow loads calculation.

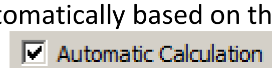
3.1.1.1 Wind

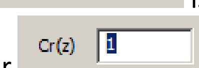
Define wind parameters according to Eurocode 1, completing the dialog box:



Select from the list: "Regulation" and "Zone" and automatically update the respective fields.
In "Soil Type": select type from the list, category, and distance from the coast.
In "Orthography Factor": define topography and wind direction.

The other fields complete automatically based on the previous selections.

In "Roughness Factor": when 

is activated, the program calculates automatically Cr(z) value, otherwise, type a number 

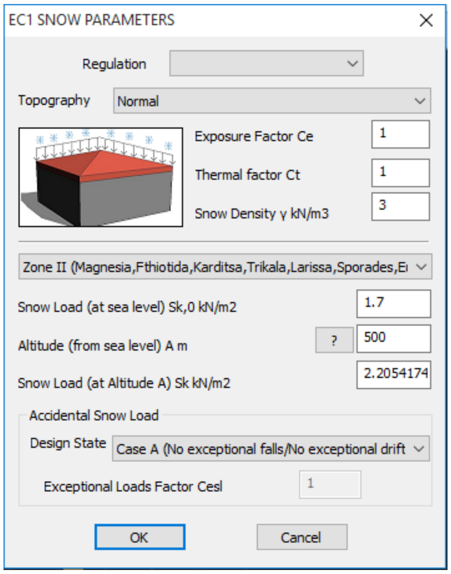
.

Press "OK" command to save the parameters.

⚠ *The user can modify the calculated values. Typing different values in the fields and the program updates automatically.*

3.1.1.2 Snow

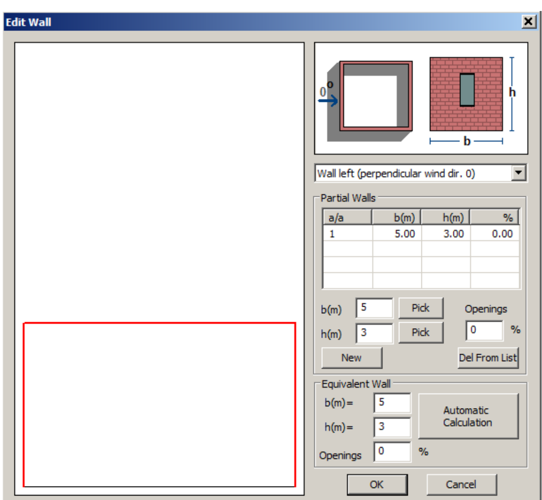
Define snow parameters according to Eurocode 1, completing the dialog box:



Select from the list: “Regulation”. “Topography” and “Zone” and automatically update the respective fields.
In “Accidental Snow Load”: select a condition.
Press “OK” command to save the parameters.

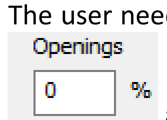
3.1.2 Edit: Wall-Roof

3.1.2.1 Walls

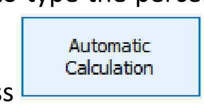


The τake advantage of the "Templates" command, when all the geometric characteristics of the walls filled in automatically by the program, and save a lot of time and work!

Select from the list the wall according to the wind direction.
“Partial Walls” list completed automatically, without using “Pick” as previous.
The user needs only to type the percentage of openings



and press



.

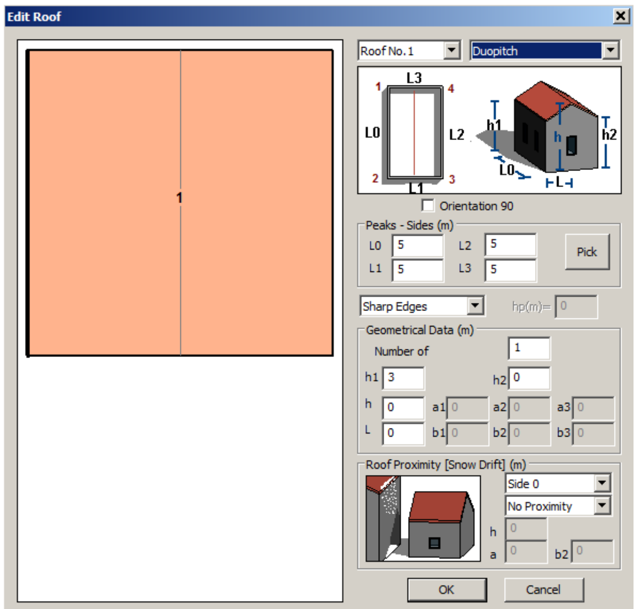
The program calculates automatically the "Equivalent Wall."
Press “OK” command to save the parameters.
Repeat for all four directions of the walls.

NOTE:

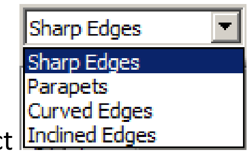
⚠ *The height of the lower wall always defined starting from level 0 even if the steel structure begins at a higher level.*

⚠ *If the front view consists of several walls at one or more levels, press the button "New" and repeat the above procedure to set the whole face.*

3.1.2.2 Roofs



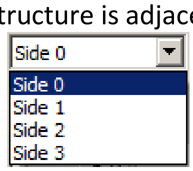
Select from the lists the roof number and the form.
“Geometrical Data” filled completed automatically.



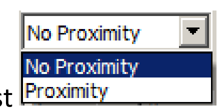
The user needs only to select , type the height of the barrier in m and define the "Roof Proximity.
Press “OK” command to save the parameters.
Repeat for all four directions of the roof (clockwise direction).

Roof Proximity

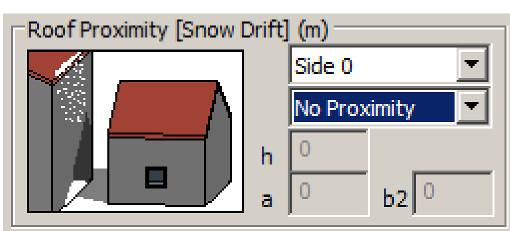
If the structure is adjacent to another building taller, in the "Roof Proximity" select the side which adjoins



and from the list

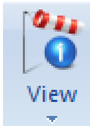


the “Proximity”.
The field change according to the proximity and the side. Type the geometry data and



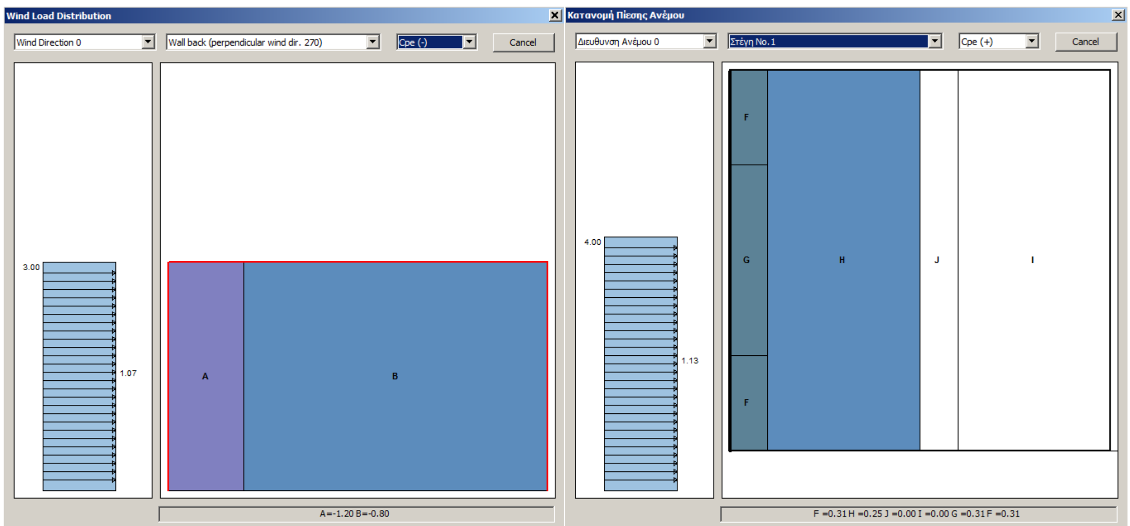
Press “OK” command to save the parameters.
Repeat for all four directions of the roof (clockwise direction).

3.1.3 View: Wind-Snow



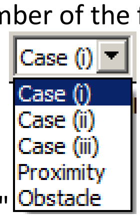
3.1.3.1 Wind

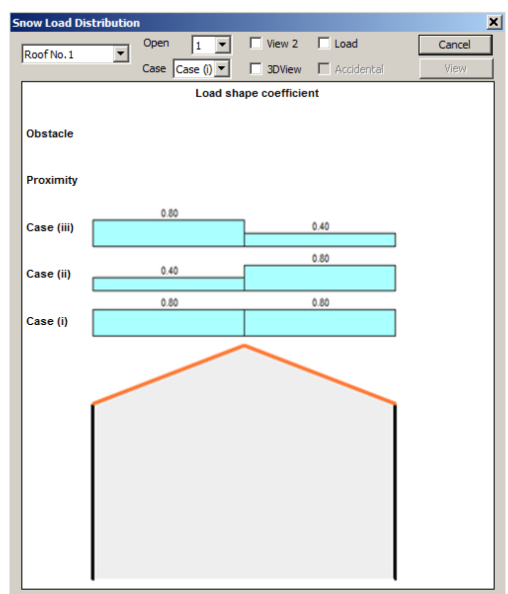
Select the command to see the wind pressure distribution on the walls and the roofs of the building. In the dialog box, select the wind direction, the wall or the roof and the type of pressure. The distribution is automatically displayed with colors. The zones with different pressure have a different color.



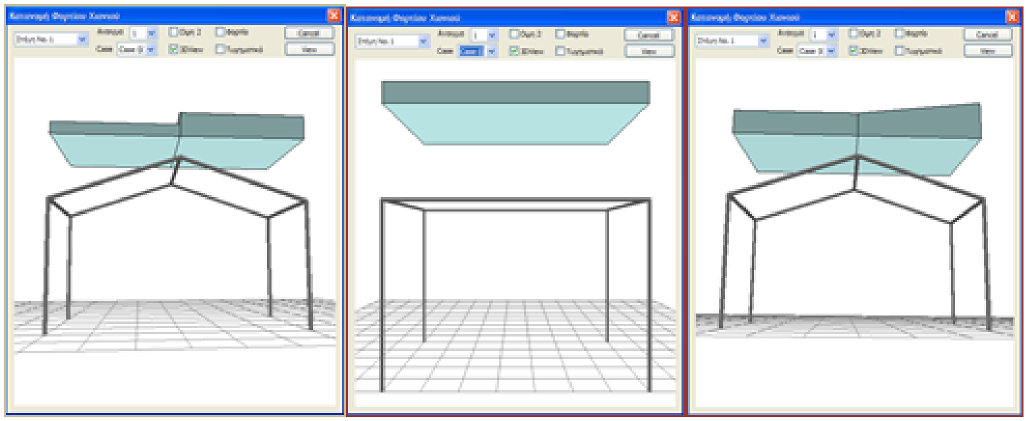
3.1.3.2 Snow

Select the command to see the snow distribution on the roofs

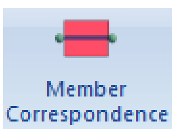
In the dialog box, select from the list the number of "roof" of the "open" meaning the number of the frame,(if there are more than one), and "Case"  for the load distribution of snow.



Activate “Load” checkbox to display the values and “3DView” checkbox to receive snow distribution the following configuration.



3.1.4 Member correspondence

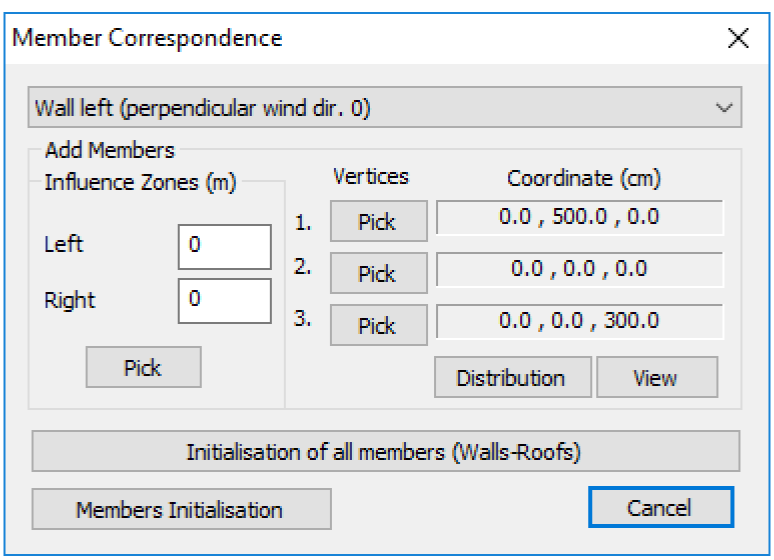


To assign the calculated loads to the members, through the influence zones.
Select the command and in the dialog box: select a wall, or a roof and define the dimension of the influence zones.

In the new version of SCADA Pro, completed and integrated the automatic calculation of influence zones for linear members to make the distribution of wind and snow loads.

⚠ *Remind that until now the automatic distribution was only for the structures derived from Templates. Now enable this distribution on any surface.*

By selecting the command now opens the following dialog box



The part of the old definition of the influence zones did not change but added to the right a new part to define the area with three points.
The definition always concerns the active area

Wall left (perpendicular wind dir. 0)

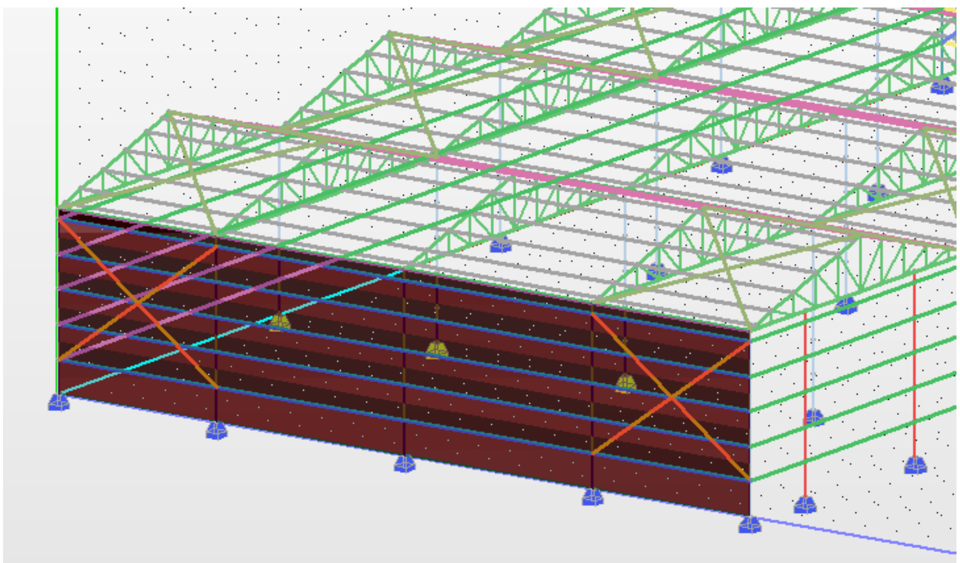
*Better to start either the **manual** or **semi-automatic** procedure by pressing the "Members Initialization” button.*

Attention:

⚠ *In the **automatic** procedure coming from the TEMPLATES, DO NOT press "Members Initialization” button, because it will delete the automatic load distribution to members!!!*

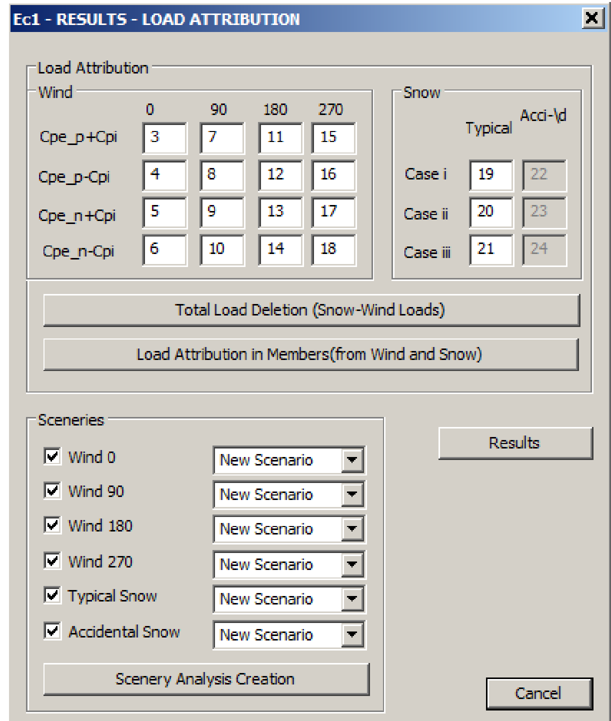
- **Automatic Procedure - Using “TEMPLATES”**

Activated “Purlins” and “Girders” in “Load Attribution” of “Templates”/, just select "Pick" and the program automatically calculates the influence zones distributing the pressure in all purlins and girders.



3.1.5 Post-Processor

The last command is "Post-Processor".



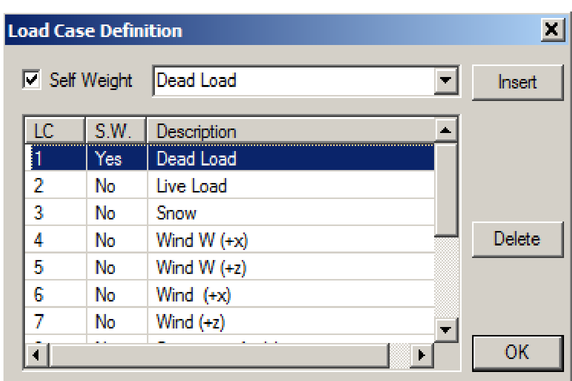
In the dialog box, in "Load Attribution" there are two labels:
one with the wind loads, 4 cases for four directions, i.e. 12 cases to each load and
one with snow loads, 3 cases for typical snow.

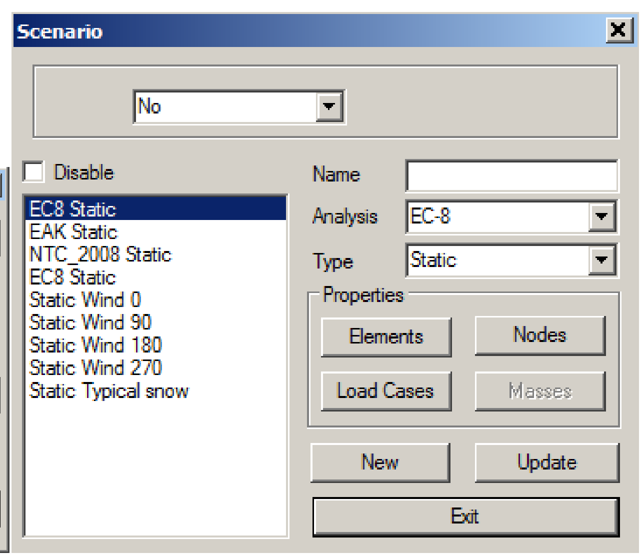
The numbers that appear on labels correspond to the Load Cases serial numbers.
Remember:
Load Case1: Dead
Load Case2: Live
and now there are 16 new Load Cases for the Wind (from 3 to 18) and 3 for the Snow (19, 20 and 21).

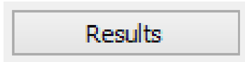
Select
Total Load Deletion (Snow-Wind Loads) to attribute wind a snow loads on the structural members, or Load Attribution in Members(from Wind and Snow) to delete them all.

In "Sceneries" there is a list with the analysis sceneries creating automatically selecting Scenery Analysis Creation command.

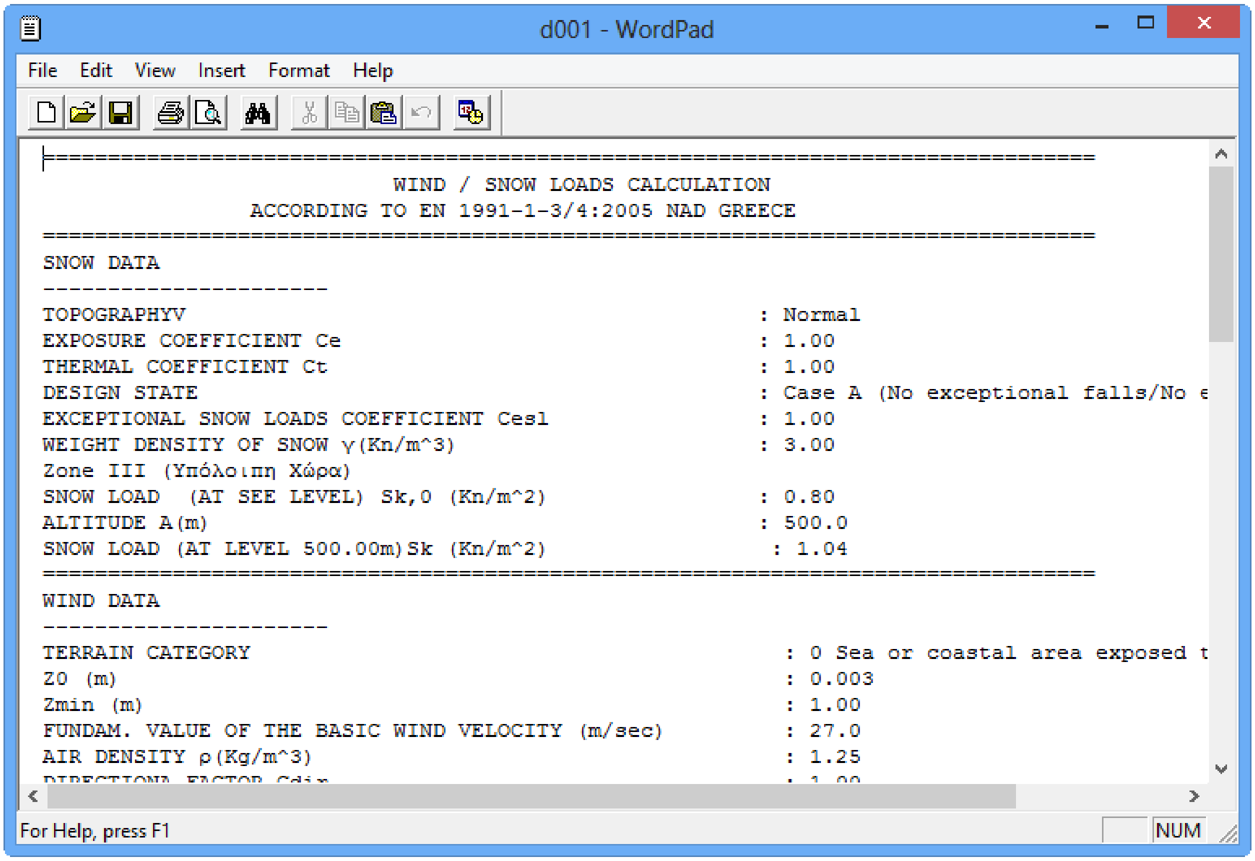
⚠ SCADA Pro not only calculates automatically the load distribution of wind and snow but also creates automatically all the analysis sceneries, saving the user from hard work and much time.





Press 

to open the .txt results file, containing in detail all data and calculations derived from all Eurocode 1 procedure.



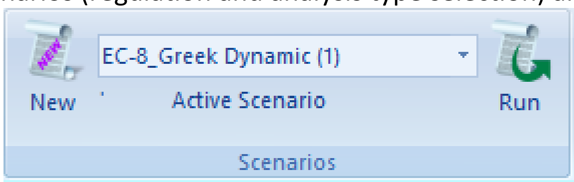

```
================================================================================
                        WIND / SNOW LOADS CALCULATION
               ACCORDING TO EN 1991-1-3/4:2005 NAD GREECE
================================================================================
SNOW DATA
----------------------
TOPOGRAPHYV                                           : Normal
EXPOSURE COEFFICIENT Ce                               : 1.00
THERMAL COEFFICIENT Ct                                : 1.00
DESIGN STATE                                          : Case A (No exceptional falls/No e
EXCEPTIONAL SNOW LOADS COEFFICIENT Cesl               : 1.00
WEIGHT DENSITY OF SNOW γ(Kn/m^3)                      : 3.00
Zone III (Υπόλοιπη Χώρα)
SNOW LOAD  (AT SEE LEVEL) Sk,0 (Kn/m^2)               : 0.80
ALTITUDE A(m)                                         : 500.0
SNOW LOAD (AT LEVEL 500.00m)Sk (Kn/m^2)                : 1.04
================================================================================
WIND DATA
----------------------
TERRAIN CATEGORY                                        : 0 Sea or coastal area exposed t
Z0 (m)                                                  : 0.003
Zmin (m)                                                : 1.00
FUNDAM. VALUE OF THE BASIC WIND VELOCITY (m/sec)        : 27.0
AIR DENSITY ρ(Kg/m^3)                                   : 1.25
```

4. ANALYSIS

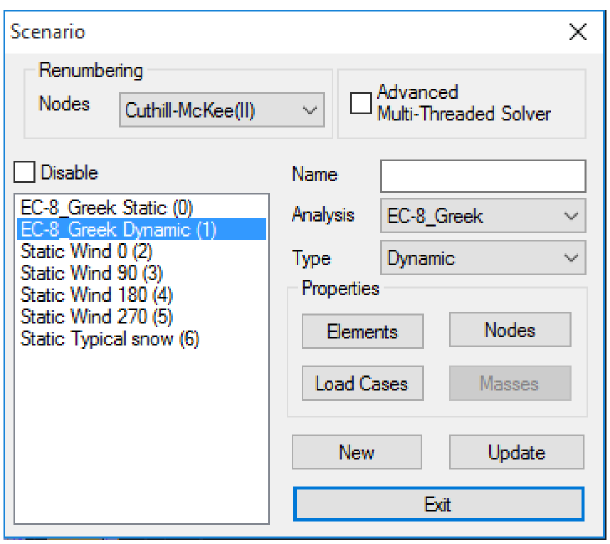
After the modeling and the distribution of the loads to the members of the structure, the analysis of the structure, by the selected regulation, the creation of the load combination and the results of the checks are next.

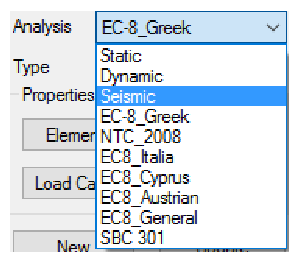
4.1 How to create an analysis scenario:

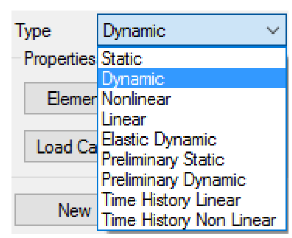
Through the “Analysis” unit, the commands of the “Scenarios” group allow the creation of the analysis scenarios (regulation and analysis type selection) and the execution.

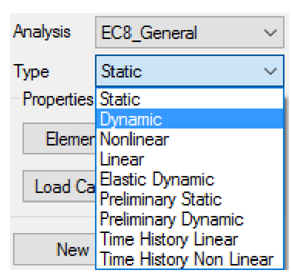


To create more analysis scenarios, select “New”. In the dialog box that opens, besides the predefined ones, you can create as many scenarios as you want.









Make your selections from the “Analysis” and the “Type” lists and click New to create a new scenario. If you want you can type a name as well.

Select among the possible scenarios provided in SCADA Pro:

<u>For Greece:</u>

<u>LINEAR – NON-LINEAR METHODS</u>

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

<u>For other countries:</u>

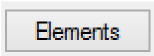
<u>LINEAR – NON-LINEAR METHODS</u>

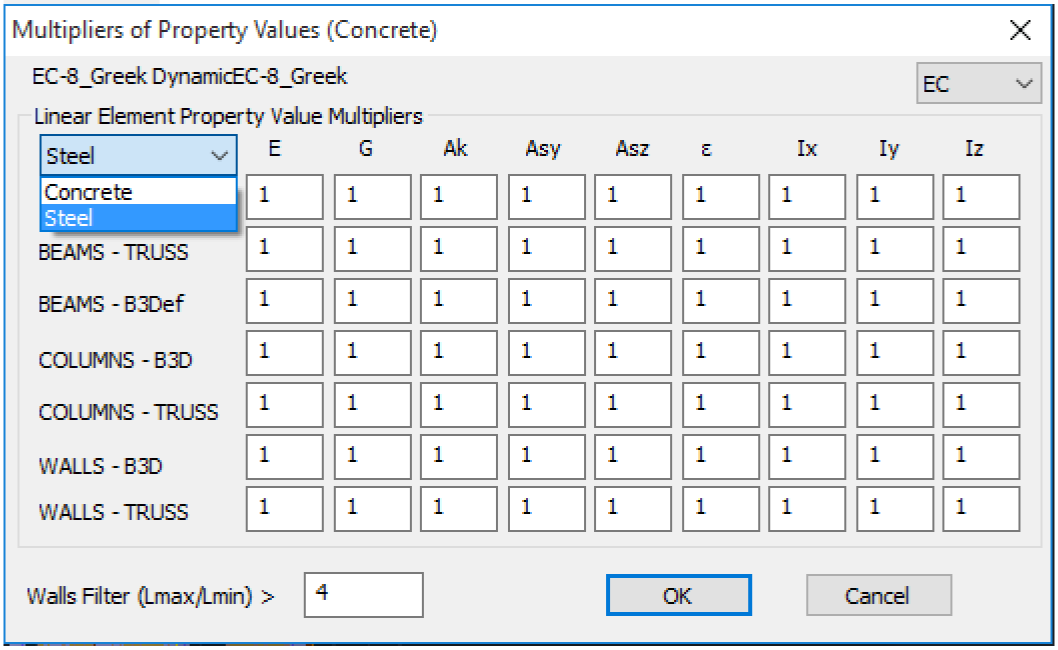
- NTC 2008	Seismic analysis according to the Italian Regulation 2008
- EC8 Italia	Seismic analysis according to Eurocode 8 and the Italian Appendix
- EC8 Cyprus	Seismic analysis according to Eurocode 8 and the Cyprus Appendix
- EC8 Austrian	Seismic analysis according to Eurocode 8 and the Austrian Appendix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

In this example, you’ll only choose the scenarios EC8 dynamic for the earthquake, as well as the scenarios Snow Typical, Wind 0 and Wind 90, which were automatically created as previously explained.

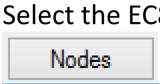
Select the EC8 Dynamic. The command **Elements** includes the properties modifiers for the beam members.

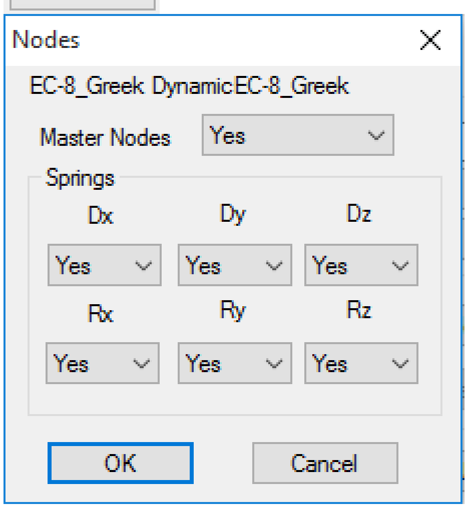
⚠ *The program automatically chooses the appropriate inertial modifiers, by the selected regulation while you can modify at any time these modifiers.*





Select the EC8 Dynamic. The command **Nodes** opens the following window:

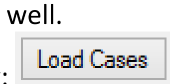


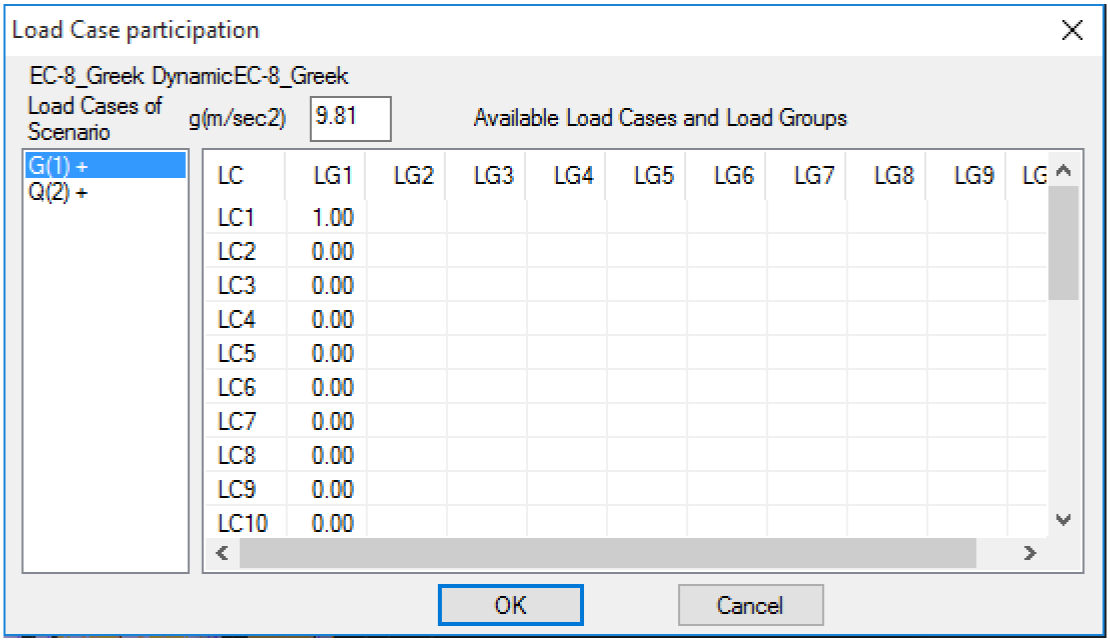


Here you can choose to perform the analysis without considering Rigid Link Constrain at any level even if master nodes exist and consider a fixed base for the whole model even if an elastic foundation is defined.

In cases of Dynamic Analysis, if you select "Nodes" and you "open" the springs "Yes", then you will be able to use the combinations of the dynamic analysis for the design of the footing as well.

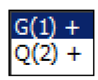
Select the EC8 Dynamic. The command **Load Cases** opens the following window:



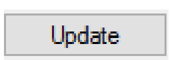


Where, for each scenario load case (for the current scenario only) on the left column, you match one or more Load Cases (LC) of those that you created.

Select the value 1.00 for LC1 (after having selected the category “Dead Loads” – G(1), that are colored blue) and 1.00 for LC2 (after having selected the category “Live Loads” – Q(2), that are colored blue).

The “+” sign next to the load category 

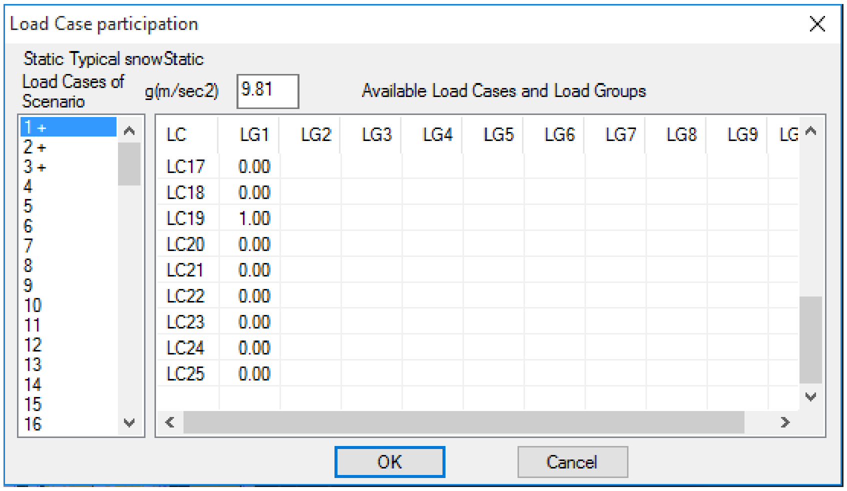
shows that for the specific category (scenario Load Case) there is a load participation. The maximum “+” signs for each scenario is 4.

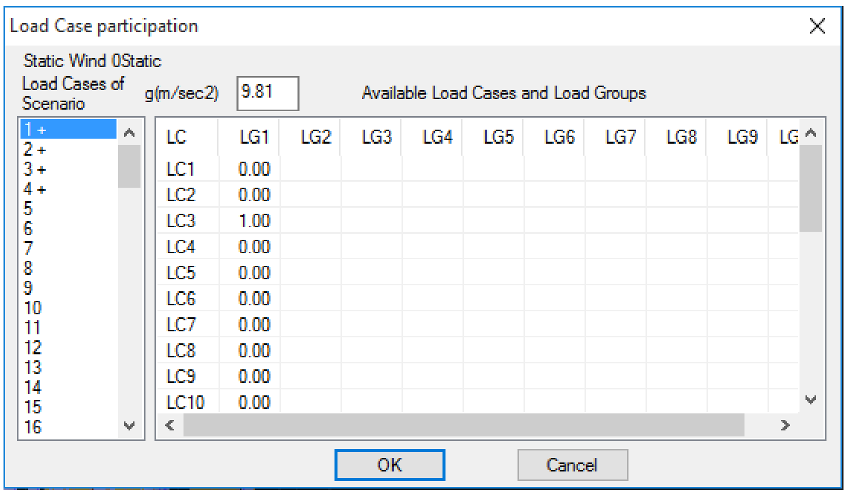
Click 

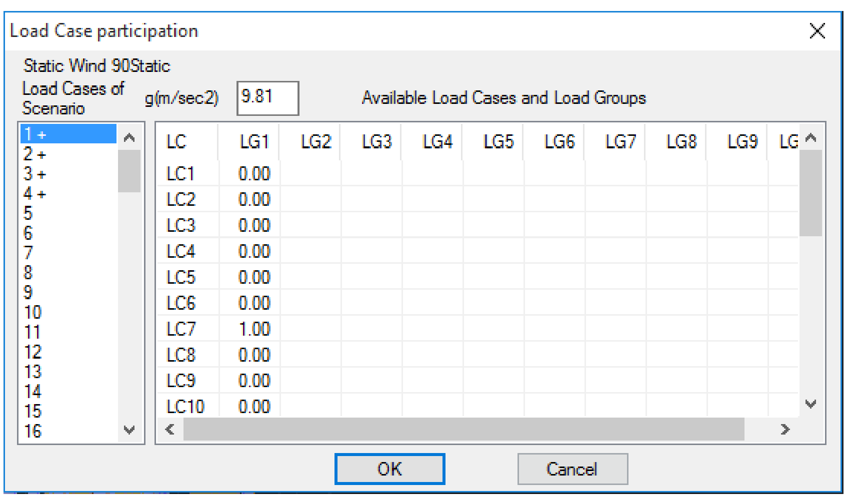
to update the scenario by the performed modifications.

The program fills automatically a unit factor to the corresponding Load Case. Any modification is acceptable here.

For Static wind and snow scenarios, the respective loads participate to the corresponding categories without including the dead and live loads derived from cases LC1 and LC2, since these are already included in the seismic analyzes.





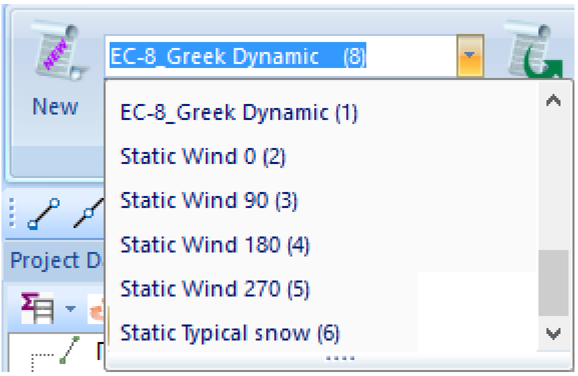


When a category is activated the + symbol appears next to it.

⚠ **NOTE**

For each scenario, you can activate up to 4 scenario load cases.

4.2 How to run an analysis scenario:



Inside the scenarios list, besides the two predefined scenarios, the scenarios related to wind and snow now exist. Select each scenario and define the corresponding parameters of the selected analysis.

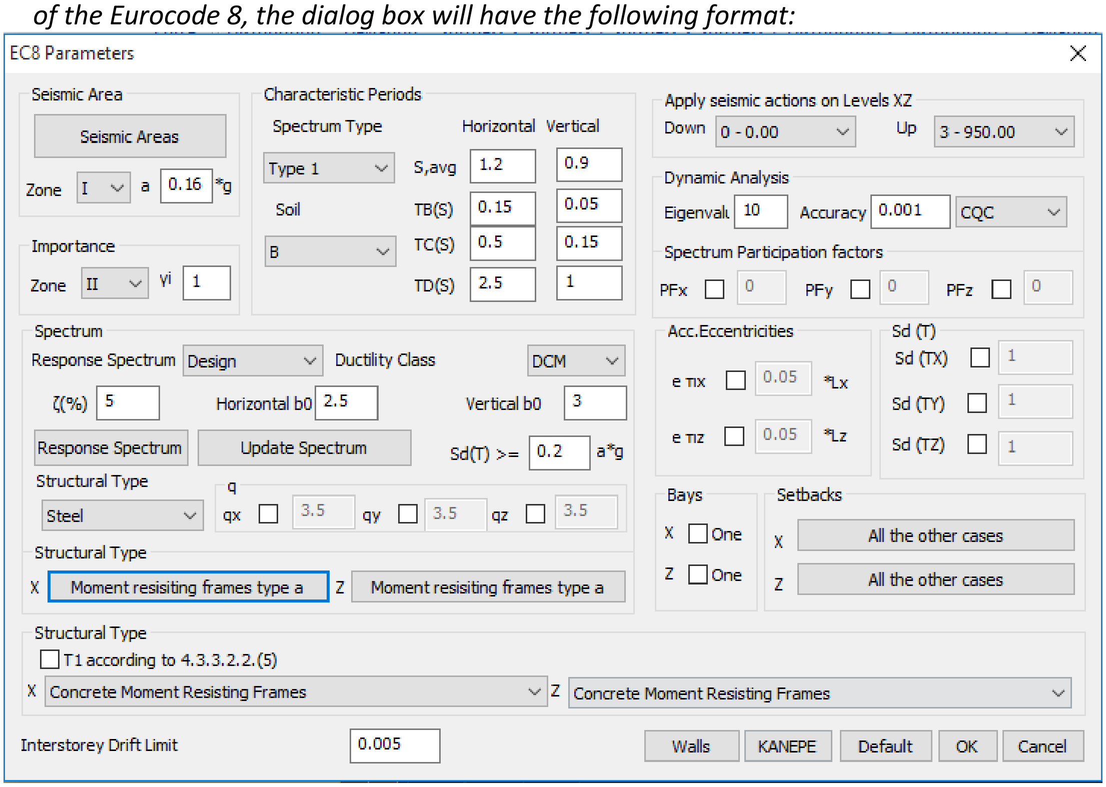
By clicking the “Run” button, depending on the selected scenario, the following dialog box opens:

- ✓ **Eurocode** scenarios
- ✓ **Static** scenarios

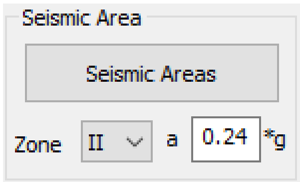
First of all, select Update to update the parameters of the current scenario and delete the data of the previously executed analysis.

Then, select Parameters to define the parameters of the current scenario.

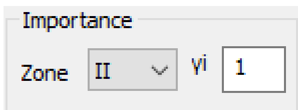
⚠ *Depending on the selected scenario, the dialog box differs. In this example having selected the scenario of the Eurocode 8, the dialog box will have the following format:*



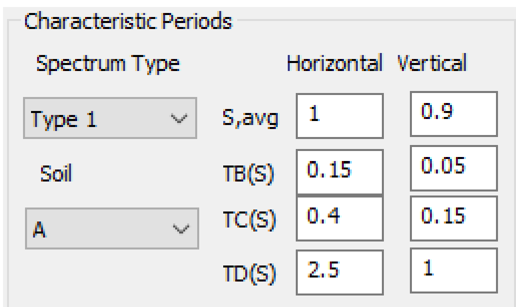
In this dialog box, you enter all the necessary data related to the seismic region, the soil the importance factor, the safety coefficients and the levels of the seismic loads application.



Click the “Seismic Areas” button and be informed of the considered zone parameters. Select the “Zone” from the list and the coefficient “a” is automatically filled in.



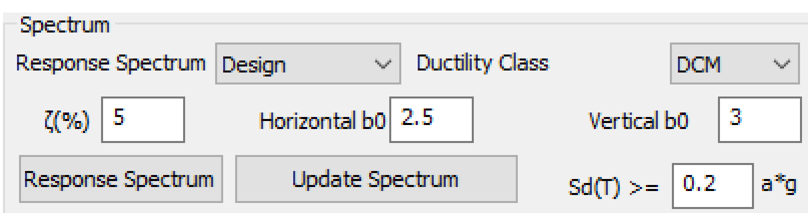
Select the important factor and the coefficient “γι” is automatically filled.



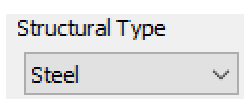
Next define the Spectrum Type and the soil type, so that the horizontal and vertical spectrum coefficients are automatically calculated.

⚠ *You can modify any of these fields and fill in your very own parameters set.*

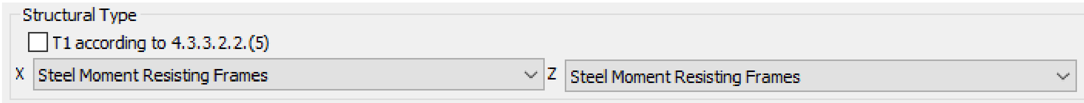
Select the “Spectrum Type” and the “Ductility Class” before you click “Update Spectrum”

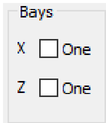


Select the “Structural Type”



A) Select the “Structural Type” along X and Z direction to calculate the basic eigen period

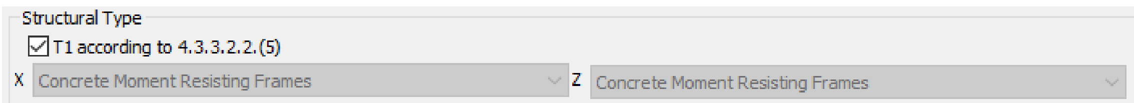


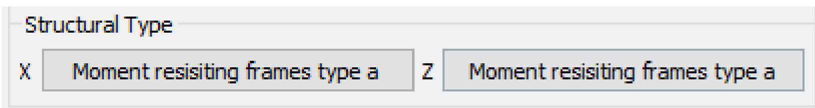
(in case of structures with a single frame along X or/and Z direction check the respective checkbox on the "Bays" group 

)

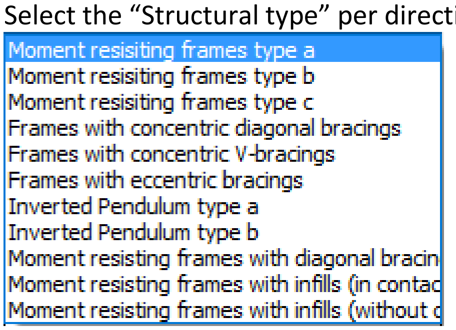
Otherwise

B) activate the checkbox to calculate the T1 by the paragraph 4.3.3.2.2. of the EC8 regardless of the structural type





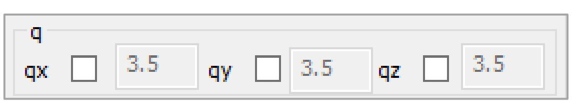
Select the "Structural type" per direction from the list:



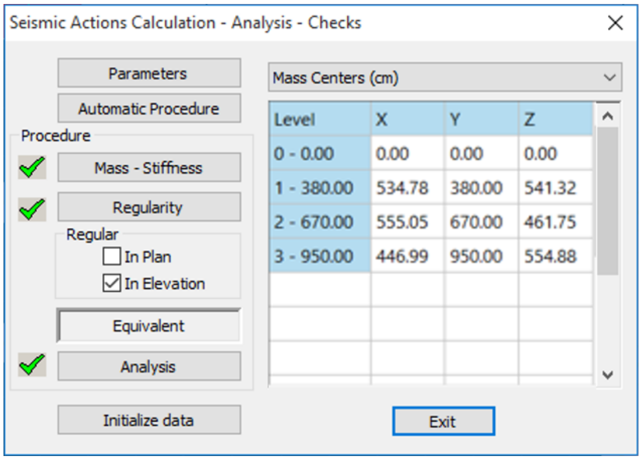
According to Eurocode the **"Behavior Coefficient q"** derives from calculations and the **"Structural type"** must follow specific criteria.

⚠ *SCADA Pro calculates automatically the q factor and the structural type. The process is explained next:*

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



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

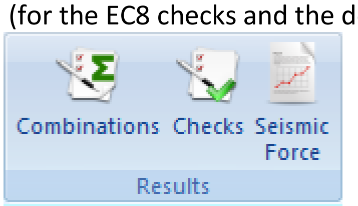


❖ Now, the proposed values for the "Behavior coefficient q" can be found in the "Parameters" dialog box.

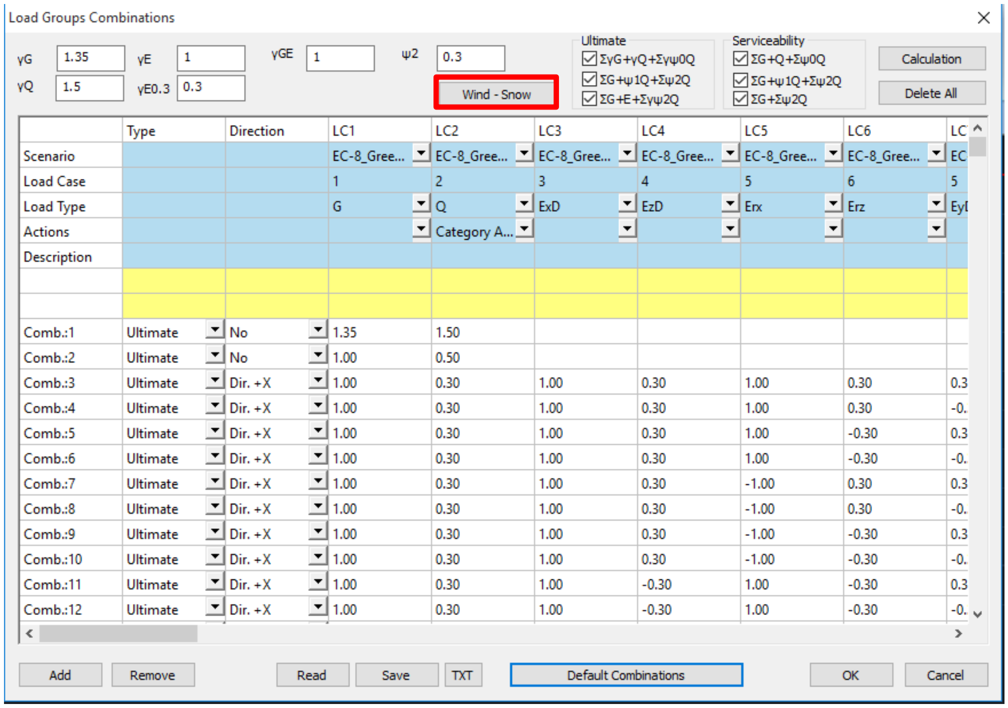
❖ The proposed values may be kept or altered (the latter one is an option that could be utilized from the beginning of the procedure, however, in this occasion the software would not propose any values by EC8).

4.3 How to create load combination:

Right after the analysis execution, use the command group "Results", to create the load combinations (for the EC8 checks and the design) and display the results of the analysis:



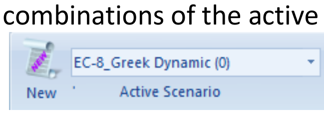
The "Combinations" command, opens the "Load Groups Combinations" dialog box where you can create your very own combinations or call the predefined combinations that SCADA Pro has.



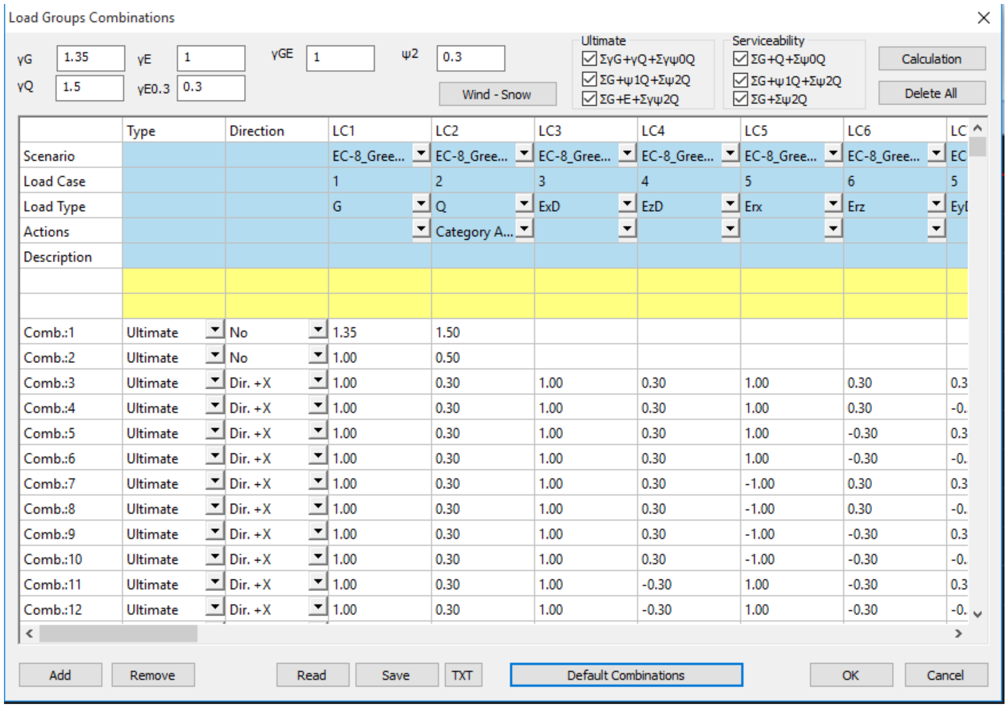
	Type	Direction	LC1	LC2	LC3	LC4	LC5	LC6	LC
Scenario			EC-8_Gree...	EC-8_Gree...	EC-8_Gree...	EC-8_Gree...	EC-8_Gree...	EC-8_Gree...	EC
Load Case			1	2	3	4	5	6	5
Load Type			G	Q	ExD	EzD	Erx	Erz	Eyl
Actions				Category A...					
Description									
Comb.:1	Ultimate	No	1.35	1.50					
Comb.:2	Ultimate	No	1.00	0.50					
Comb.:3	Ultimate	Dir. +X	1.00	0.30	1.00	0.30	1.00	0.30	0.3
Comb.:4	Ultimate	Dir. +X	1.00	0.30	1.00	0.30	1.00	0.30	-0.
Comb.:5	Ultimate	Dir. +X	1.00	0.30	1.00	0.30	1.00	-0.30	0.3
Comb.:6	Ultimate	Dir. +X	1.00	0.30	1.00	0.30	1.00	-0.30	-0.
Comb.:7	Ultimate	Dir. +X	1.00	0.30	1.00	0.30	-1.00	0.30	0.3
Comb.:8	Ultimate	Dir. +X	1.00	0.30	1.00	0.30	-1.00	0.30	-0.
Comb.:9	Ultimate	Dir. +X	1.00	0.30	1.00	0.30	-1.00	-0.30	0.3
Comb.:10	Ultimate	Dir. +X	1.00	0.30	1.00	0.30	-1.00	-0.30	-0.
Comb.:11	Ultimate	Dir. +X	1.00	0.30	1.00	-0.30	1.00	-0.30	0.3
Comb.:12	Ultimate	Dir. +X	1.00	0.30	1.00	-0.30	1.00	-0.30	-0.

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

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



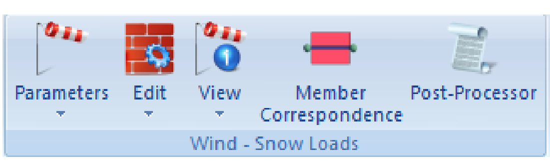
The default combinations of the executed analysis, are automatically saved by the program.

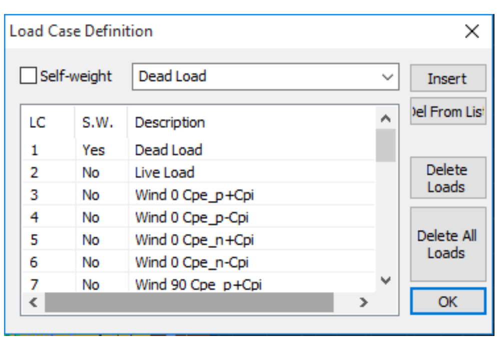


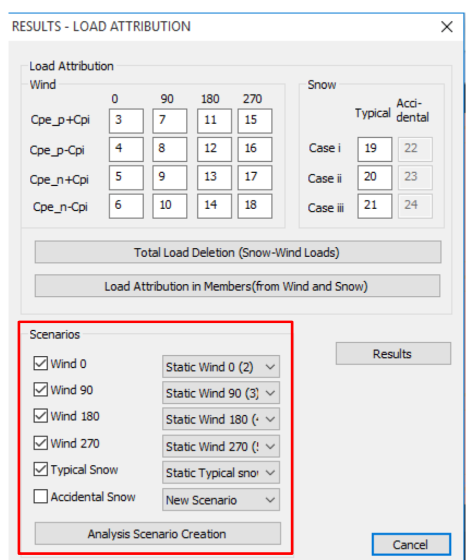
You can create your combinations without using the "Default", or add more loads of other scenarios and calculate the new combinations either by modifying the defaults or deleting all "Delete All" and typing other coefficients. The tool “Load Groups Combinations” works like an Excel file offering possibilities like a copy, delete using Ctrl+C, Ctrl+V, Shift and right click.

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

The **automatic** mode requires that the automatic procedure for the calculation and distribution of loads of wind and snow as well as the automatic creation of the loads and combinations (as in the current example) is already done.

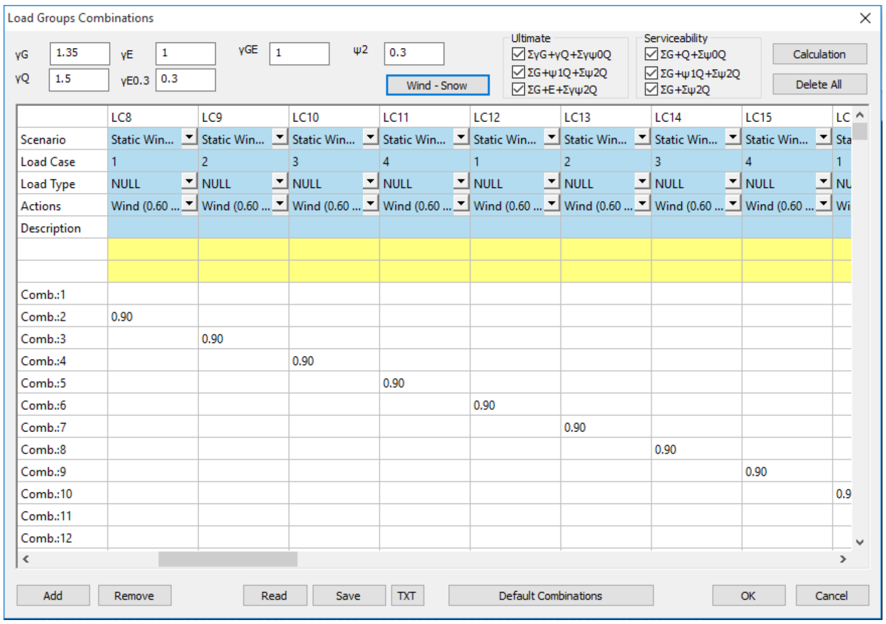






Concerning the above conditions, it is possible to automatically create wind and snow combinations by using the command Wind - Snow.

After running the seismic scenario and all the static scenarios of wind and snow, activate the seismic scenario and choose the command "Combinations". The combinations of the active seismic scenarios are completed automatically. To create automatically the combinations of all wind and snow loads, press Wind - Snow. Automatically the coefficients of all wind and snow scenarios will be filled, offering a complete loads combinations file.

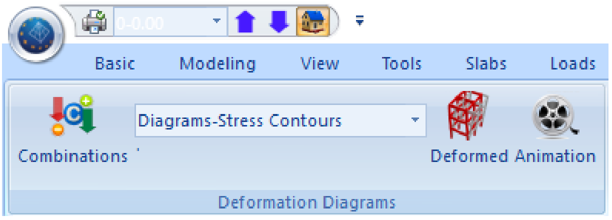


Press Save to save the file.

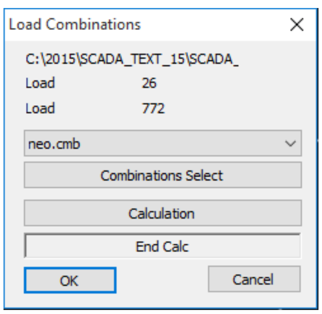
5. POST-PROCESSOR

5.1. How to display diagrams and deformations:

Activate “Post-Processor” to view the deformed shapes of the model for each load case or/and combination scaled accordingly and see the M, V, N diagrams for each member as well.

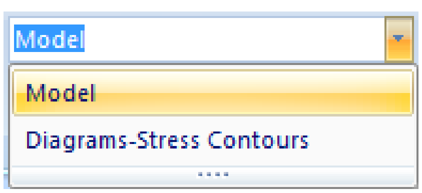


First, select “Combinations” and load a combination’s file, depending on the results you want to see. In the dialog box:



- Choose a combination from the list that includes the combinations of all the analyses that have been performed, and wait to complete the calculation automatically, or
- press “Combinations Select”, select the combinations file from the correspondent folder and press "Calculation".

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

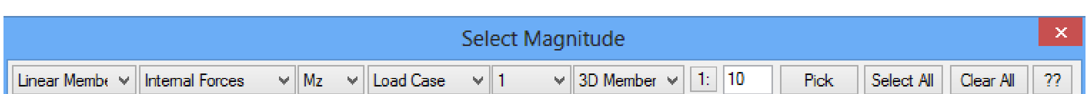


From the list on the right, based on the required results, select:

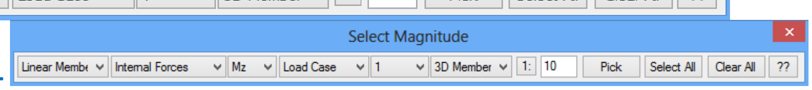
- Model or
- Diagrams Stress-Contours

Model +

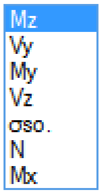
Diagrams – Stress Contours +

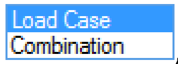


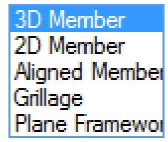
Diagrams – Stress Contours +



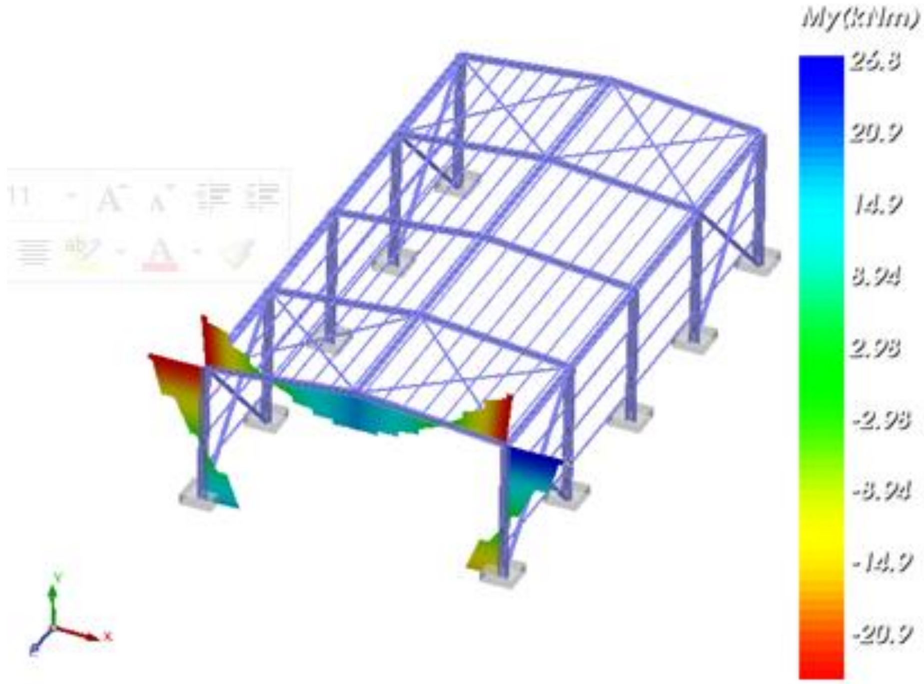
For Linear Members you can see:

tensions 

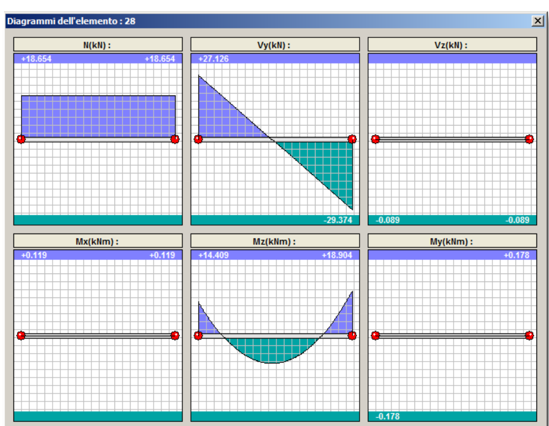
, for each 

, on a 

, a scale 

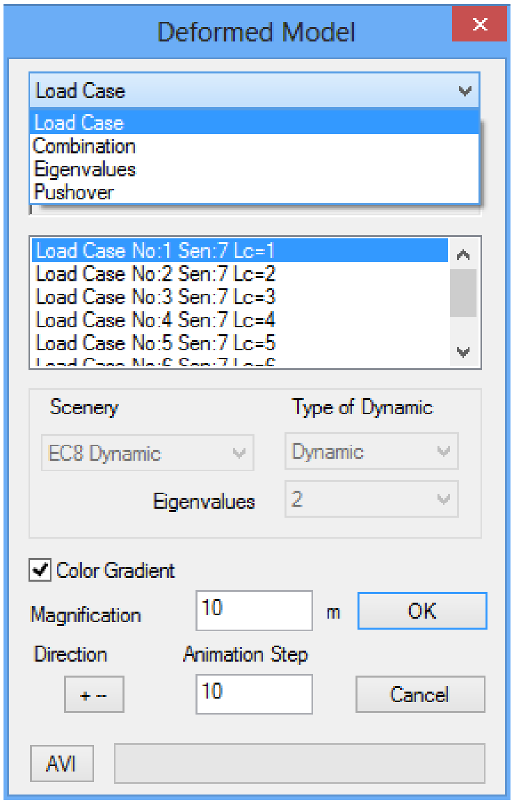


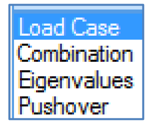
Select “2D Member” (view) to see all six internal forces of a linear member concentrated in one window. Also, while moving the mouse you can see the member values of each force.

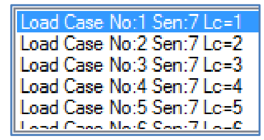


Sign convention is according to member’s local axes.

Model + "Deformed":



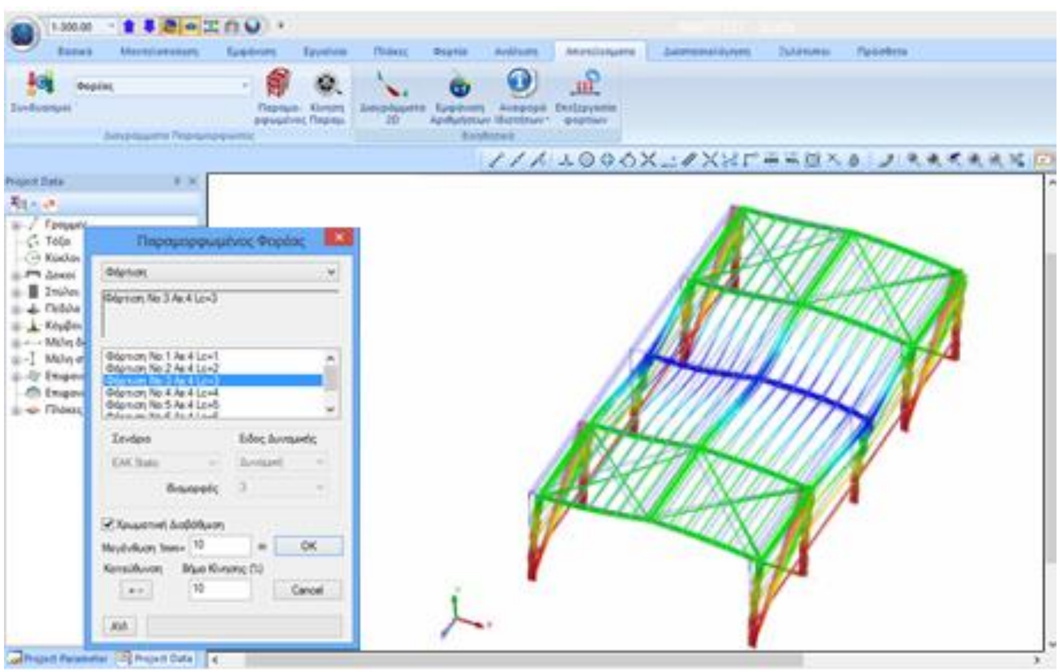
Choose 

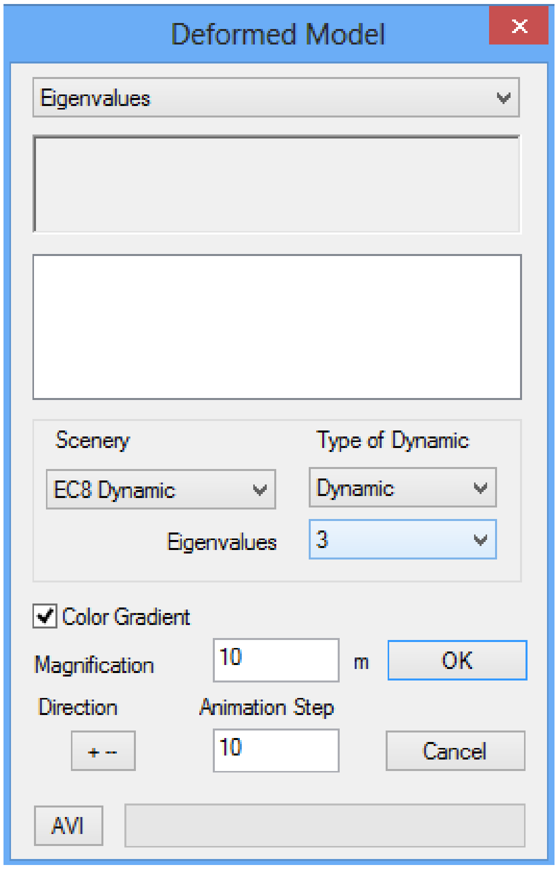
the general origin of deformation and from the next list, the specific one. 

Activate 

, modify "Magnification" and "Animation Step" , to receive a better visualization..

"AVI" button gives the possibility to register a short video with the deformation of the structure.





“Deformed Model” window remains on the screen attending to select the next deformation origin. Press Cancel to close the window.

According to the selected .cmb file, you can see the respective deformation.

Checking model deformation helps you to understand structure behavior and sometimes realize If there is some model mistake that makes it behaving inappropriately.

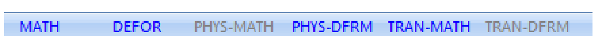
Loading Static combination you can’t see Eigenvalues deformation.

Static analysis scenarios produce deformation for each Load case or Combination.

To receive Eigenvalues deformation, first you have to operate a dynamic analysis, creating a dynamic scenario, and then to select a Dynamic combination file.

Select “Eigenvalues”, the corresponding “Scenery”, the Type and the number of Eigenvalues.

On the “Status Bar” check (double click, blue=activate, grey=deactivate) the kind of visualization of the deformed model.



Model + “Animation”:

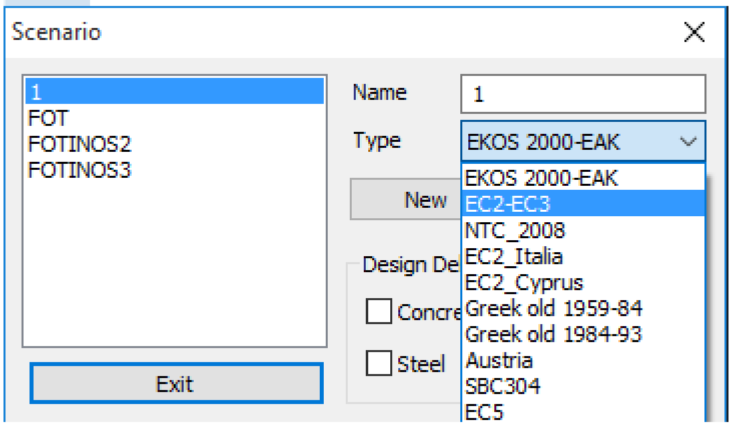
“Animation” command is a button that activates and deactivate the deformed structure of animation, according to the selections made in “Deformed Model” dialog box.

6. STEEL MEMBER DESIGN

After model analysis comes adequacy check of the static elements of the structure, according to the norms selected by the scenarios created in "Member Design", checking the sections and inserting the necessary reinforcement.

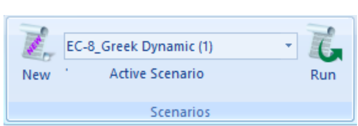
6.1 Members Design Scenarios creation :

Open "Member Design" tab and press "New" to create scenarios according to Eurocodes. Type a name and select EC2-EC3.



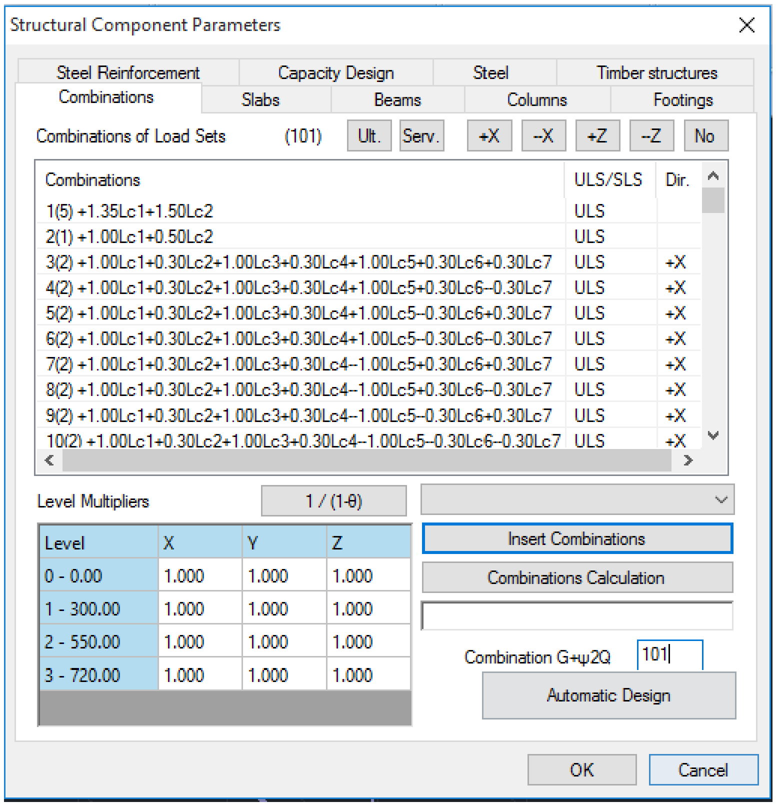
For this example, a Eurocode scenario was used.
Comment: *For steel structures, the EC3 is applied to every scenario. EC2 regards the analysis method as well as the design method of concrete cross-sections.*

6.2 How to define the parameters of the steel members design:



Select from the Scenarios list the scenario that you want to use for the design.

Click "Parameters" to open the following dialog box



Prerequisite for the design is the calculation of the load combinations.
The selection of the .cmb file that was stored after the analysis is performed either:

- by selecting from the list [default.cmb / EC-8_Greek Dynamic (1).cmb / total.cmb] which automatically performs the calculation

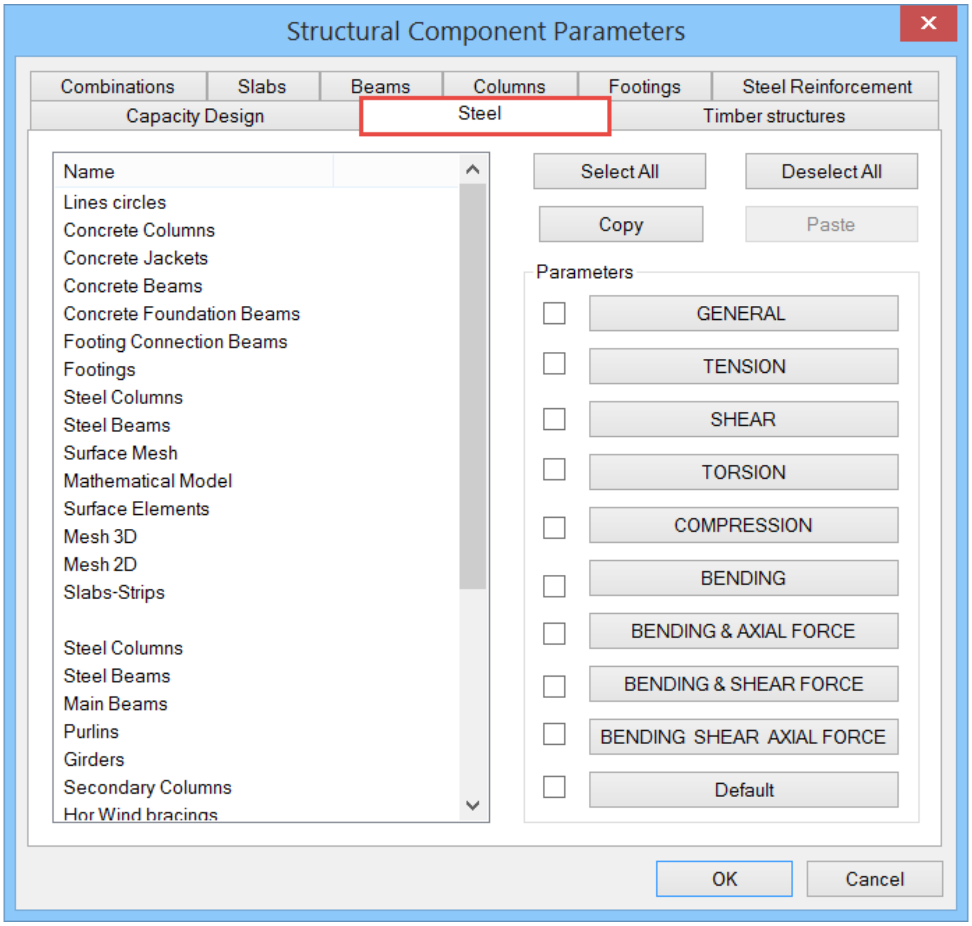
- or through the [Insert Combinations] command, which is located inside the project folder, you select from the existing ones the desired combination file according to which the design will be performed and next click the [Combinations Calculation] button.

⚠ *In this example, the combination file of the dynamic analysis, including wind and snow will be used.*

The **Plates, Beams, Columns, Footings, Steel Reinforcement** tabs, include the parameters that affect concrete sections.
For steel structures, to define the design parameters of the steel sections, select the “Steel” field.

The dialog box that opens is divided into two areas: on the left, there is a list with all the layers and on the right, there is the checks list including all of the respective parameters.
First, select a layer. Click one from the list, or more using “ctrl”, or all using “Select All”.
(By pressing the button “Deselect All” cancel the previous layers’ selection.)
Then activate one or more design checks with a tick on the corresponding checkbox and press the corresponding button to specify the parameters.

The parameters defined for one layer can be copied to other layers, using the command "Copy".
Select a layer → define the parameters → press “Copy” → select another layer → press “Paste”.



The definition of the design parameters for steel sections is performed per layer. First, you select the layer of which the parameters are to be defined, (for example Steel Columns) and for each check category (General, Tension, Shear etc.), you set the respective parameters. As soon as you defined the parameters for one layer, the program gives you the ability to copy these parameters to another layer using the Copy - Paste commands.

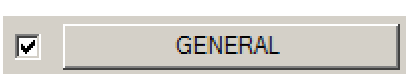
Suppose you have set all parameters for the layer Steel Columns and you want to pass these parameters to Steel Beams. Activate the checkbox next to "Default" and automatically all parameters become selected. Then press "Copy", select layer Steel Beams and press "Paste" (which is now activated). Now all the parameters defined for Steel Columns are defined also for the layer Steel Beams.

An alternative method to set the same parameters to all layer including steel sections is selecting all layer pressing "Select all" button and set once the parameters for each check category.

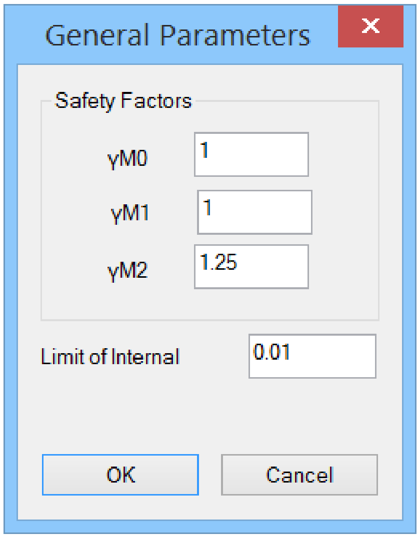
Note that at least one (or more) layer should be selected to set parameters.

Next, all the parameters for each category are analytically explained.

- By clicking the button "**GENERAL**" the following dialog box opens:



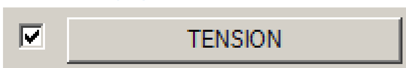
to set the γ_{Mi} safety factors:



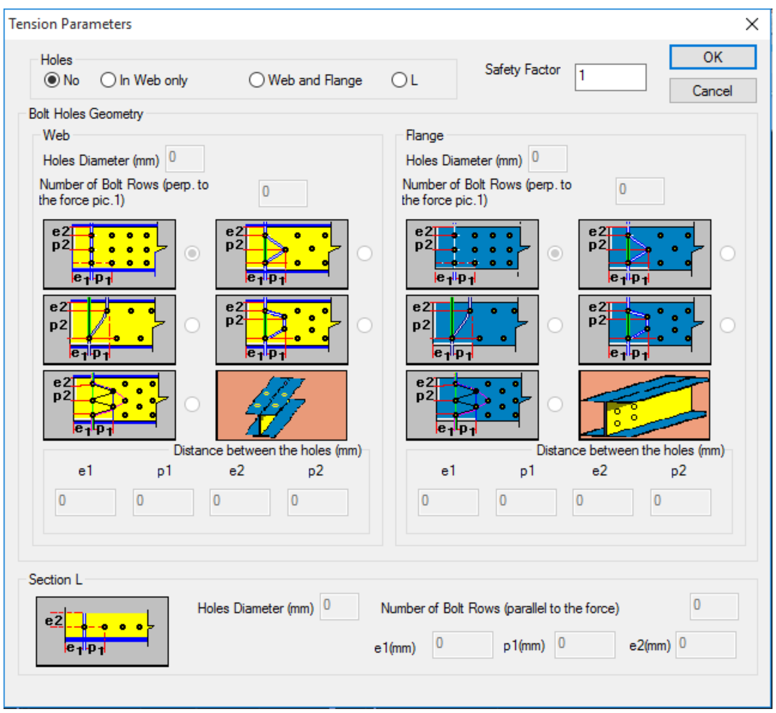
γ_{M0} : partial factor for resistance of cross-sections whatever the class is
γ_{M1}: partial factor for resistance of members to instability assessed by member checks
γ_{M2}: partial factor for resistance of cross-sections in tension to fracture

In the “Limit of Internal” field define an upper limit. Under this value, the program will not consider the corresponding stress resultants. These values are recommended by Eurocode.

- “**TENSION**”



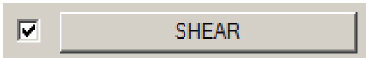
to define the parameters that correspond to the shear design check as well as the position of the hole check (EC3 §1.8 §3.5):

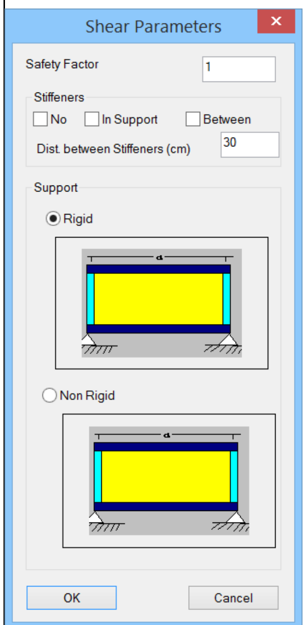


Specify the spacing of the centers of two consecutive holes, the diameter of the hole and the number of rows of bolt holes.
In case of L section specify the parameters on the bottom of the dialog box in the field “Section L”.

Here the user defines whether to consider the reduction of the tensile strength of the section due to the bolt holes of the connections or not. The data in the fields of the dialog box are derived from the design checks of the connections. For that reason, the verification of the connections must be preceded.
The safety factor for all design checks is fixed and equal to one, which means that the program calculates the ratio of the stress resultant versus the resistance. A value of the calculated ratio greater than 1.0 indicates failure.

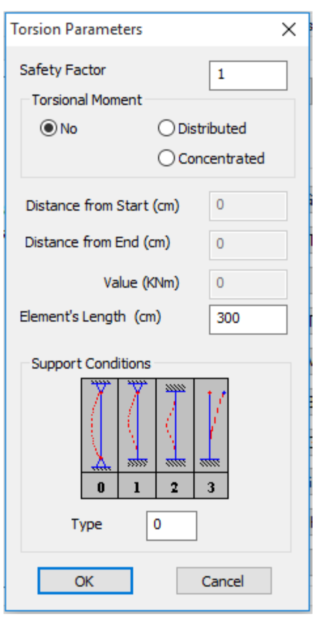
- "**SHEAR**"





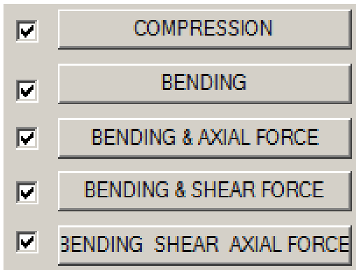
Here define if the elements of the selected layer contain stiffeners and what type; web stiffeners or intermediate stiffeners. Also define the spacing between the stiffeners and the type of the connection (rigid or not rigid).

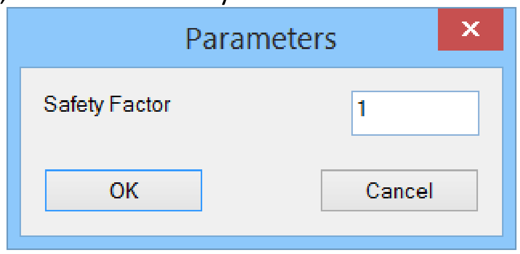
- "**TORSION**"



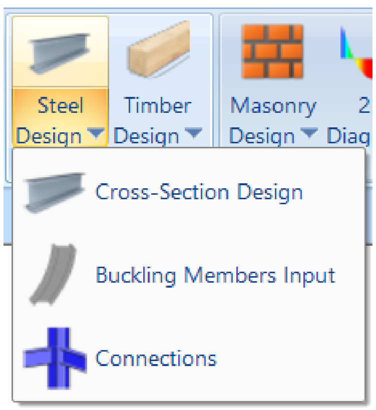
Here you define whether the structural elements of the selected layer are loaded by a distributed or concentrated torsional moment, or not. If yes you define the load data. You also define the support conditions based on the corresponding figures.

For all design checks presented in the figure on the left, define the “**Safety Factor**” in the dialog box that appears when you click one of the five buttons. The safety factor is the ratio of the resistance value versus the corresponding design value, which is set 1.0 by default.





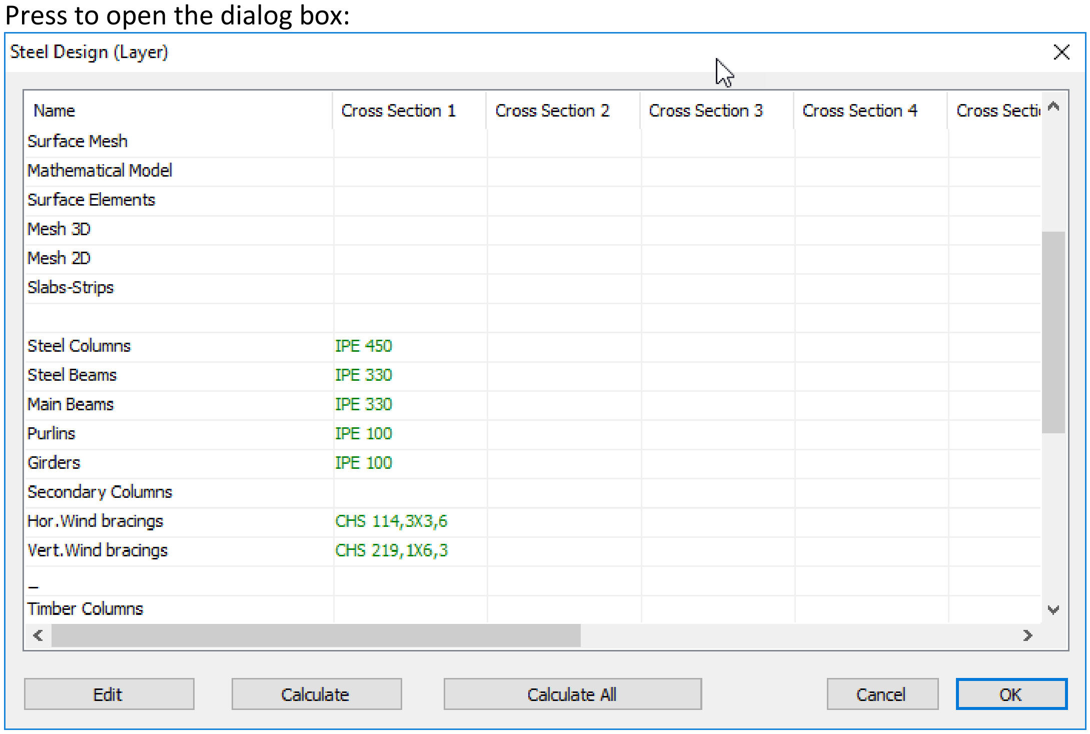
6.3 Steel members design:



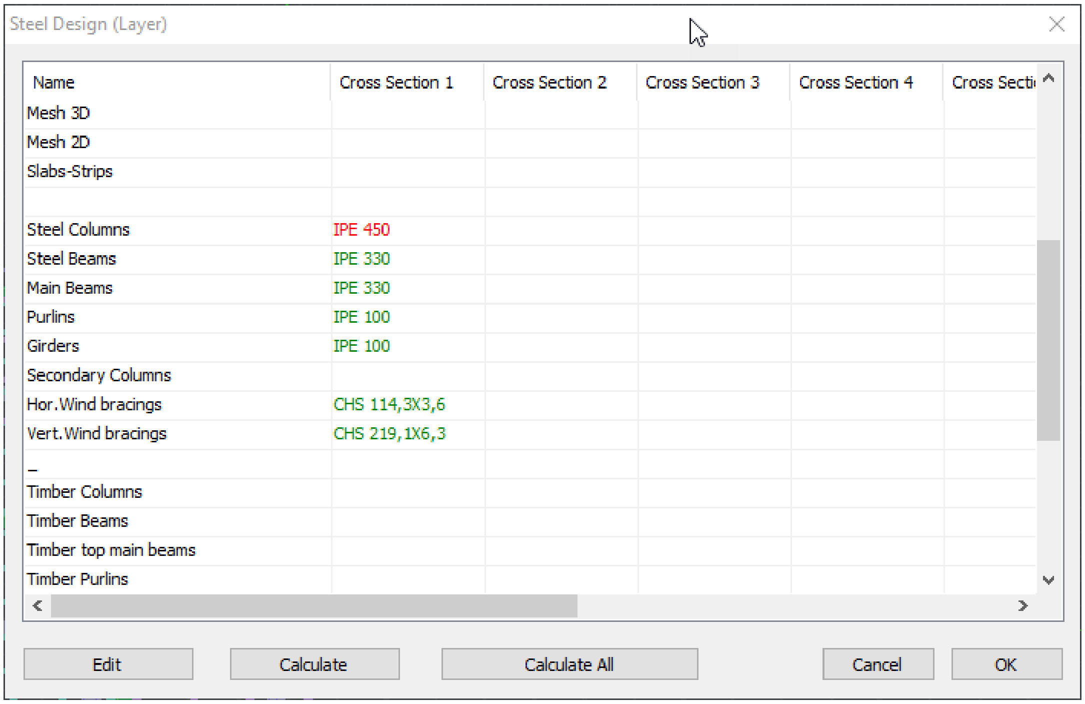
“Steel Design” command group contains commands for the cross-sections design, the buckling resistance, and the steel connections design.

6.3.1 Cross Section Design

Cross Section Design check steel cross-section resistance.
Press to open the dialog box:



displays the layers list on the right, and the corresponding cross sections of the current project.
Select “Calculate All” for all sections calculation.
Alternatively, select the layers one by one and then click “Calculate”.

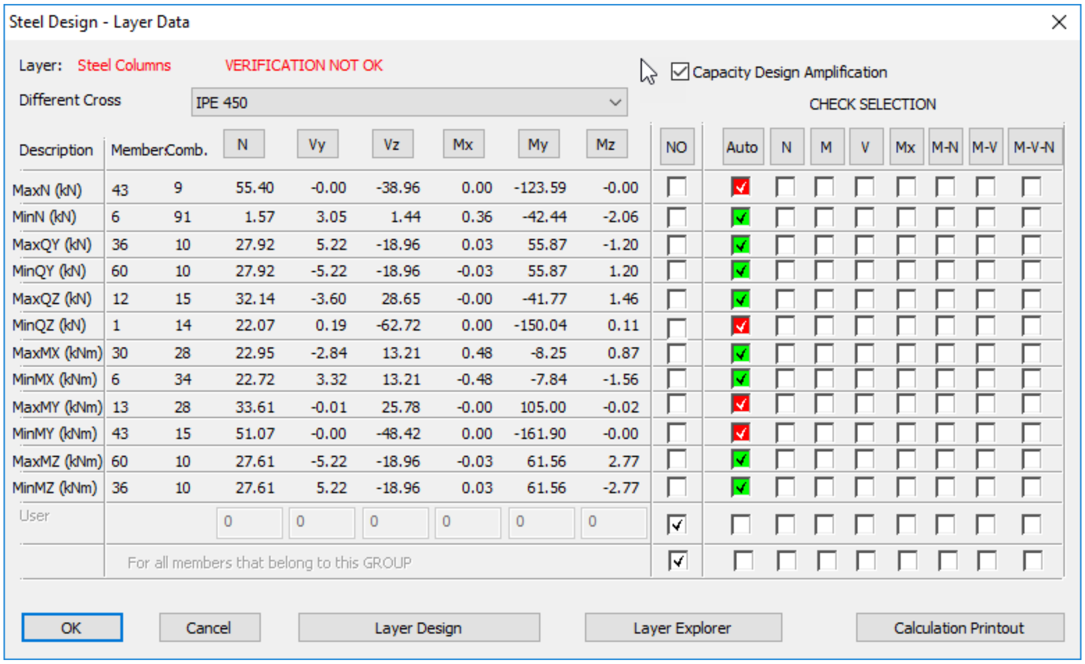


In the dialog box, the green indication on the top means that all sections of this layer are sufficient (force/resistance ≤1) and red otherwise.

To see which columns failed the checks select the layer “Steel Columns” and click “**Edit**”.
On the dialog form, you can view in a tabular format the checking results of the cross sections of the selected layer with colored values.

During the automatic process, the program locates the 12 most critical combinations of every member for each type of stress (Max N and the rest of the stress types, Min N etc.) and performs the design-checks (see Members-Design Manual).

When you mouse over a red cell, the value (greater than the unit which indicates failure) will show up.



Description	Member	Comb.	N	Vy	Vz	Mx	My	Mz
MaxN (kN)	43	9	55.40	-0.00	-38.96	0.00	-123.59	-0.00
MinN (kN)	6	91	1.57	3.05	1.44	0.36	-42.44	-2.06
MaxQY (kN)	36	10	27.92	5.22	-18.96	0.03	55.87	-1.20
MinQY (kN)	60	10	27.92	-5.22	-18.96	-0.03	55.87	1.20
MaxQZ (kN)	12	15	32.14	-3.60	28.65	-0.00	-41.77	1.46
MinQZ (kN)	1	14	22.07	0.19	-62.72	0.00	-150.04	0.11
MaxMX (kNm)	30	28	22.95	-2.84	13.21	0.48	-8.25	0.87
MinMX (kNm)	6	34	22.72	3.32	13.21	-0.48	-7.84	-1.56
MaxMY (kNm)	13	28	33.61	-0.01	25.78	-0.00	105.00	-0.02
MinMY (kNm)	43	15	51.07	-0.00	-48.42	0.00	-161.90	-0.00
MaxMZ (kNm)	60	10	27.61	-5.22	-18.96	-0.03	61.56	2.77
MinMZ (kNm)	36	10	27.61	5.22	-18.96	0.03	61.56	-2.77
User			0	0	0	0	0	0

Getting closer the mouse indicator over a green cell, displays a value <1 (sufficiency), while on a red cell, displays a value >1 (failure).

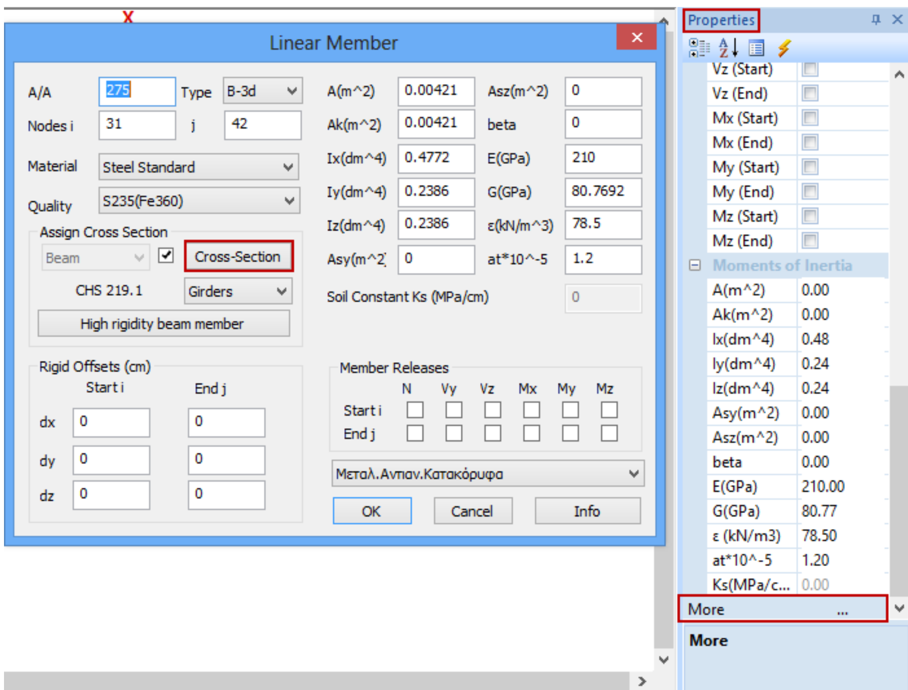
MORE DETAILS:

For each layer and each section of this layer, the program calculates, for each load combination, the max and min value of all the internal forces (N, Mx, My, Mz, Qx, Qy, Qz,). Identified the combination that gives, for ex. N max value and the corresponding member, filled under N, while the remaining cells of the same line filled with the corresponding values obtained for the same member and the same combination.
In this way filled the table with 12 lines (max and min) and six columns (6 internal forces).
-Max N …and the relative values for Mx, My, Mz, Qx, Qy
-Min N … and the relative values for Mx, My, Mz, Qx, Qy
-Max Mx… and the relative values for N, My, Mz, Qx, Qy
-Min Mx… and the relative values for N, My, Mz, Qx, Qy
-Max My… and the relative values for N, Mx, Mz, Qx, Qy
-Min My… and the relative values for N, Mx, Mz, Qx, Qy
-Max Mz … and the relative values for N, Mx, My, Qx, Qy
-Min Mz … and the relative values for N, Mx, My, Qx, Qy
-Max Qy … and the relative values for N, Mx, My, Mz, Qx
-Min Qy … and the relative values for N, Mx, My, Mz, Qx
-Max Qz … and the relative values for N, Mx, My, Mz, Qy
-Min Qz … and the relative values for N, Mx, My, Mz, Qy
“Member” column, brings the number of the member with the max or min action.
“Comb.” column, brings the number of the combination responsible for the max or min of this action.
“NO” column, allows excluding one or more max or min values obtained. To exclude, for example, max Mz and min Mz, activate in “NO” the relative checkboxes. So, for this checks, Mz max and Mz min will be excluded.

The sign condition used from the program:
The axial force with NEGATIVE sign => TENSION
The axial force with POSITIVE sign => COMPRESSION

But in. txt files there is the classic condition: (+)TENSION (-)COMPRESSION.

If you want to change the profile of a member then close the dialog box e left click on the member. In “Properties” column on the right press “More” and change the section in the respective dialog box.



6.3.2 Buckling Members Input

The buckling resistance check is one of the main design checks for steel structural members. Select the command “Buckling Members Input”, to apply on each member of each layer the following resistance checks:

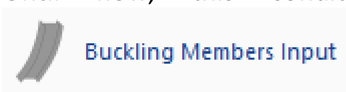
ULS (Ultimate limit state)	SLS (Serviceability limit state)
Flexural Buckling check	Member Deflection check
Torsional Flexural Buckling check	Node Displacement check
Lateral Buckling check	
Lateral Torsional Buckling check	

Merge Elements

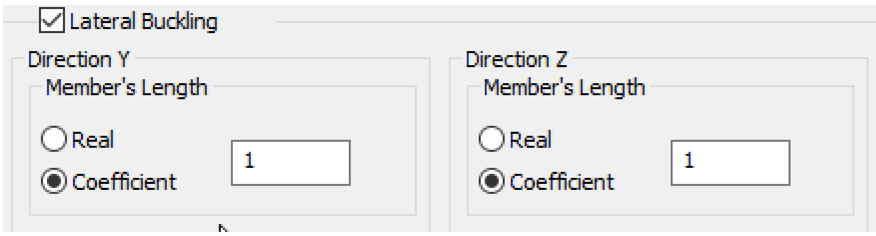
In the new version of the program, added a new command group, which concerns merging of steel (and timber) members for the calculation and buckling and deformation checks display according EC3.

IMPORTANT NOTES:

- Using this command, is now possible to define correctly, the initial length of the member per direction to be taken into account in the buckling checks. 

- Until now, this condition was considered defining the length coefficients (see 

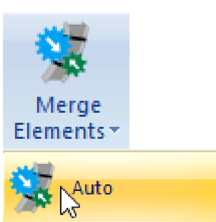
)



- Now, using merging per direction, there is no need for the coefficient process, and merging will be, in most cases, automatically.
- Also, note that with the merging process, the buckling length, is calculated correctly, and in the print outs of the results a merged element is printed once with annotation of the individual members that contains.
- Basic concepts of buckling along major and minor axies you can find in the User's Manual Chapter "9. Members Design"

NOTE:
Generally, making a rule, we could say that, we consider the **merged length Ly** in the direction where the local axis y-y is parallel to the supporting elements. While in the other direction, if no supporting elements, **Lz** is **the length of each member**.

Press Merge Elements command and then Auto:



Merge elements means that, either automatically or manually, the individual parts of a single element, merge in each buckling direction.
Meaning that, the buckling length is considered computationally, not the actual length of the element, but the unified from the beginning to the end of the column or beam, respectively.

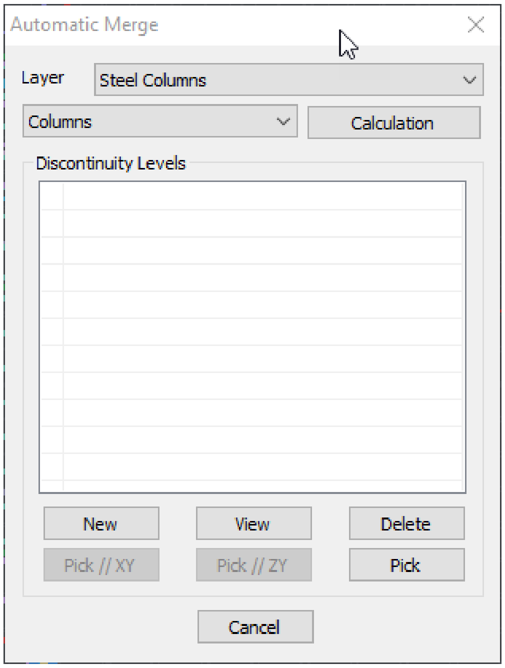
Also, in the presentation of the results, for these merged elements, the worst results display only once and not for each part, as it was so far.

Finally, in automatic merging, there is the definition of discontinuity levels, horizontal or vertical, used as merging boundaries of a continuous element.

Discontinuity levels are levels that are boundaries of beams and columns, used to break merging in each direction.

Auto merge

Using this command displays the following dialog box



First, choose the layer of the elements to merge.

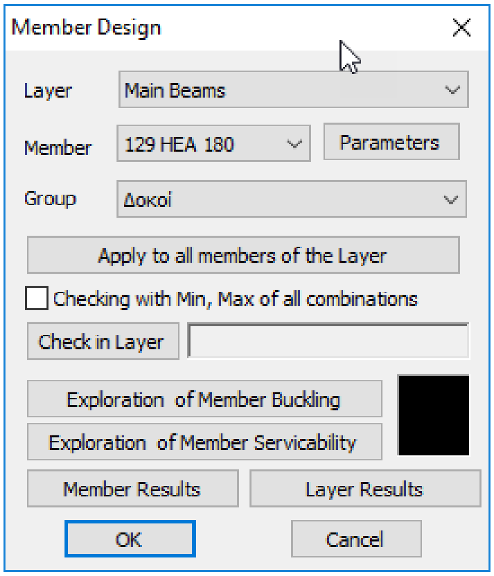
Just below, specify the type of element contained in the selected layer.
The program automatically understands the type of the element: Column if vertical, Beam all the others.

Merged elements display colored:

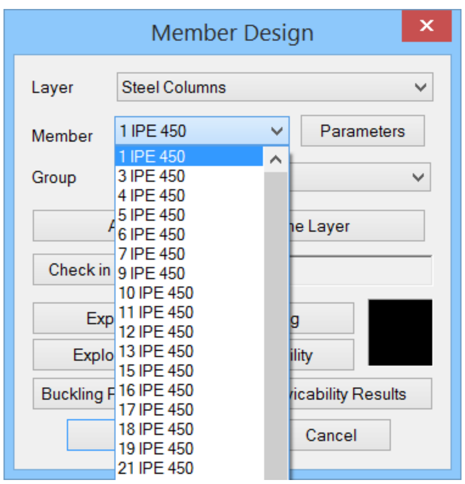
- Yellow color for the merged elements along y-y local axis
- Cyan color for the merged elements along z-z local axis
- Pink color for the merged elements along both local axes

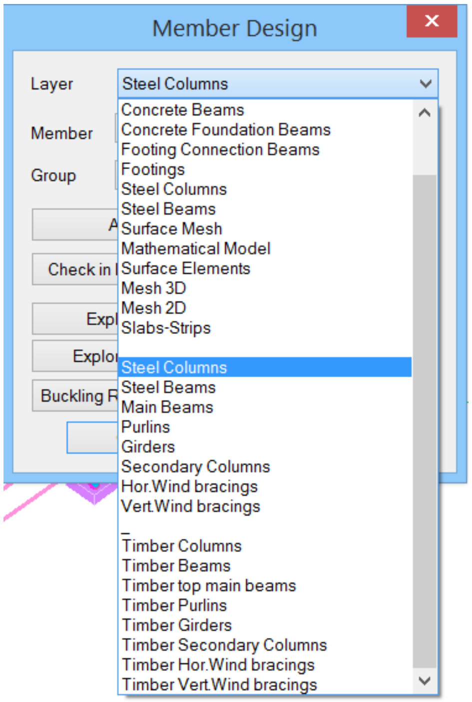
Press **"Calculation"** and the program merge the elements of the active layer, based on what was mentioned above.

Selecting the command opens the following window:

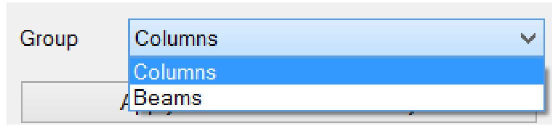


Layer performs checking. So first select the layer from the drop-down list and the "Member" list loads all members of this layer and its cross-sections.



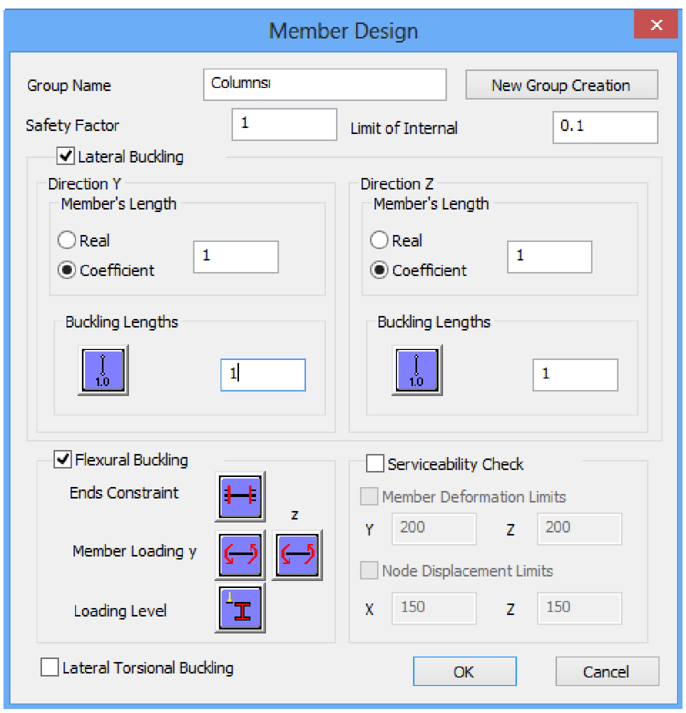


Select from the drop-down list the layer "Steel Columns". In the "Members" list all the structural members that belong in the selected layer are displayed. If you want to define different parameters to some of them, you can create different "Groups" in the same layer.
The program has two default Groups: "Beams" and "Columns".



If you want to apply the same parameters to all members of the layer, then set the parameters once, keep the default name "Columns" and press the "Apply to all members of the layer".
Calculations will consider the same parameters for all members of the layer.
Otherwise, to set different parameters for some of the members of the layer, the procedure that should be followed is explained below. But first, let's see how to set the parameters.

Select a "Layer" and click on the "Parameters", and the following dialog box opens:

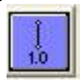


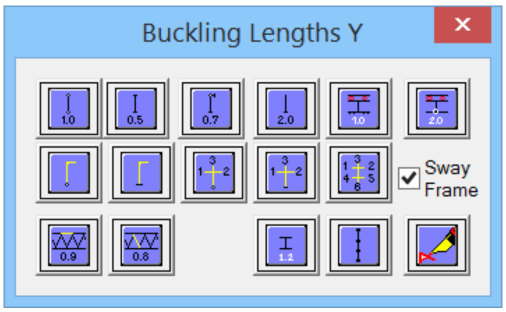
In the "Group Name," you see the name of the parameter group. If you want to create your group, give a new name and press the button "New Group Creation".
In the "Safety Factor," you can set the limit for the program for the design checks: the intensive forces to the respective strength of the member. The default value is 1.
The "Limit of internal forces" is the limit that the program uses to take into consideration (or to ignore) the intensive sizes.

The rest of the form is divided into four parts, one for each check:

- For Lateral Buckling check: Because of the “Merging” of the elements, there is no need anymore to define the Member’s Length. The program will consider the length resulting after merge.

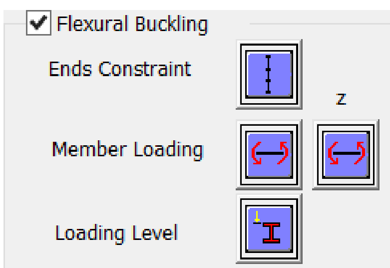
The parameter "Buckling Lengths" depends on the support conditions of the member.

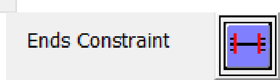
Click on the following button  to open the following list and select the appropriate conditions so that the program automatically inserts the corresponding factor.



The icons are divided into two groups:
The first group includes icons with a specific factor depending on the member support conditions

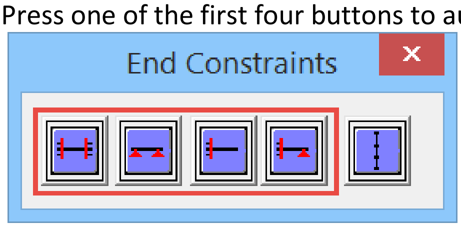
- Flexural Buckling resistance check:



Activate the checkbox and press 

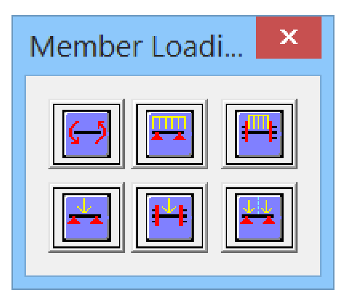
The “End Constraints” window, containing the various types of constraints opens.

Press one of the first four buttons to automatically calculate the flexural buckling factor:



The next parameter 

refers to the load type of the member at the local axis y, and z respectively. By selecting the corresponding icon, the following options appear:

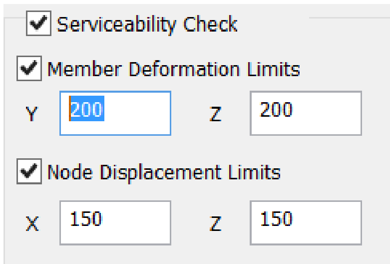


Where you choose the type of Member Loading.

- For Lateral Torsion Buckling check: activate the checkbox.

⚠ **NOTE**: For the lateral buckling and the lateral torsion buckling resistance check, the parameters are the same.

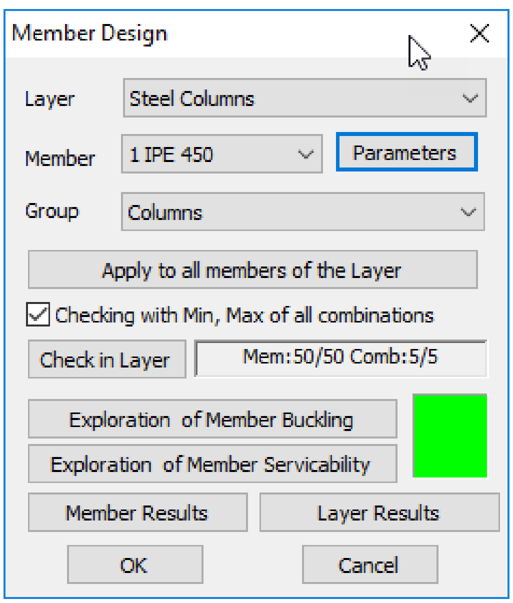
➢ For Serviceability checks: activate the checkbox “Serviceability Check” and the checkboxes “Member Deflection Limits” and “Node Displacement Limits”.

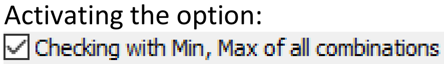


Then type the corresponding values in each direction, X, and Z. For example in the figure on the left, the limits are defined as l/200 and l/150, where l is the member’s length.

Finish the parameters’ input and then press the button "OK" to return to the previous dialog box.

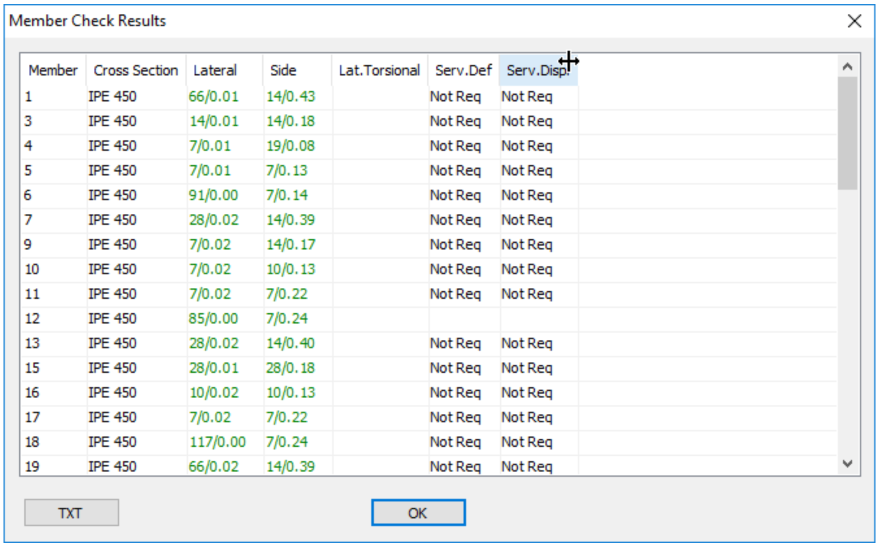
To apply the parameters that you set to all members of a layer, select the command "Apply to all members of the Layer".
Click the button "Check in Layer" to check every member of the current layer, for every load combination. The results of the design checks are displayed in the black window that becomes green if it the checks are satisfied with all members of the active layer and red, if not.



Activating the option:


, in checks will be taking into account only the maximum and minimum values of the intensive forces resulting from all combinations, excluding the intermediate values so that the process will be completed at noticeably shorter times.

Double click ok the colored window, opens the dialog box containing members check summary results:



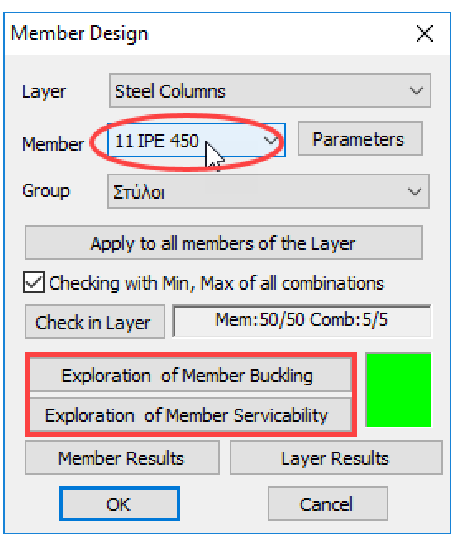
Member Check Results

Member	Cross Section	Lateral	Side	Lat.Torsional	Serv.Def	Serv.Disp
1	IPE 450	66/0.01	14/0.43		Not Req	Not Req
3	IPE 450	14/0.01	14/0.18		Not Req	Not Req
4	IPE 450	7/0.01	19/0.08		Not Req	Not Req
5	IPE 450	7/0.01	7/0.13		Not Req	Not Req
6	IPE 450	91/0.00	7/0.14		Not Req	Not Req
7	IPE 450	28/0.02	14/0.39		Not Req	Not Req
9	IPE 450	7/0.02	14/0.17		Not Req	Not Req
10	IPE 450	7/0.02	10/0.13		Not Req	Not Req
11	IPE 450	7/0.02	7/0.22		Not Req	Not Req
12	IPE 450	85/0.00	7/0.24			
13	IPE 450	28/0.02	14/0.40		Not Req	Not Req
15	IPE 450	28/0.01	28/0.18		Not Req	Not Req
16	IPE 450	10/0.02	10/0.13		Not Req	Not Req
17	IPE 450	7/0.02	7/0.22		Not Req	Not Req
18	IPE 450	117/0.00	7/0.24		Not Req	Not Req
19	IPE 450	66/0.02	14/0.39		Not Req	Not Req

The first column indicates the number of the member, the second column indicates the cross-section and in the next five columns, the least favorable ratio of strength and the combination number from which this ratio resulted is displayed.
Greens are the ratios below unity and red the ratios above it.
"Not Required" means that there is no corresponding size or that the intensive axial force is tensile and not compressive.

NOTES:
⚠ The check for the three types of buckling is performed for each member and all combinations.
For each group of (N, My and Mz) the checks are made four times based on the following combinations:
N with min My and min Mz
N with min My and max Mz
N with max My and min Mz
N with max My and max Mz

⚠ That's why in the output results and the exploration text the number of the combination has two numbers: The first is the number of the combination and the second refers to the number for each of the four previous cases.



Selecting the Exploration of Member (Buckling /Serviceability) open the files containing the analytical results of all checks for all combinations for the active member.

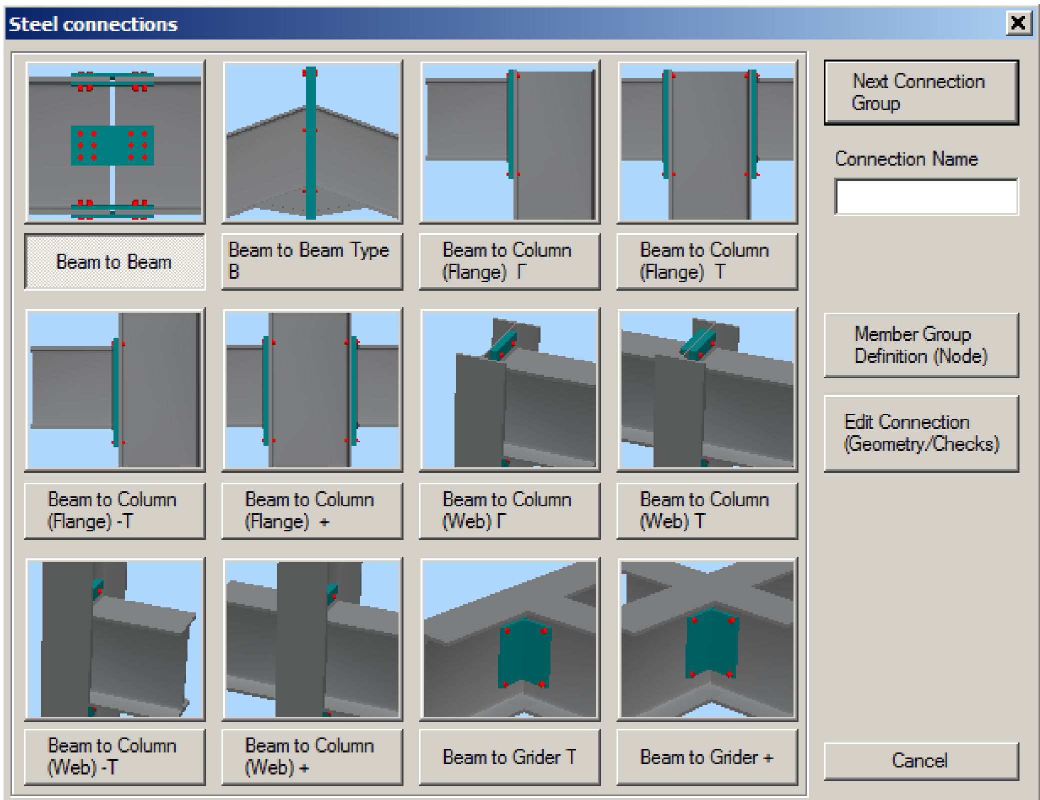
Selecting Results to open the files that include the summary results of the checks on the active member Member Results and all members of the active layer Layer Results.

7. Connections

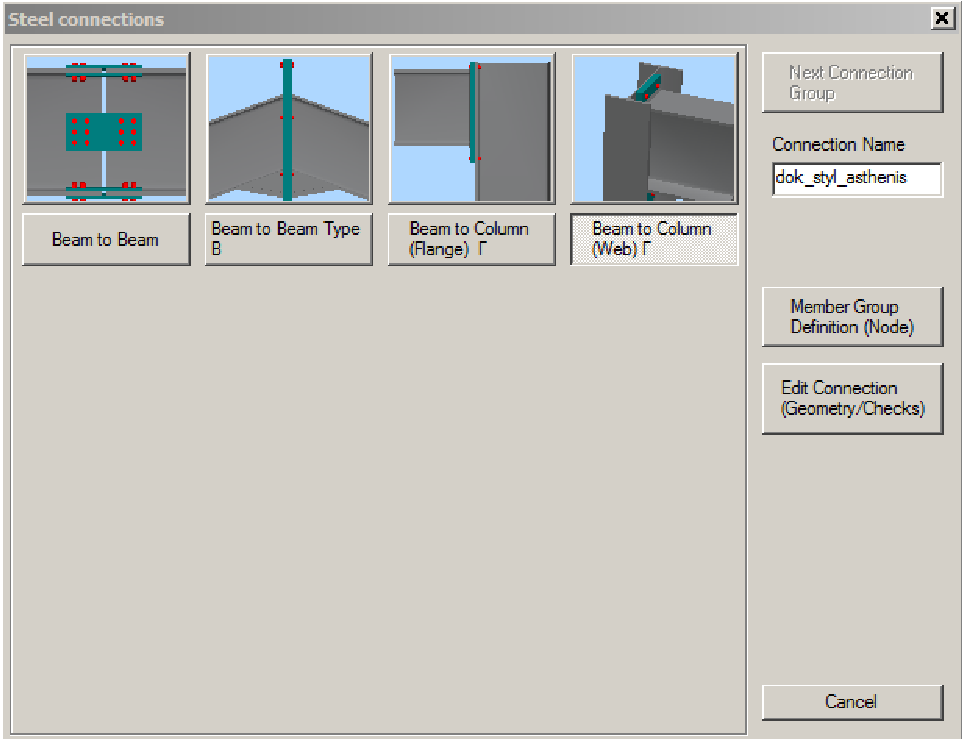
7.1 Create and check steel connections:

The last command of "Steel Members Design" is "Steel Connections", used for connections in design. Select the command and choose the next step:

A) Right click on the screen to open the library containing all the available connections and select the appropriate. (Click on "Next Connection Group" to see more connections).



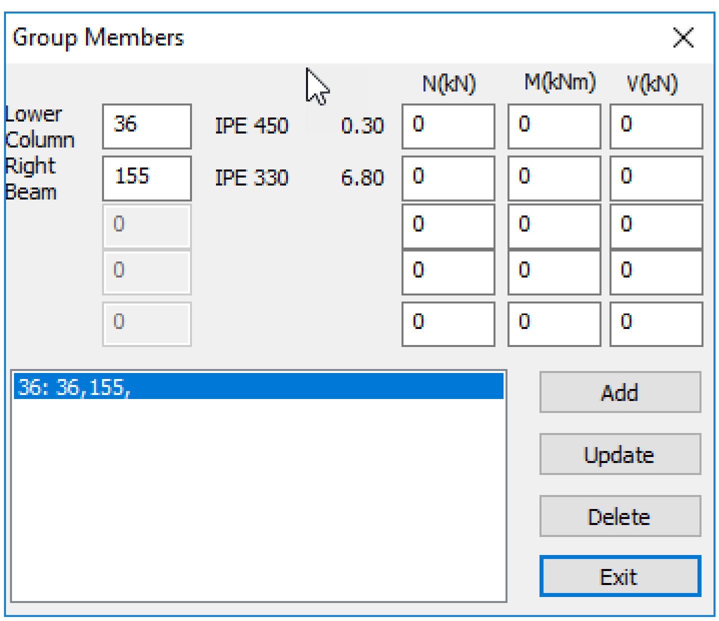
B) Or, select by left click, the members that you want to connect. Then right click to open a library that contains only the more adapt connections for the selected members.



Left click to select the column member and then the beam member, and right click to open the library with the four possible types of connection. Select the last one “Beam to Column (Web) Γ” along the main axis. Next set the name of the current connection.

⚠ The name must contain only characters from the Latin alphabet and no spaces between the words are allowed.

Then, select the “Member Group Definition (Node)” command
and in the dialog box, you can add more groups of members with the same connection features (i.e. column – beam) or type your values for the stress resultants N, M, V for the existing groups.

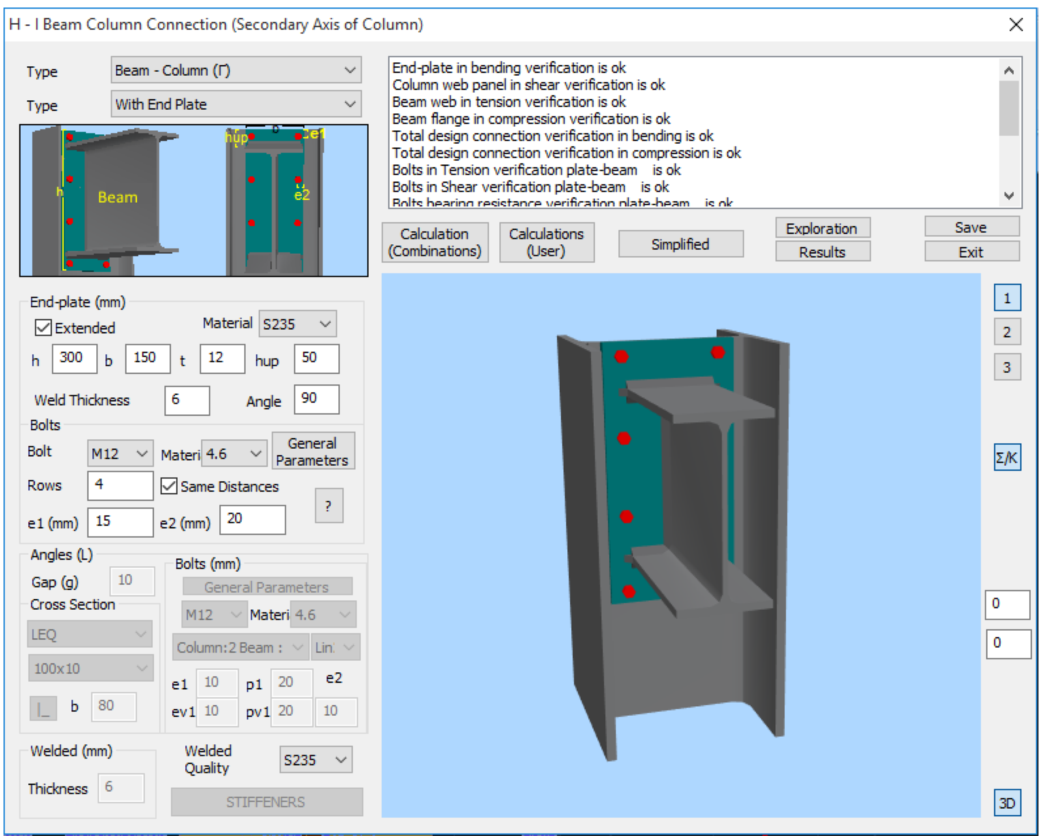


To add groups of members, click into the field “Lower Column” and pick the column. Then click on the field “Right Beam” and pick the beam (or just enter the numbers in the corresponding fields) and then click the button “Add”.
Use this dialog box for the design of steel connections with the same type and the same cross-sections (column IPE 450 - beam IPE 330).

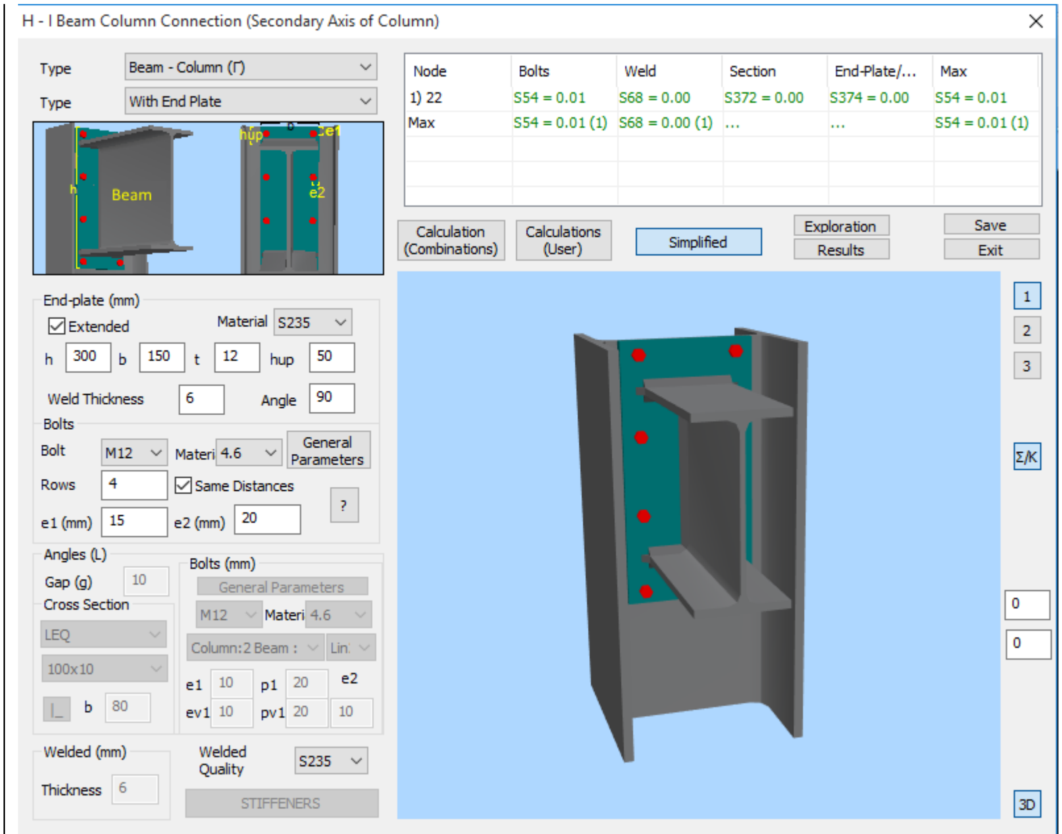
The program calculates automatically the forces and proceeds with connection’s design, based on the less favorable load combination. So you don’t have to guess the point of your structure, where the less favorable beam-column connection in the minor axis will be developed. Furthermore, if this connection is satisfied, then all the other connections with the same type will be automatically satisfied, too.

In the end, click “Exit” and select the command “Edit Connections (Geometry/Checks)”. In the new dialog box, you can define the type and the geometry of the specific connection. Select the type and enter the geometrical parameters of the cross-section or create your connection.

First, the program performs the geometrical checks of the connection (e.g. if the bolts are located too close to the edge of the plate). If there is a problem, the corresponding error message appears in the field on the right. In the specific connection, change the distance e1 from 10 to 15 cm and then click again the button "Calculation (Combinations)".



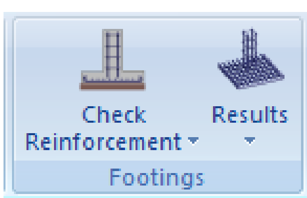
Click the button “3D” to see a three-dimensional representation of the connection that is updated as you change the parameters.
The buttons “1”, “2”, “3” are used for the display of the two side views (1 & 2) and the plan view (3). The button “Σ/K” is used for the display of the three-dimensional representation of the welds and bolts.



8. FOOTING DESIGN

8.1 How to perform footing design:

As soon as you complete the connection design, you can move on to the footing design.



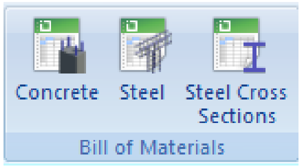
The "Footing" command group contains commands for footing design check, design calculation, editing and the respective results.

Select the command "**Check Reinforcement>Overall**" to perform the design checks for all the footings on the current level.
The color of the node indicates that the design checks of the footing:

 were satisfied or  failed.

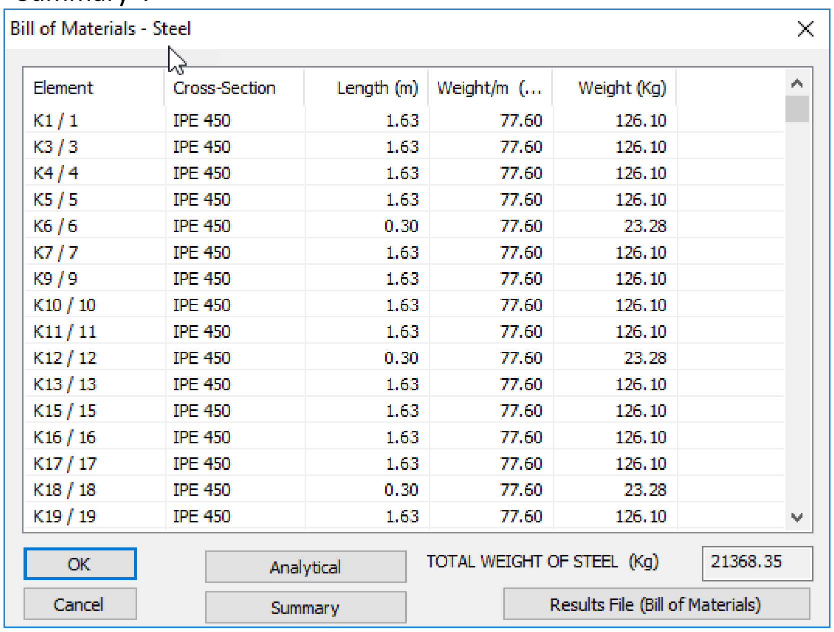
⚠ A necessary precondition for the footing designing is the columns designing in level 1.

9. BILL OF MATERIALS



The "Bill of Materials" command group contains the commands related to the estimation of the materials' quantities and the corresponding cost.

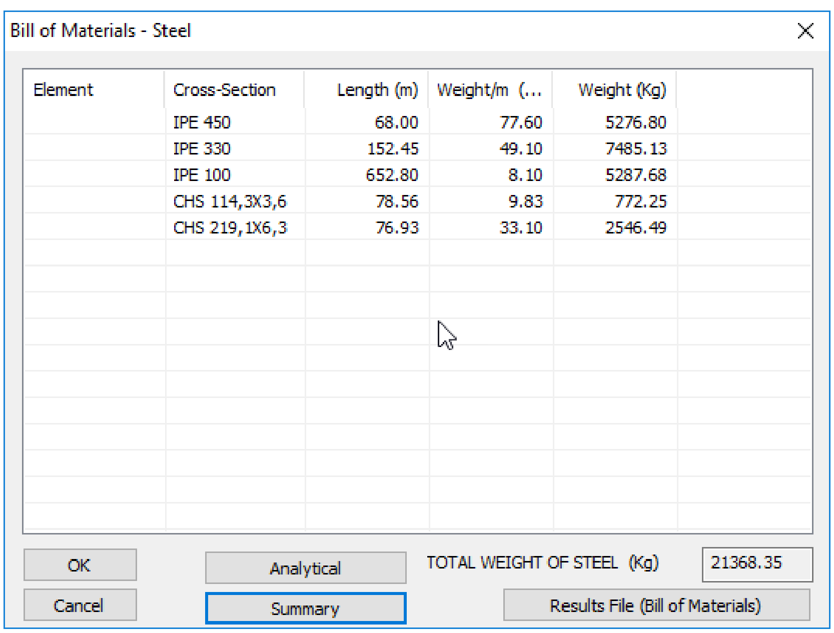
Steel Cross Sections: It calculates the quantity of the structural steel.
"Analytical": per element and cross-section concerning the length (m), weight in Kg (per m or in total); "Summary".



Bill of Materials - Steel

Element	Cross-Section	Length (m)	Weight/m (...	Weight (Kg)
K1 / 1	IPE 450	1.63	77.60	126.10
K3 / 3	IPE 450	1.63	77.60	126.10
K4 / 4	IPE 450	1.63	77.60	126.10
K5 / 5	IPE 450	1.63	77.60	126.10
K6 / 6	IPE 450	0.30	77.60	23.28
K7 / 7	IPE 450	1.63	77.60	126.10
K9 / 9	IPE 450	1.63	77.60	126.10
K10 / 10	IPE 450	1.63	77.60	126.10
K11 / 11	IPE 450	1.63	77.60	126.10
K12 / 12	IPE 450	0.30	77.60	23.28
K13 / 13	IPE 450	1.63	77.60	126.10
K15 / 15	IPE 450	1.63	77.60	126.10
K16 / 16	IPE 450	1.63	77.60	126.10
K17 / 17	IPE 450	1.63	77.60	126.10
K18 / 18	IPE 450	0.30	77.60	23.28
K19 / 19	IPE 450	1.63	77.60	126.10

OK | Analytical | TOTAL WEIGHT OF STEEL (Kg) 21368.35
Cancel | Summary | Results File (Bill of Materials)



Bill of Materials - Steel

Element	Cross-Section	Length (m)	Weight/m (...	Weight (Kg)
	IPE 450	68.00	77.60	5276.80
	IPE 330	152.45	49.10	7485.13
	IPE 100	652.80	8.10	5287.68
	CHS 114,3X3,6	78.56	9.83	772.25
	CHS 219,1X6,3	76.93	33.10	2546.49

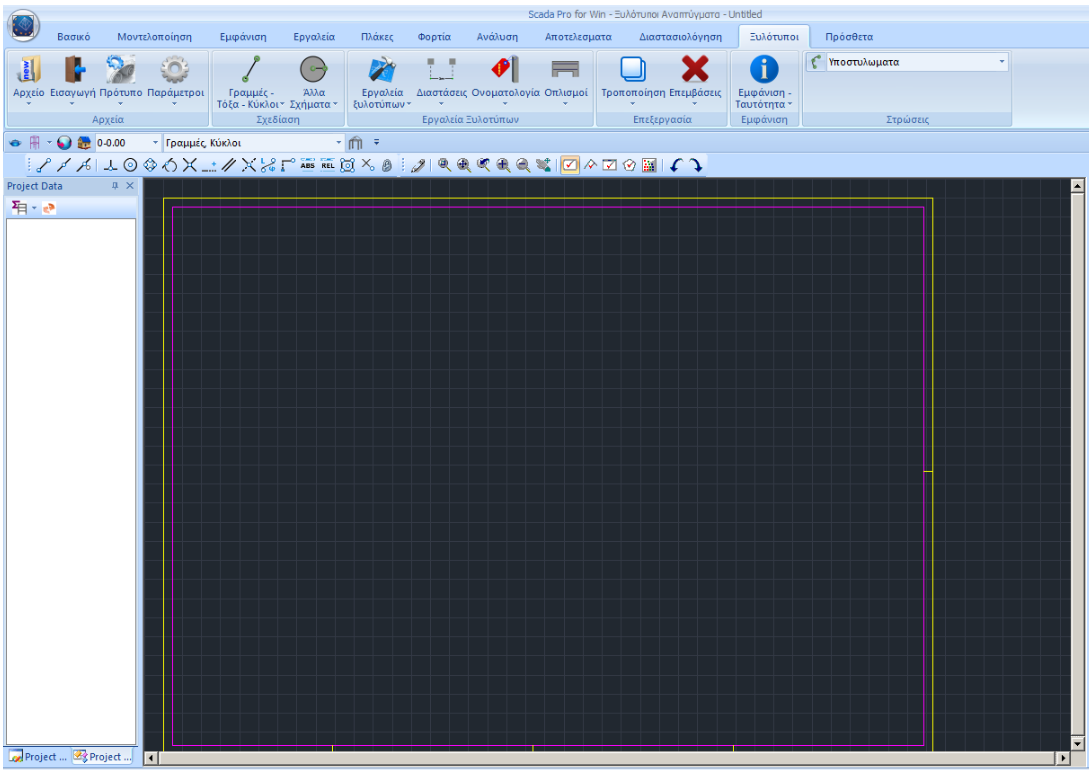
OK | Analytical | TOTAL WEIGHT OF STEEL (Kg) 21368.35
Cancel | Summary | Results File (Bill of Materials)

SCADA Pro gives you the ability to have analytic bills of materials for each steel cross section per member or aggregated bills per section category.
Press "Results File (Bill of Materials)" to attach the Calculation Printout.

10. DRAWING

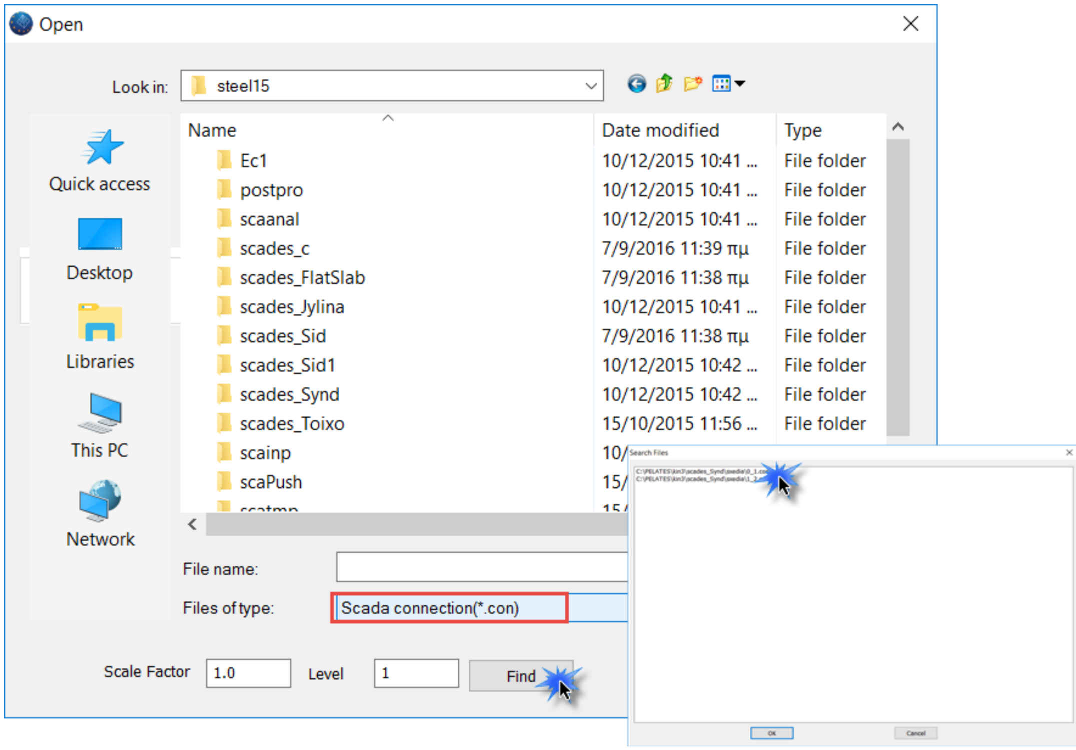
After Members Design and reinforcement editing for concrete structures or connections creation for metal projects, in "Drawing-Detailings" Tab you can open, modify and finally produce all the designs.

“Drawing-Detailings” Ribbon incorporates a designing application in the same interface

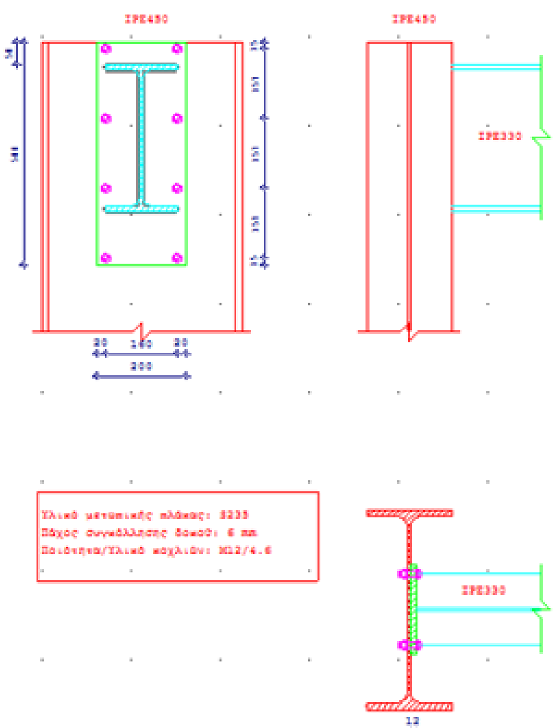


Insert connections drawing:

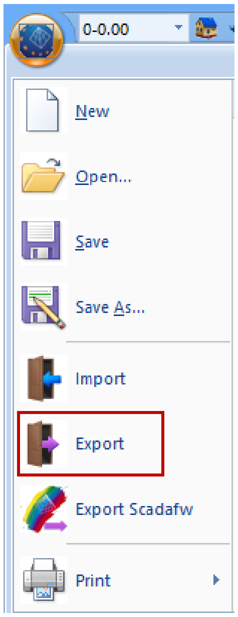
Selecting “Import” command opens the following dialog box for choosing the project’s folder.
Then select:
In **List files of Type: select “Scada connection *.con”**
(In **Directories:** find the path**C:\scadapro\“STEEL” \scades_Synd\sxedia)**

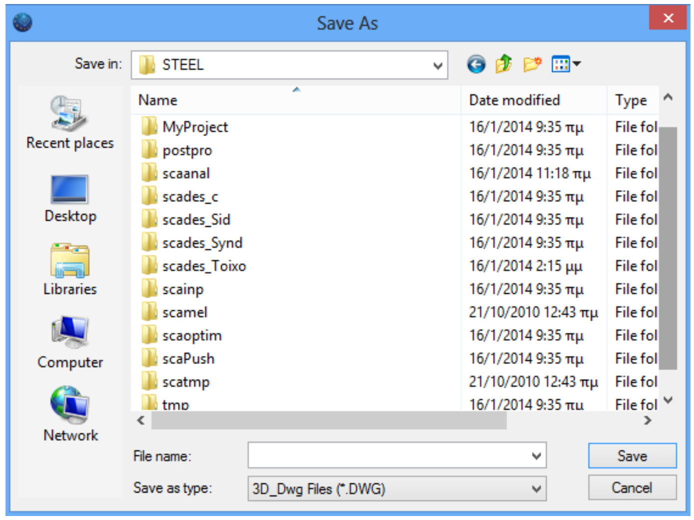


Then select the connection name (become blue), press "ok" and then click on the desktop to define the insert point of the drawing. Automatically created a floor plan and two sides of the connection details selected.

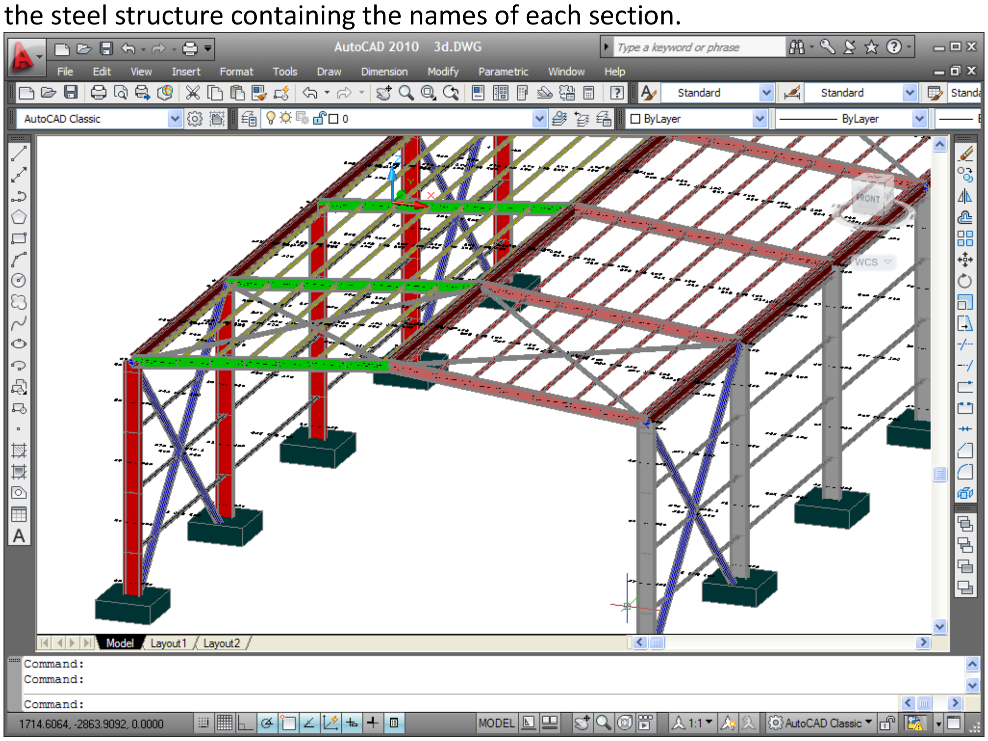


Following this procedure, you can produce a total of over 120 different types of connections covered by the program.
To create respective views, floor plans and sections of the total body of the steel structure will follow a different way:
From "File" button/press "Export" that opens a new dialog box:





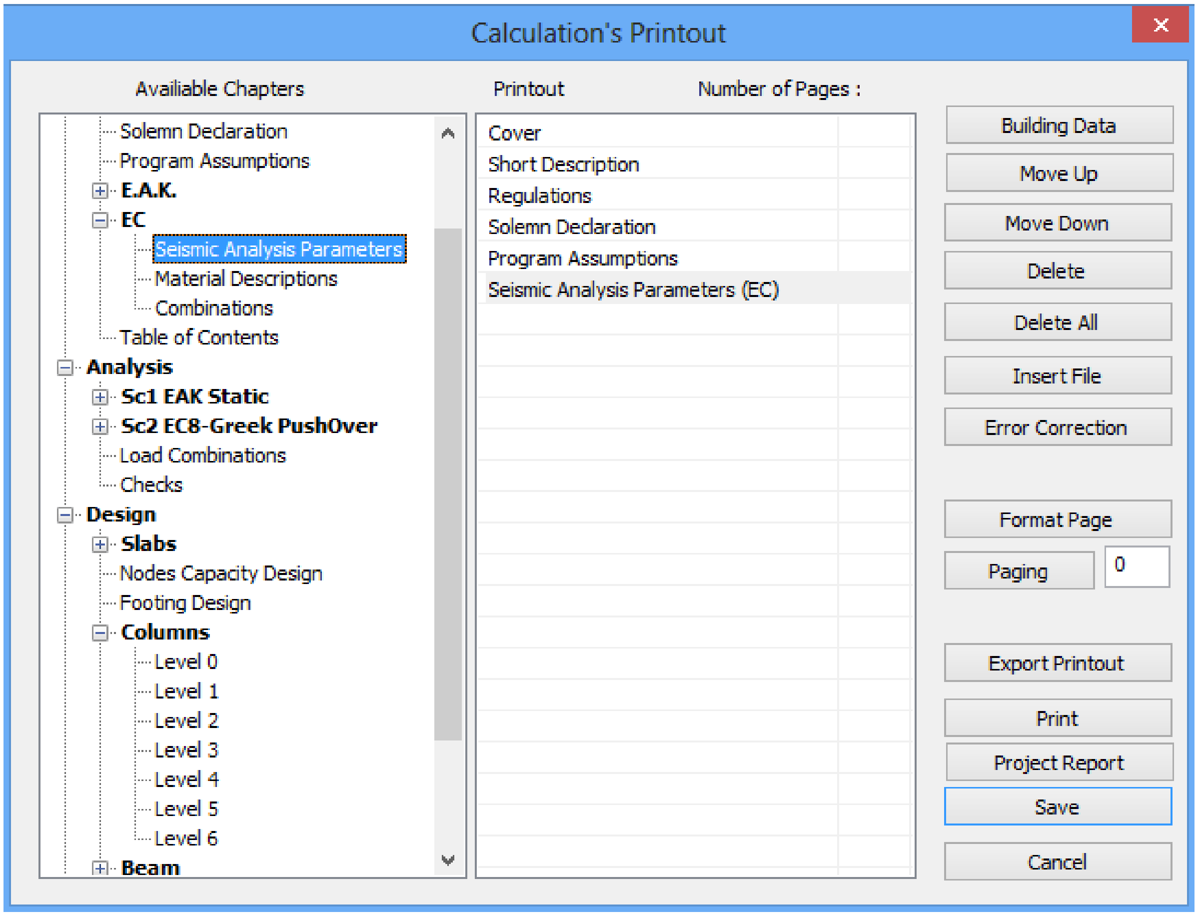
Allowing exporting a SCADA Pro file in *. Dwg format. In the field "save in" select the projects folder for a three-dimensional extraction of the structure. Type a name in the File name field and then in "save as type" choose the form 3D_dwg Files (*. DWG).
Opening the generated * dwg file with a cad program you receive a detailing three-dimensional model of the steel structure containing the names of each section.



11. CALCULATIONS PRINTOUT

Open “Add-ons” Tab and press Print command.
In the dialog box “Calculation Printout” on the left, displays the list with the Available Chapters. Double click on the selected chapter to bring it in the right list.

Complete the Printout list clicking two times on the available chapters and press Project Report.



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