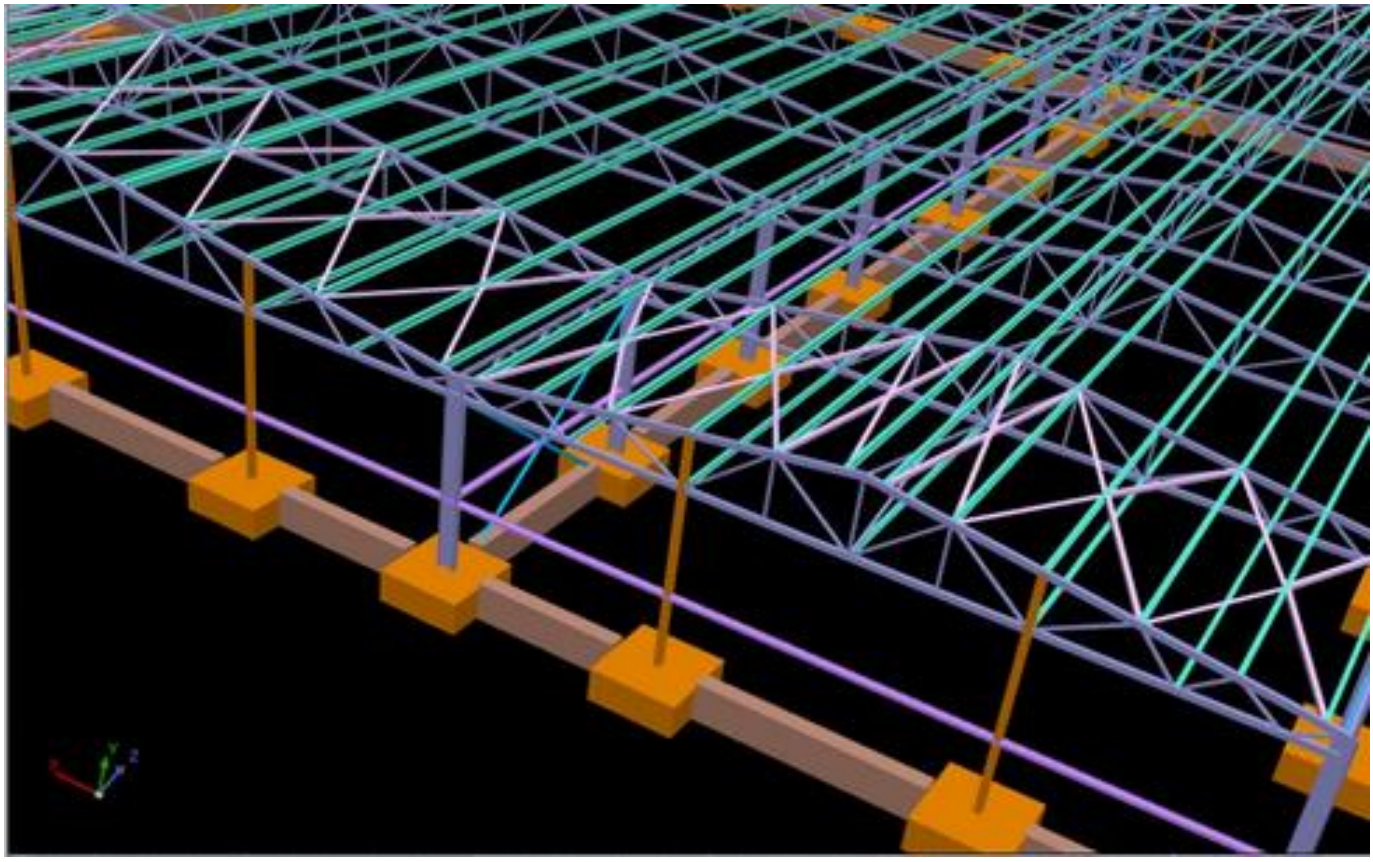
SCADA Pro™ 18
Structural Analysis & Design

User's Manual

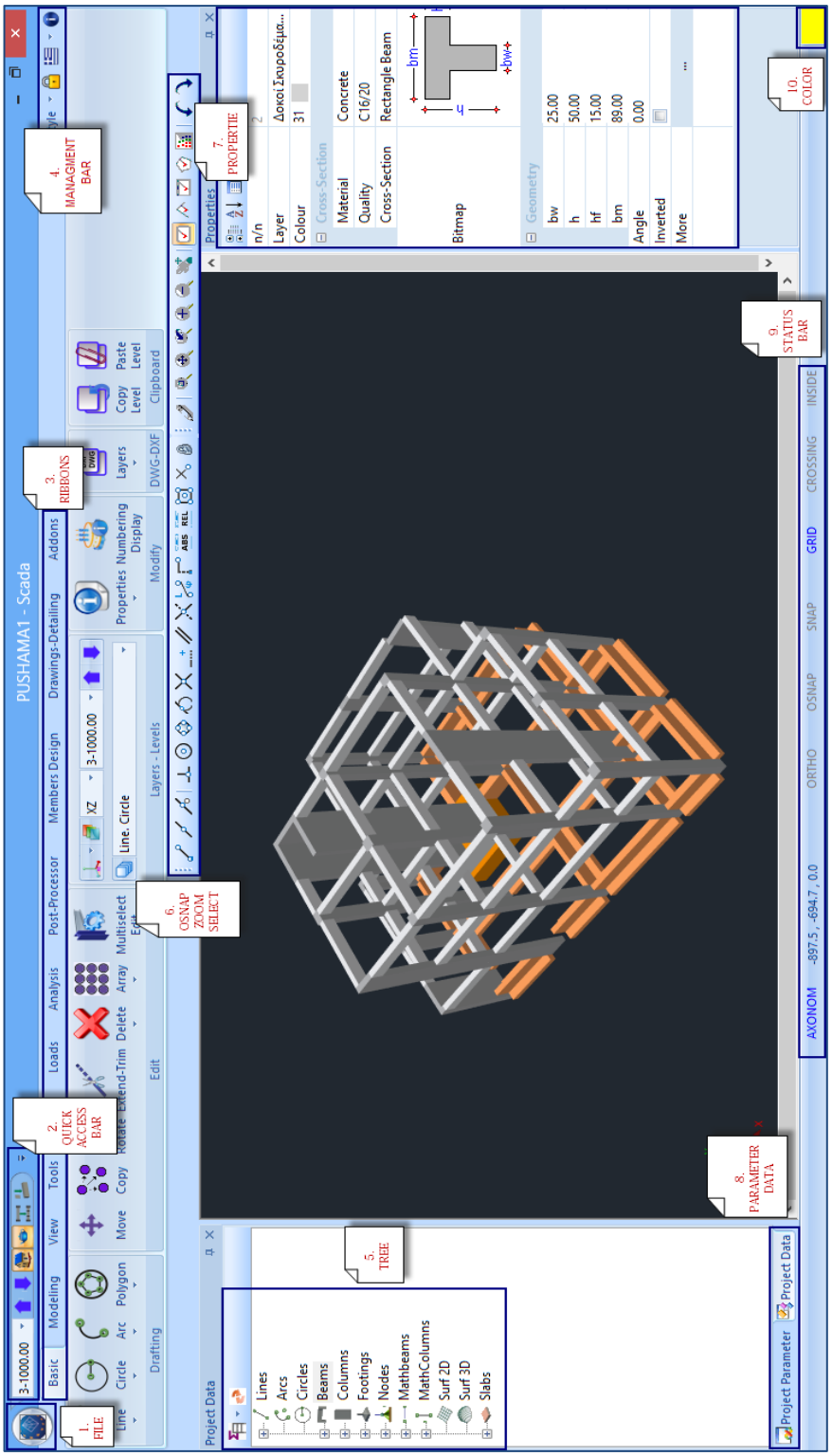
9B.MEMBERS DESIGN
SBC306_17_STEEL



ACE HELLAS
INTEGRATED SOLUTIONS

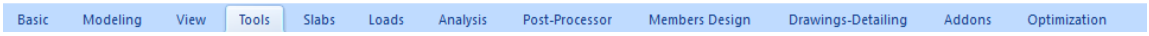
CONTENTS

I. THE NEW UPGRADED INTERFACE of SCADA Pro

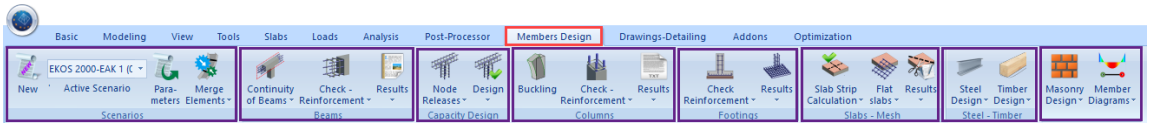


II. DETAILED DESCRIPTION OF THE NEW INTERFACE

In the new upgraded SCADA Pro, all program commands are grouped in 12 Units.



Members Design



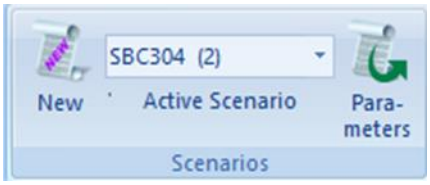
The 9th Unit entitled "Members Design" contains the following 8 groups of commands:

1. **Scenarios**
2. **Beams**
3. **Capacity Design**
4. **Columns**
5. **Footings**
6. **Slabs - Mesh**
7. **Steel**
8. **Timber**
9. **Masonry Design – 2D Diagrams**

⚠ Since model analysis has been completed, the design checks of the structural elements are applied according to the design code provisions, defined in the tab "Member Design". The reinforcement of the structural elements is calculated according to the design checks.

---

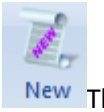
1. Scenarios



The "Scenarios" command group contains the commands for the creation of a new scenario as well as the editing of the parameters of the design checks and reinforcement in every type of structural elements.

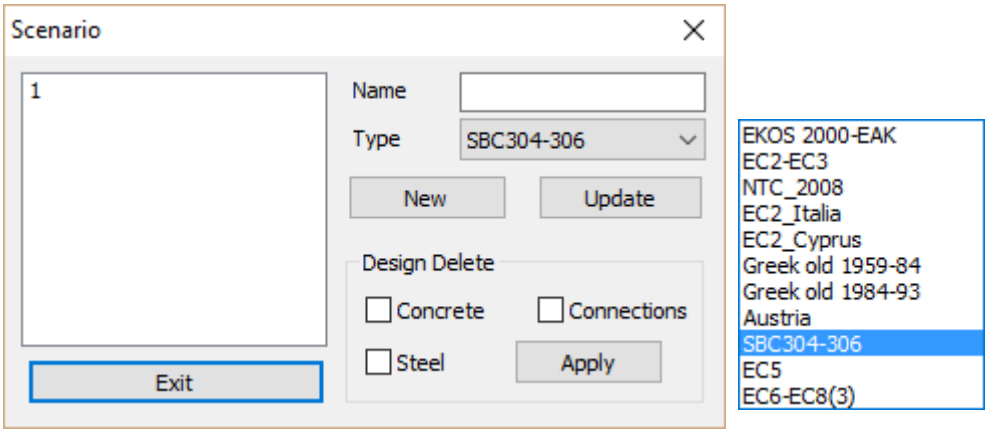
Furthermore, a new group of commands is added, concerning merging of elements (steel and timber)

1.1 New



This command is used in order to create a new scenario. Type a name, select the corresponding design regulation and then press the button 

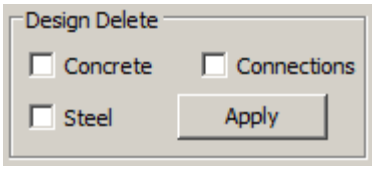
.



NOTES:

The selection of the design code corresponds to the design checks of the structural elements and the calculation of the steel reinforcement. Since you choose analysis' scenario and load combinations according to Eurocodes (see Ribbon "Analysis"), you must create the Eurocode scenarios for the design checks, too.

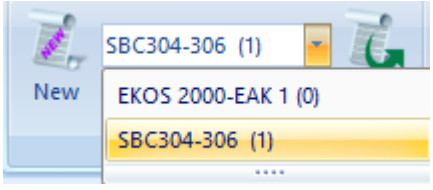
In order to modify an existing scenario press the button "Update"**/.**



In the field "Design Delete" activate the corresponding checkbox and then press "Apply", to delete the results of the previous design checks. Repeat this procedure using other combinations or parameters or scenarios, etc.

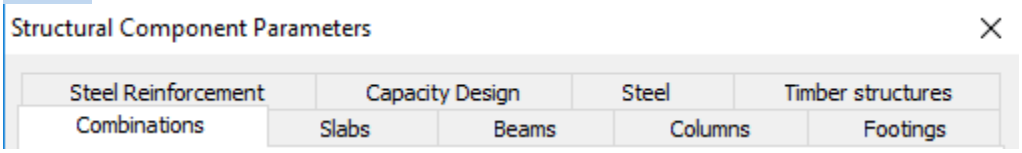
1.2 List

The drop-down list includes all created scenarios. Since you select one scenario, it becomes active. This means that the scenario will be used for the design checks.



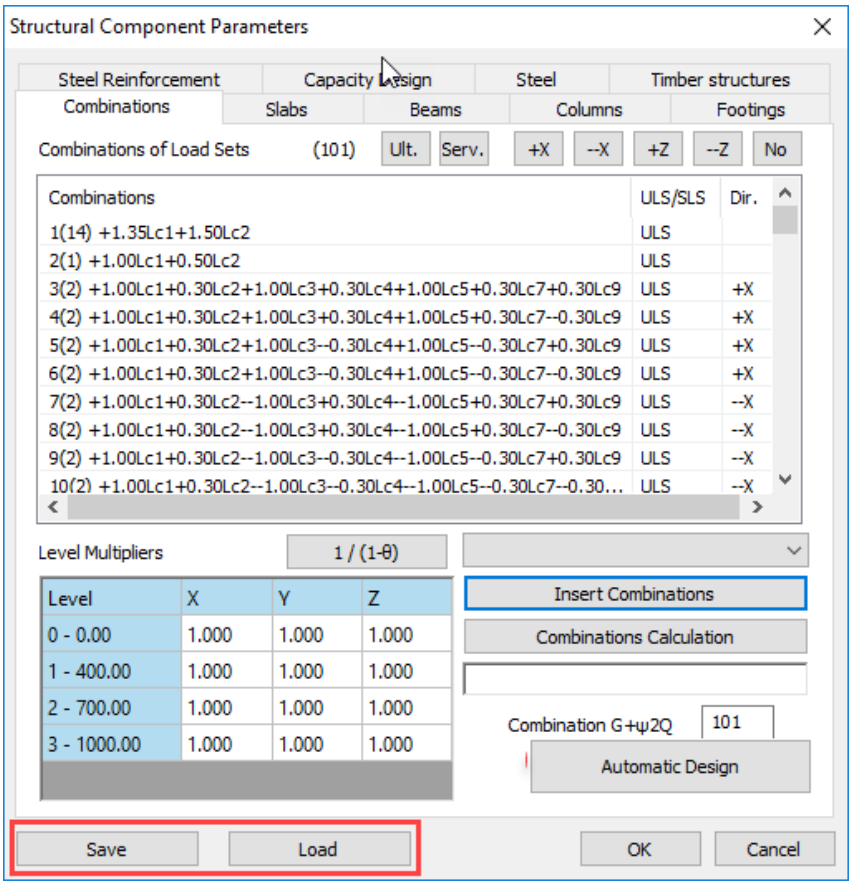
1.3 Parameters

This command is used for the definition of all design check parameters:

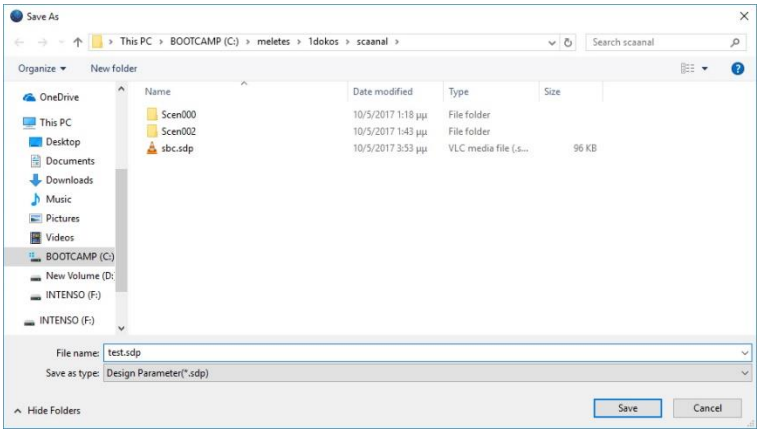


NOTE

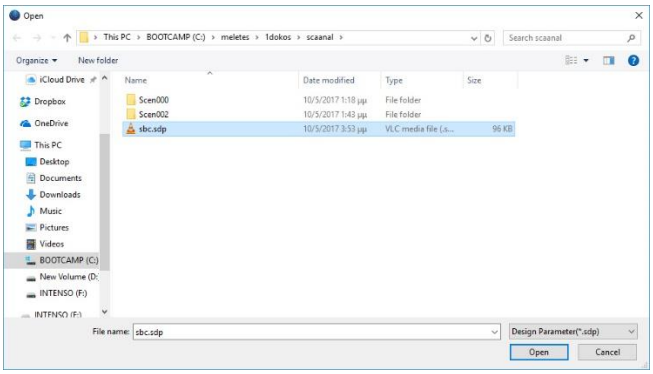
The parameters dialog box in the new SCADA Pro version contains two more commands for saving and reading the design parameters of the active scenario.



Once you configure the dimensioning parameters, you can now save them to a file in order to use them in your next projects.
Press “Save” and type a name

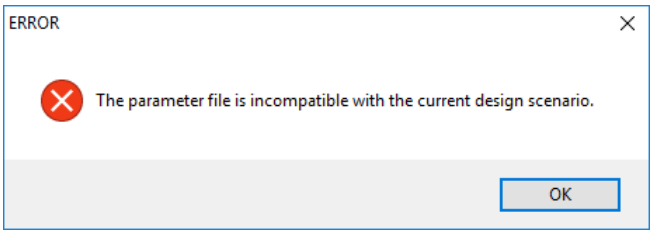


The file extension is sdp scenery design parameters.
Use “Load” command to apply the parameters that are already saved.



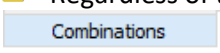
ATTENTION

A precondition for loading a parameter file is that the current design scenario is the same as the scenario of the parameters. Otherwise, you will see the following message:

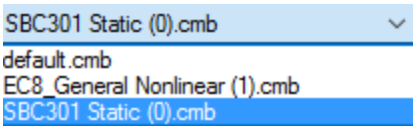


1.3.1 Combinations

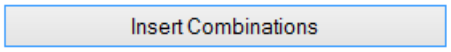
Regardless of the material, the calculation of combinations is a condition for designing.

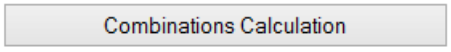


The selection of the existing .cmb combinations file is made:

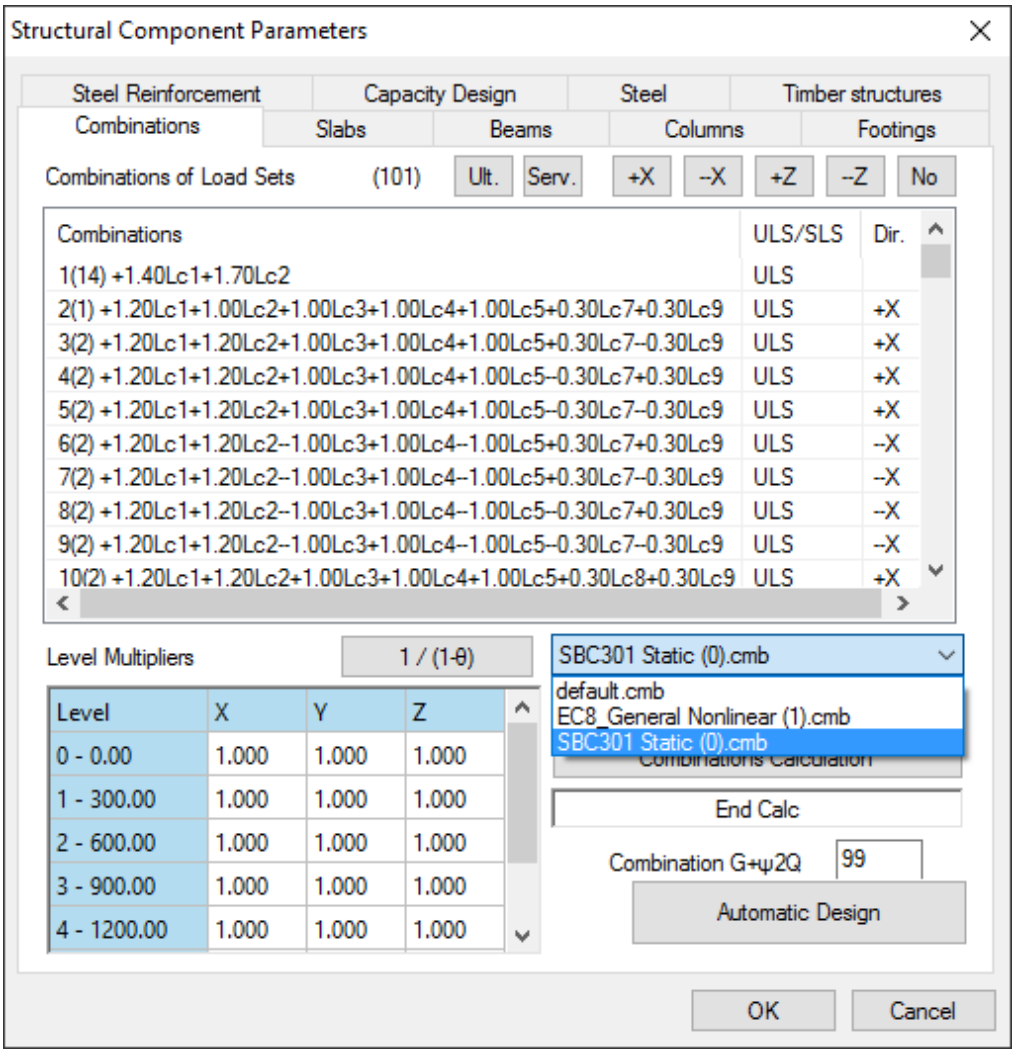
-from the dropdown list 

with automatic calculation or

-through the command 

that opens the folder with the registered .cmb files. Select the file and press 

.

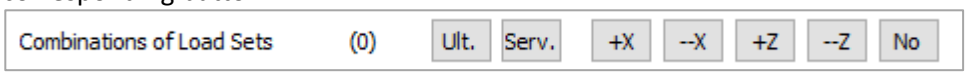


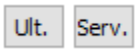
⚠ **NOTES:**

Depending on the case and the fulfilled conditions, you can use either the static or dynamic combination for design. You can also select combinations from different analysis scenarios to check the deviations, on the designing members, between them.

In "Combinations" tab the combinations list is displayed. The first number is the load combination's serial number.

The column "ULS/SLS" indicates the limit state of the combination and the column "Dir." indicates the direction of the participation for the specific capacity design combination.
By using the following bar, you can modify both the limit state and the direction by pressing the corresponding button.



In the column "ULS/SLS" that indicates the limit state of the combination, in case you want to change the status of the combination, first select it and then press the respective button 

.

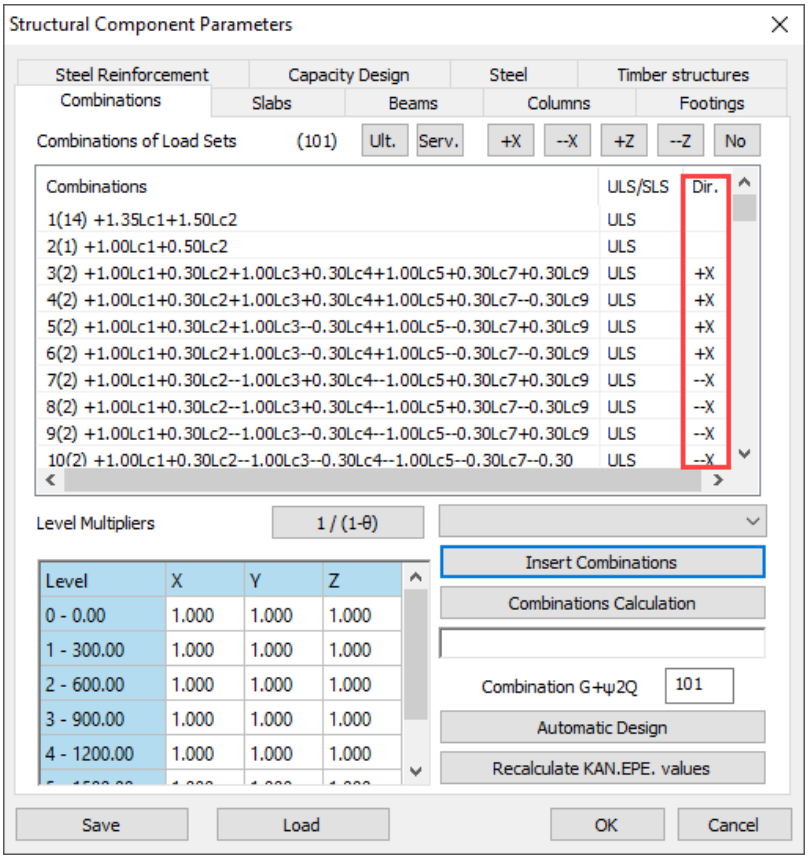
In the column "Dir." That indicates the direction of the participation for the specific capacity design combination, by selecting the corresponding button 

, you can define the direction of the participation. The label "No" means that the specific combination is excluded from the capacity design.

NOTES:

The regulation concerns the capacity design and its necessity for execution, as long as it is applied per earthquake direction and not per column direction.

Therefore, the exclusion of one direction from the capacity design check for one or more columns is implemented in SCADA, by setting a zero value to the incremental coefficient acd for the seismic combinations in which the seismic force, along with the particular direction, participates with a unit. The characterization of the combinations, which appears in members' design, has that meaning too.



A combination is defined per x or z if the corresponding seismic force has a unit coefficient.

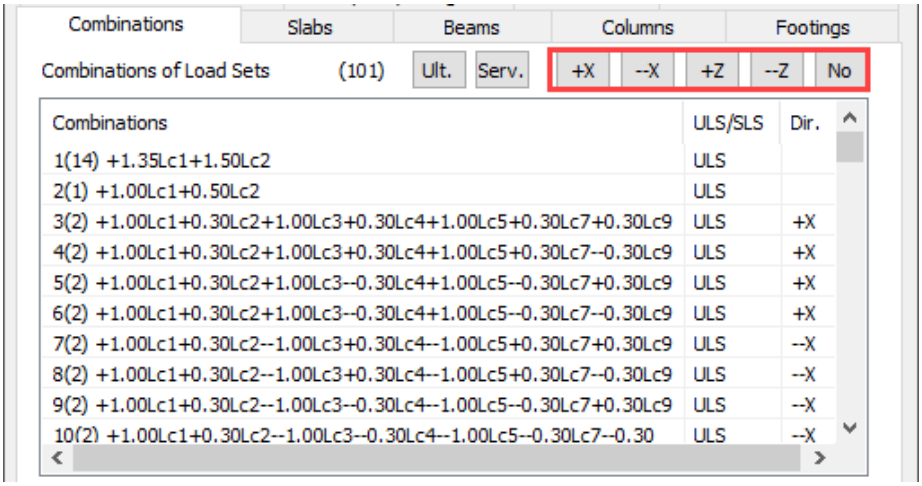
In conclusion, we would assume that in case we want to exclude one direction from one column in order to avoid the capacity design check, we move to the definition and choose the direction of the local axis that is parallel to the direction of the earthquake we want to exclude.
In case that either the column or the fictitious axes are twisted, we choose the local axis with the smallest angle from the corresponding seismic axis we want to exclude. In this way, the program will calculate the acd just for the particular seismic direction (apparently also for the two local axes of the column), while it will not calculate the acd for the seismic combinations of the other directions.

Indicatevely, in the following printout:

Node = 15
Col. bottom = 14

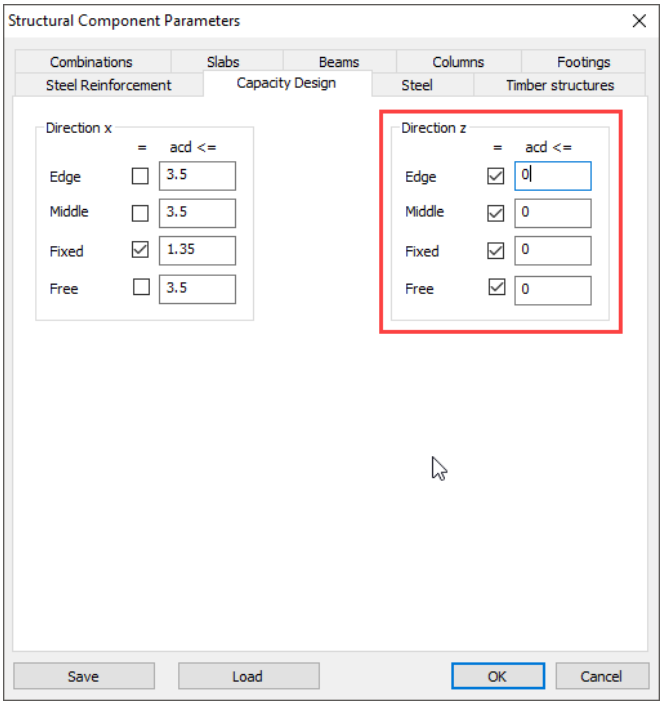
COMB.	SMRby	SMEby	acdy calc	acdy	SMRbz	SMEbz	acdz calc	acdz
3	134.000	15.876	10.973	4.000	134.000	2.907	59.929	4.000
4	134.000	15.876	10.973	4.000	134.000	2.907	59.929	4.000
5	134.000	15.569	11.189	4.000	144.800	4.605	40.880	4.000
6	134.000	15.569	11.189	4.000	144.800	4.605	40.880	4.000
7	144.800	15.569	12.091	4.000	134.000	4.605	37.831	4.000
8	144.800	15.569	12.091	4.000	134.000	4.605	37.831	4.000
9	144.800	15.876	11.857	4.000	144.800	2.907	64.759	4.000
10	144.800	15.876	11.857	4.000	144.800	2.907	64.759	4.000
11	134.000	15.569	11.189	4.000	134.000	3.416	50.993	4.000
12	134.000	15.569	11.189	4.000	134.000	3.416	50.993	4.000
13	134.000	15.876	10.973	4.000	144.800	5.114	36.808	4.000
14	134.000	15.876	10.973	4.000	144.800	5.114	36.808	4.000
15	144.800	15.876	11.857	4.000	134.000	5.114	34.063	4.000
16	144.800	15.876	11.857	4.000	134.000	5.114	34.063	4.000
17	144.800	15.569	12.091	4.000	144.800	3.416	55.103	4.000
18	144.800	15.569	12.091	4.000	144.800	3.416	55.103	4.000
19	134.000	14.853	11.728	4.000	134.000	4.605	37.831	4.000
20	134.000	14.853	11.728	4.000	134.000	4.605	37.831	4.000
21	134.000	14.547	11.975	4.000	144.800	2.907	64.759	4.000
22	134.000	14.547	11.975	4.000	144.800	2.907	64.759	4.000
23	144.800	14.547	12.941	4.000	134.000	2.907	59.929	4.000
24	144.800	14.547	12.941	4.000	134.000	2.907	59.929	4.000
25	144.800	14.853	12.673	4.000	144.800	4.605	40.880	4.000
26	144.800	14.853	12.673	4.000	144.800	4.605	40.880	4.000
27	134.000	14.547	11.975	4.000	134.000	5.114	34.063	4.000
28	134.000	14.547	11.975	4.000	134.000	5.114	34.063	4.000
29	134.000	14.853	11.728	4.000	144.800	3.416	55.103	4.000
30	134.000	14.853	11.728	4.000	144.800	3.416	55.103	4.000
31	144.800	14.853	12.673	4.000	134.000	3.416	50.993	4.000
32	144.800	14.853	12.673	4.000	134.000	3.416	50.993	4.000
33	144.800	14.547	12.941	4.000	144.800	5.114	36.808	4.000
34	144.800	14.547	12.941	4.000	144.800	5.114	36.808	4.000
35	134.000	5.228	33.320	0.000	134.000	12.264	14.204	0.000
36	134.000	5.228	33.320	0.000	134.000	12.264	14.204	0.000
37	144.800	4.205	44.761	0.000	134.000	12.774	13.637	0.000
38	144.800	4.205	44.761	0.000	134.000	12.774	13.637	0.000
39	134.000	4.205	41.422	0.000	144.800	12.774	14.736	0.000
40	134.000	4.205	41.422	0.000	144.800	12.774	14.736	0.000
41	144.800	5.228	36.006	0.000	144.800	12.264	15.349	0.000
42	144.800	5.228	36.006	0.000	144.800	12.264	15.349	0.000
43	134.000	4.921	35.397	0.000	134.000	12.774	13.637	0.000
44	134.000	4.921	35.397	0.000	134.000	12.774	13.637	0.000
45	144.800	3.899	48.283	0.000	134.000	12.264	14.204	0.000
46	144.800	3.899	48.283	0.000	134.000	12.264	14.204	0.000
47	134.000	3.899	44.682	0.000	144.800	12.264	15.349	0.000
48	134.000	3.899	44.682	0.000	144.800	12.264	15.349	0.000
49	144.800	4.921	38.250	0.000	144.800	12.774	14.736	0.000

You can see that the acd have been calculated for combinations till the 34th one (combinations +x and –x), while post the 35th one the acd have not been calculated (combinations +z and –z)) Another way in order not to do the capacity design check in one direction is to modify the combinations' definition in members' design through the above tools.



You can also select one or more combinations, depending on the seismic direction and to characterize it by "No". In this way, they are not going to be taken into account in the capacity design check.

Finally, the last way in order not to do the capacity design check in one or more directions is to set the acd=0 limit in the field Structural Component Parameters.
The same result will be accomplished.



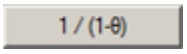
Level	X	Y	Z
0 - 0.00	1.000	1.000	1.000
1 - 400.00	1.000	1.000	1.000
2 - 700.00	1.000	1.000	1.000
3 - 1000.00	1.000	1.000	1.000
4 - 1300.00	1.000	1.000	1.000
5 - 1600.00	1.000	1.000	1.000

SECTION 2.3
COMBINING FACTORED LOADS USING STRENGTH DESIGN
2.3.2. Basic Combinations. *Structures, components, and foundations shall be designed so that their design strength equals or exceeds the effects of the factored loads in the following combinations:*
U = 1.4 (D + F) (9-1)

U = 1.4 (D + F + T) + 1.7(L + H) + 0.5 (Lr or R) (9-2)
U = 1.2D + 1.6 (Lr or R) + (1.0 L or 0.8 W) (9-3)
U = 1.2D + 1.6W + 1.0L + 0.5(Lr or R) (9-4)
U = 1.2D + 1.0E + 1.0L (9-5)
U = 0.9D + 1.6W + 1.6H (9-6)
U = 0.9D + 1.0E + 1.6H (9-7)

In the preceding expressions, the following values are used:
U = the design or ultimate load the structure needs to be able to resist
D = dead load
L = live load
Lr = roof live load
S = snow load
R = rain load
W = wind load
E = seismic or earthquake load effects

"Level Multipliers": In this field, you can increase or decrease the seismic actions in any direction and level, by typing different factors.

Press the button 

in order to take into account the P-Delta effect during the design check. The stress resultants will be increased automatically at the corresponding levels, where 0.1 < θ <θmax.

10.9.7.2
When the stability coefficient (ϑ) is greater than 0.10 but less than or equal to ϑ max the incremental factor related to P-delta effects (ad) shall be determined by rational analysis. To obtain the story drift for including the P-delta effect, the design story drift determined in Section 10.9.7.1 shall be multiplied by 1.0/(1 -ϑ).

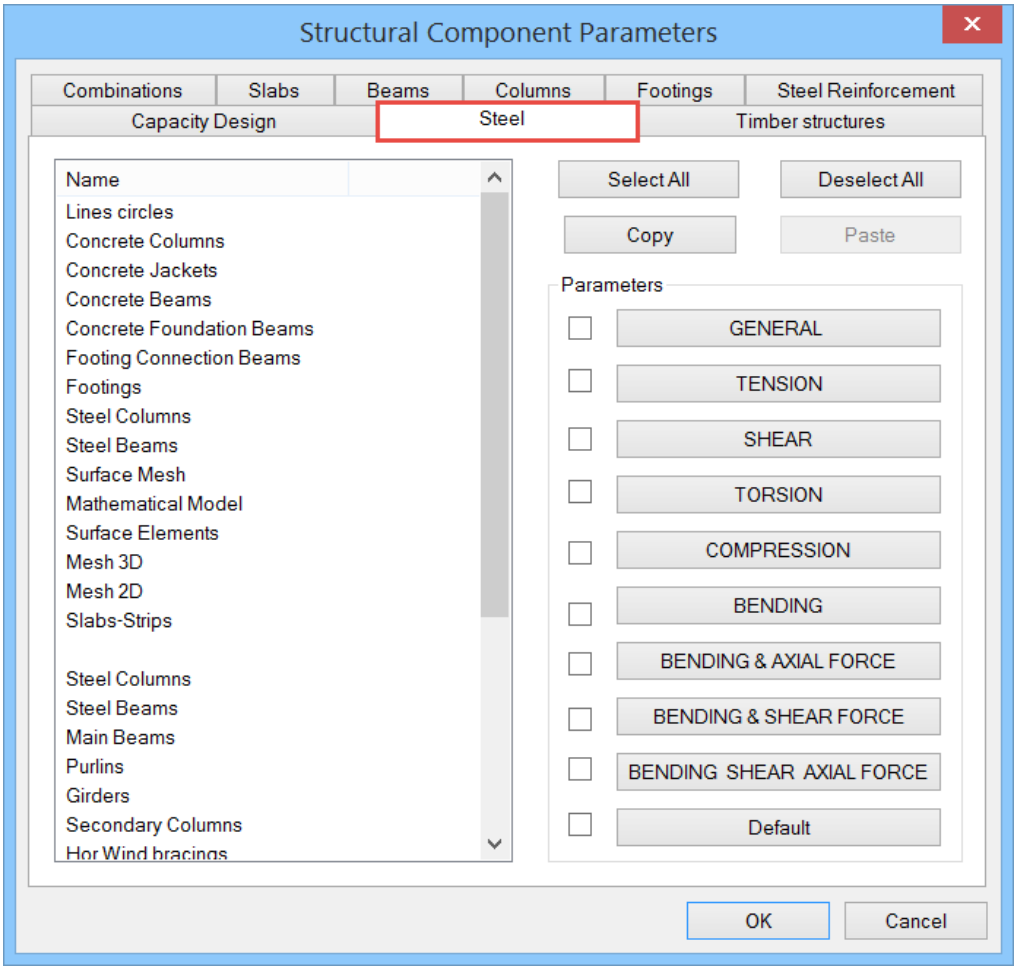
ATTENTION:

For modification purposes, press the following button 

.

The following field/concerns only the Greek EKOS.

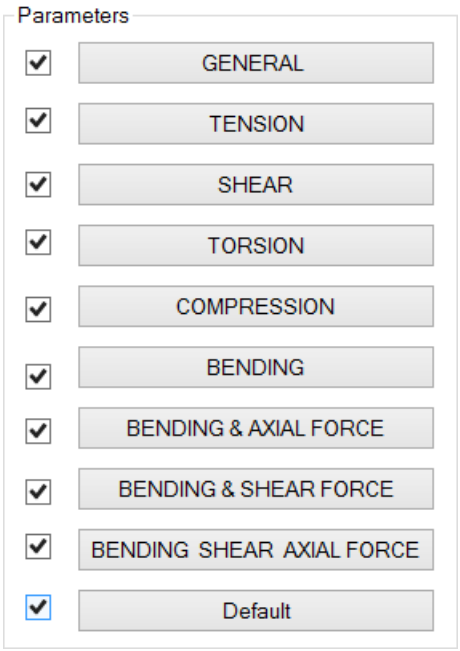
1.3.2 Steel cross sections



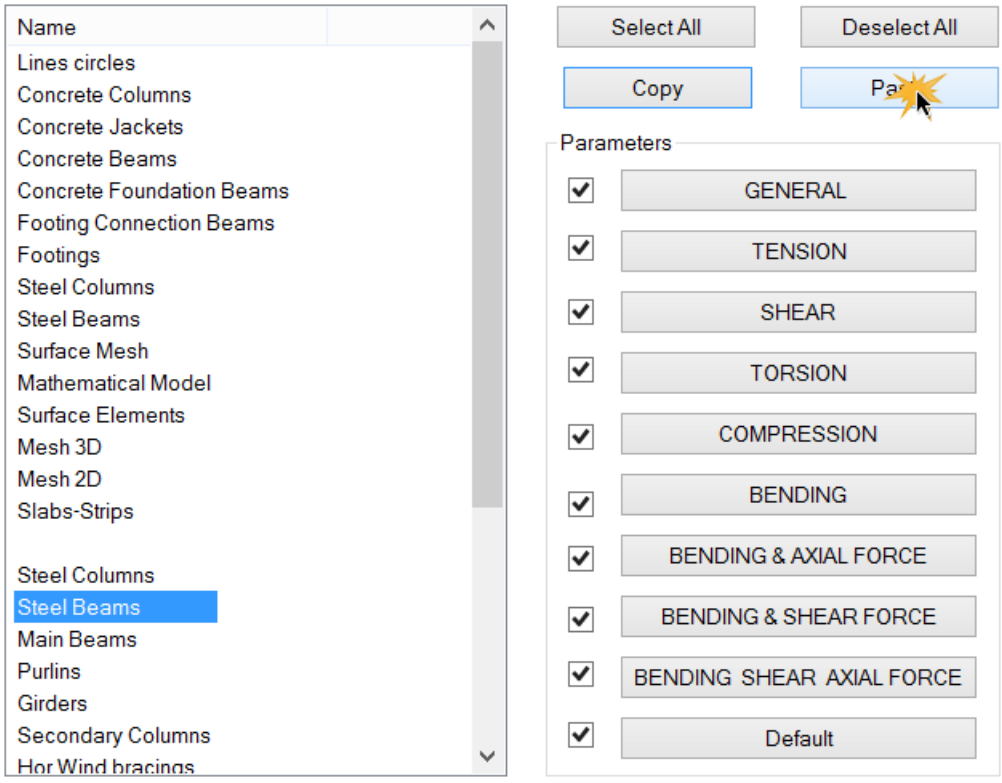
First, select a layer. Click one from the list, or more using "ctrl", or all using "Select All".
(By pressing the button "Deselect All" cancel the previous layers' selection.)
Then activate one or more design checks by clicking on the corresponding checkbox and press the corresponding button to specify 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".

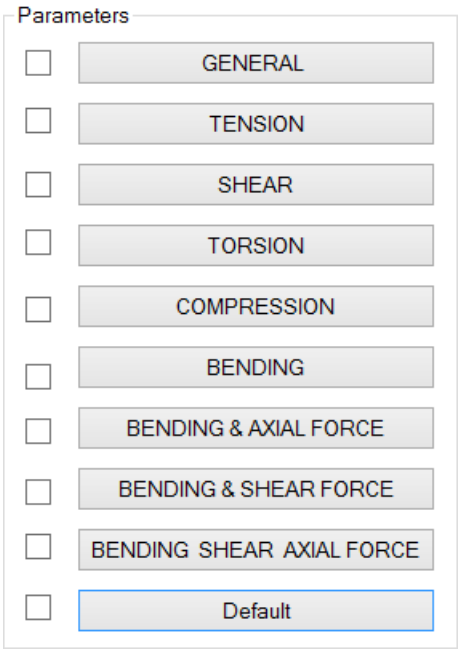
EXAMPLE:
Suppose you have set all parameters for the layer Steel Columns and you want to pass these parameters to Steel Beams. Activate the check box next to "Default" and all parameters are selected automatically.

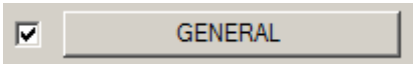


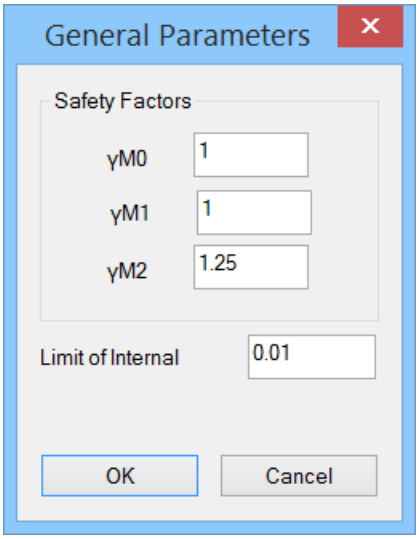
Then press "Copy", select layer Steel Beams and press "Paste" (now activated).



Now all the parameters defined for Steel Columns are defined also for the layer Steel Beams.
An alternative method in order to set the same parameters to all layer including steel sections is to select all layers by pressing "Select all" button and set the parameters once for each check category.
Note that at least one (or more) layers should be selected in order to set the parameters.



Press the button "GENERAL"  to set the γ_{Mi} safety factors:



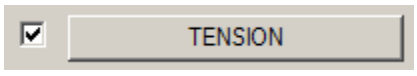
γ_{M0} : partial factor for cross-sections' resistance whatever the class is

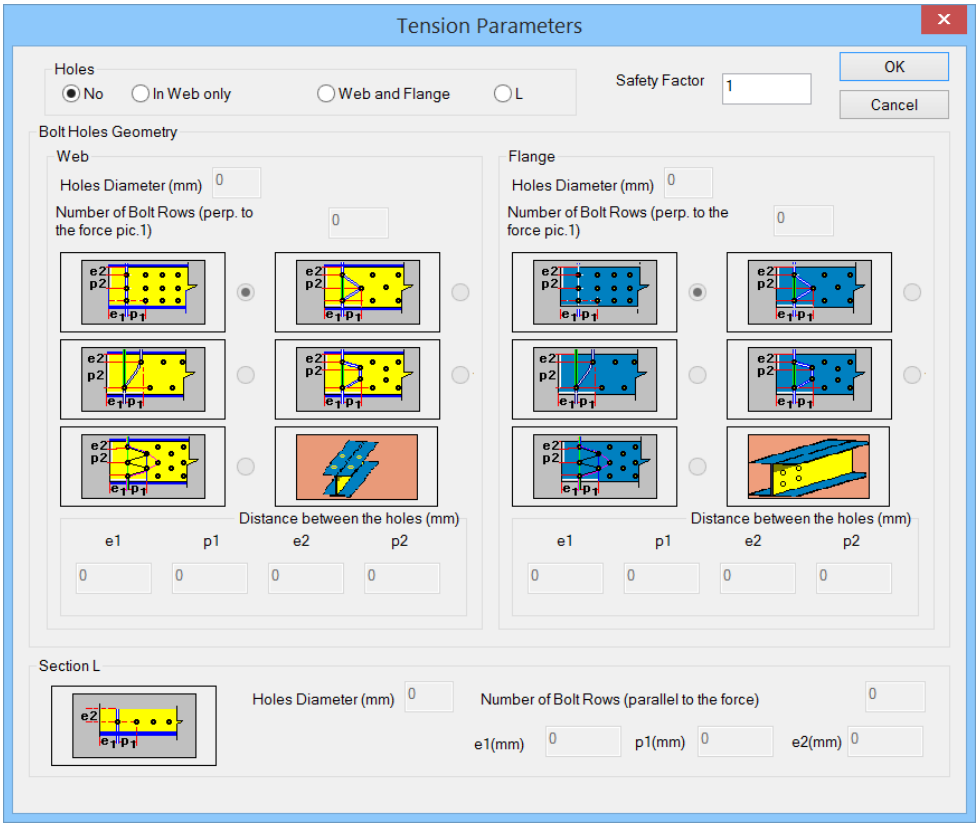
γ_{M1}: partial factor for members' resistance to buckling based on tests

γ_{M2}: partial factor for resistance of cross-sections in tension to fracture

In the "Limit of Internal" field define an upper limit. Below this value, the program will not consider the corresponding stress resultants.

These values are recommended by Eurocode.

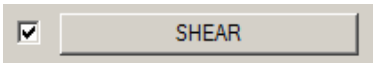
Press the button "TENSION"  to define the parameters that correspond to the shear design check as well as the position of the hole check (EC3 §1.8 §3.5):



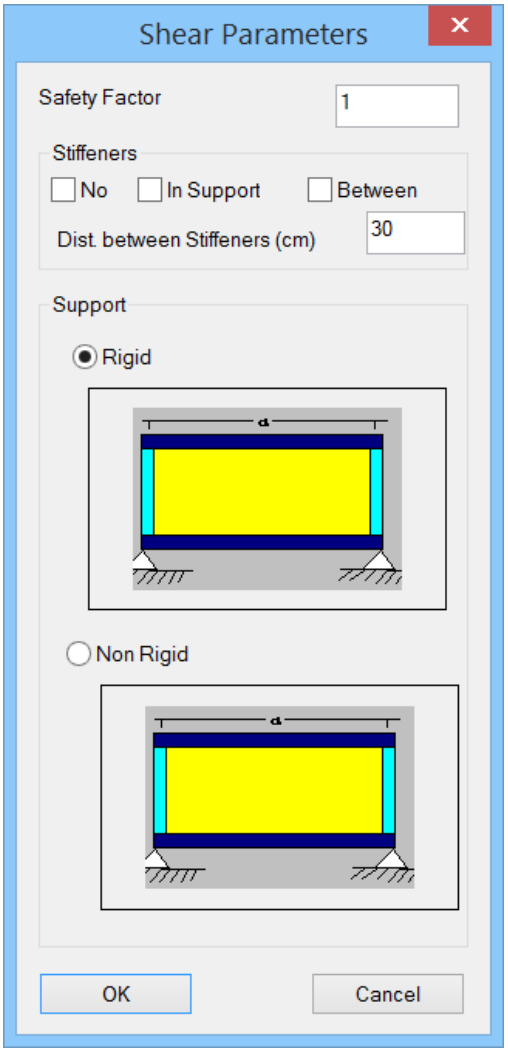
Specify the spacing of the centers of two consecutive holes, the holes diameter and the number of bolt rows.
In case of L section specify the parameters on the bottom of the dialog box in the field "Section L".

Here the user defines whether to consider the reduction of the tensile strength of the section due to the bolt rows 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.

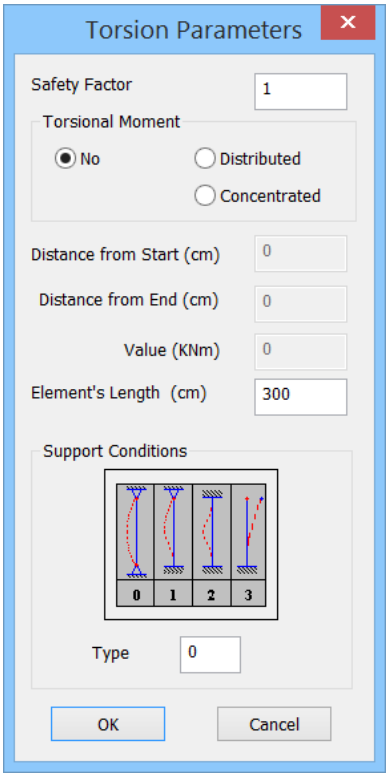
The safety factor for all design checks is fixed and equal to one, which means that the program calculates the ratio of the stress resultant versus the resistance. A value of the calculated ratio greater than 1.0 indicates failure.

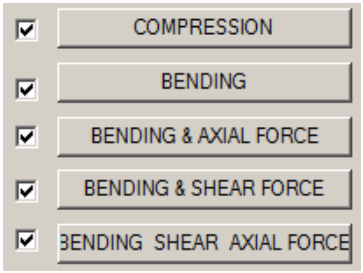
Press the button "SHEAR" 

in order to define if the elements of the selected layer contain stiffeners and if so which type; web stiffeners or intermediate stiffeners. Also define the spacing between the stiffeners and the type of the connection (rigid or not rigid).

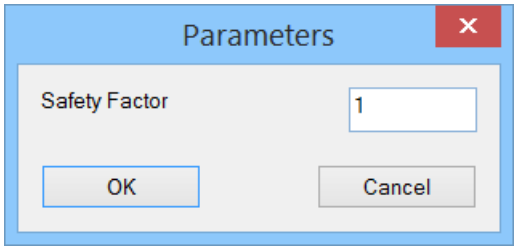


Click the button “TORSION”  to define whether the structural elements of the selected layer are loaded by a distributed or concentrated torsional moment, or not. In the following dialog box, you may as well define the support conditions based on the corresponding figures. Select the type of moment and set (i) the relative distances from the start and the end, (ii) the value of the moment and (iii) the length of the element in the corresponding fields. Also, set the support condition by typing in the "Type" field the values 0, 1, 2 or 3.





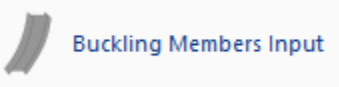
For all design checks presented in the figure on the left, define the safety factor in the dialog box that appears when you click one of the five buttons. The safety factor is the ratio of the resistance value versus the corresponding design value, which is set 1.0 by default.

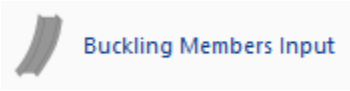


1.4 Merge Elements

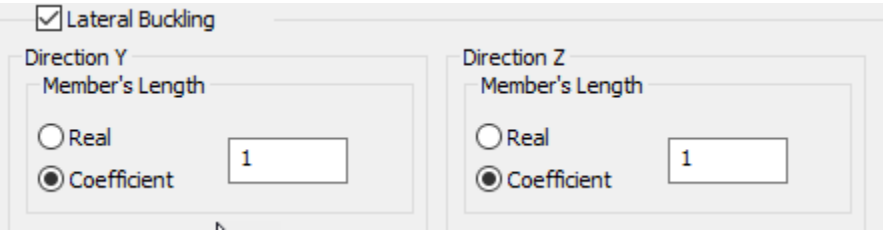
In the new version of the program, a new command group is added, which concerns merging of steel (and timber) members for the calculation as well as buckling and deformation checks display according to EC3.

IMPORTANT NOTES:

- By using this command, it 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 by defining the length coefficients (see 

)



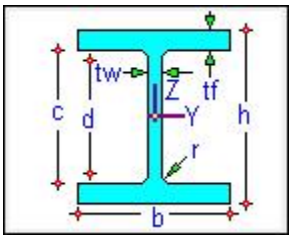
- Now, be using merging per direction, there is no need for the coefficient process, and merging will be achieved, in most cases, automatically.
- Also, note that through the merging process, the buckling length, is calculated correctly, and in the printouts of the results a merged element is printed once with the annotation of the individual members that contains.

BASIC CONCEPTS OF BUCKLING ALONG MAJOR AND MINOR AXIES.
WHAT IS LY AND LZ RESPECTIVELY.

Generally, in the double T cross sections, the local axis

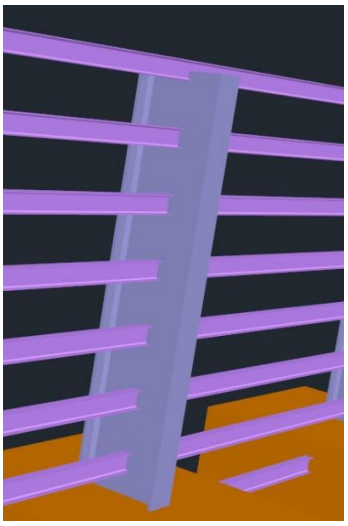
- **y-y** is the **major**, and
- **z-z** is the **minor**,

as in the figure below:

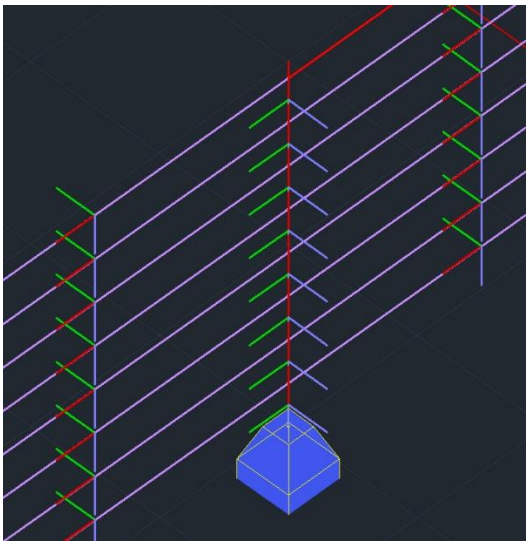


EXAMPLE:

For example, let's see the buckling length of this column below, which is connected laterally with griders. First, check the initial buckling lengths Ly and Lz for the column.



The local axes direction of the column and the griders are as shown in the figure below:



The **columns** buckling along its **major** axis **y-y** (green) means:

- Buckling because of **My** (rotation around the y-y axis), that is, buckling out of the plane, which in the specific case, the merged length should be the buckling lengh, that is, the total length of the column.

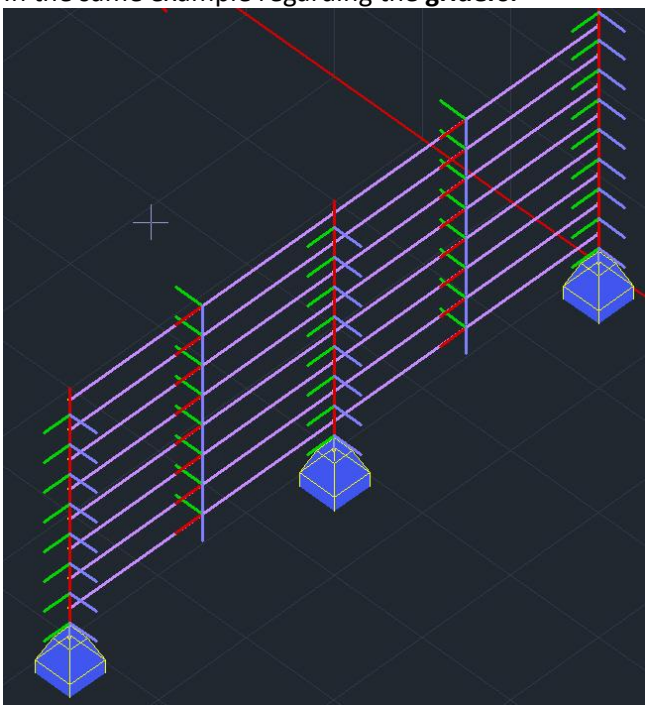
In the other direction, buckling along its **minor** axis **z-z** (blu) means:

- Buckling because of **Mz** (rotation around the z-z axis), that is buckling in the plane. The column is considered to be supported laterally by the griders, so, the buckling length Lz should be the length of each member.

NOTE:

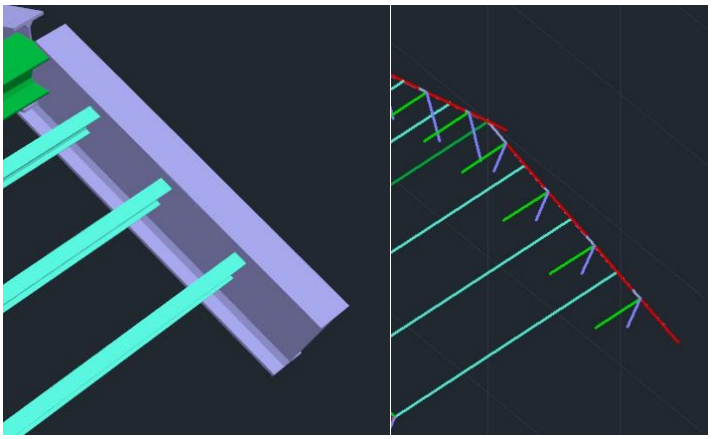
Generally, making a rule, we could say that, we consider the **merged length Ly** in the direction where the local axis y-y is parallel to the supporting elements. While in the other direction, if there are no supporting elements, **Lz** is **the length of each member**.

In the same example regarding the **griders:**



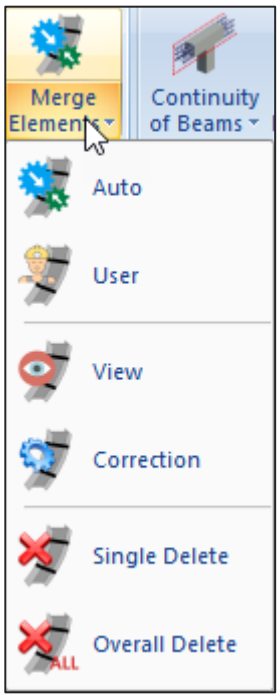
The supports from the columns are parallel to the local z-z axis (blue, out of plane) of the griders. So, merging will be in Lz (total length). While in the y-y direction (green, in the plate), Ly is the length of each member.

Respectively, for the **inclined beam** of the figure below:



The local axis of the beam that is parallel to the purlins is the y-y. So, Ly wil be the merged length of the total beam, while Lz will be the single members.

Merge group command, contains the list of commands below:



Merge elements mean that the individual parts of a single element, merge in each buckling direction, either automatically or manually.

Meaning that the buckling length is considered computationally to be not the actual length of the element, but the unified from the beginning to the end of the column or beam, respectively.

In addition, in the presentation of the results, for these merged elements, the most unfavorable results are displayed once and not for each individual one, as it was happening so far.

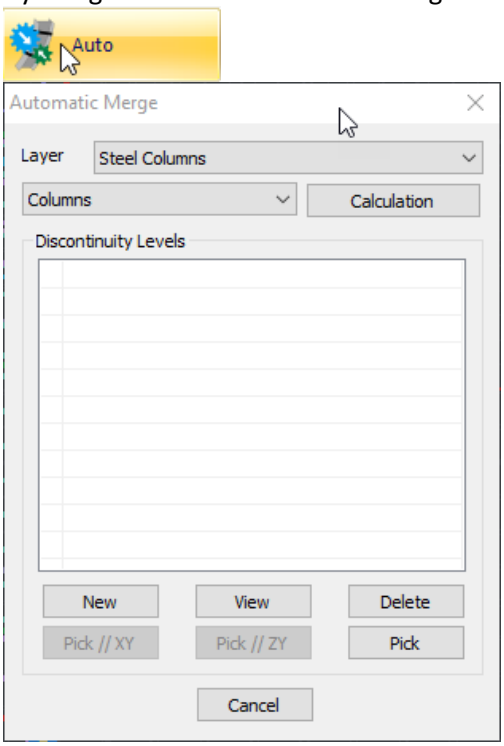
Finally, in automatic merging, there is the definition of discontinuity levels, horizontal or vertical, used as merging boundaries of a continuous element.

NOTE

It is better to work in the 3D mathematical model, displaying the local axes, whenever you use these commands.

1.4.1 Auto merge

By using this command the following dialog box is displayed:



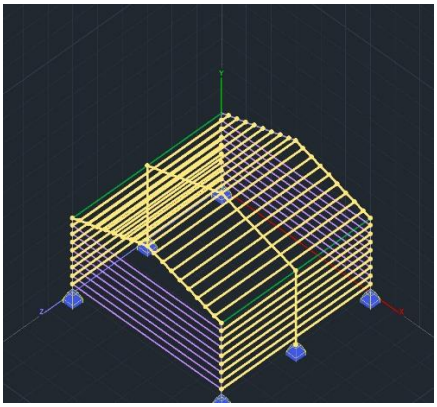
First, choose the layer of the elements to merge.

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 for all the others.

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

Merged elements are displayed with colors:

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



The next section is about defining and processing the **discontinuity levels**.
Discontinuity levels are levels that are the boundaries of beams and columns, used to break merging in each direction.

- For the columns, the discontinuity levels are horizontal levels defined by the floor levels.
- For the beams, the discontinuity levels are always vertical levels defined by two points.

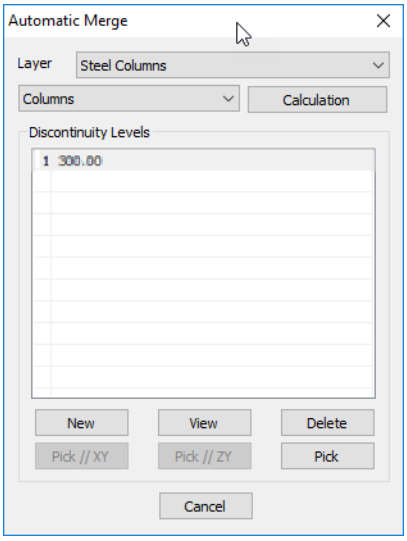
Predefined limits:

- For the horizontal levels, they are the foundation level and the last level.
- For the beams, they are the vertical limits of the model.

⚠ The predefined limits are never displayed in the **discontinuity levels** list.

EXAMPLE:

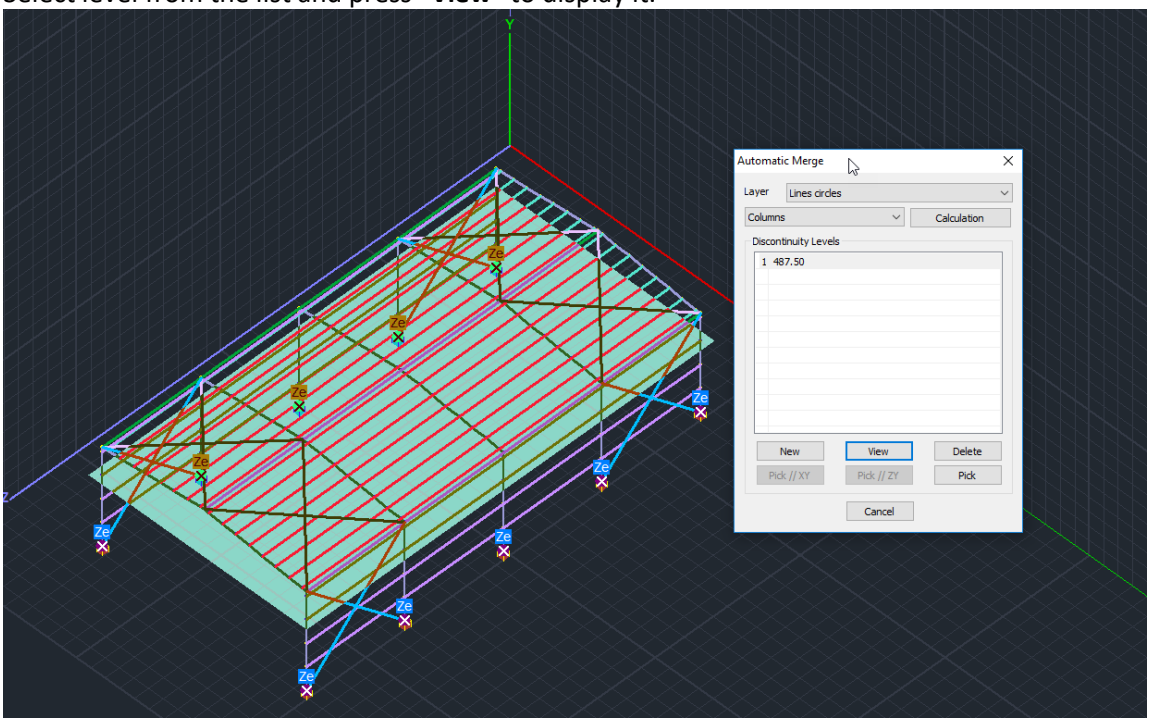
A three floor building with 0.00, 300.00 and 600.00 height levels, in **discontinuity levels** list of the columns, only the level 300.00 will be specified by default (that is, only the intermediate level without the limits)



considering that, the columns merging will be interrupted at 300.00 cm. The column will merge from 0.00 to 300.00 cm and the next floor column from 300.00 to 600.00 cm.

- To set your own **discontinuity levels** for **COLUMNS**:

press "**NEW**" and next "**Pick**" and point one point.
The horizontal level that defines the altitude of this point is a **discontinuity level**.
Select level from the list and press "**View**" to display it.

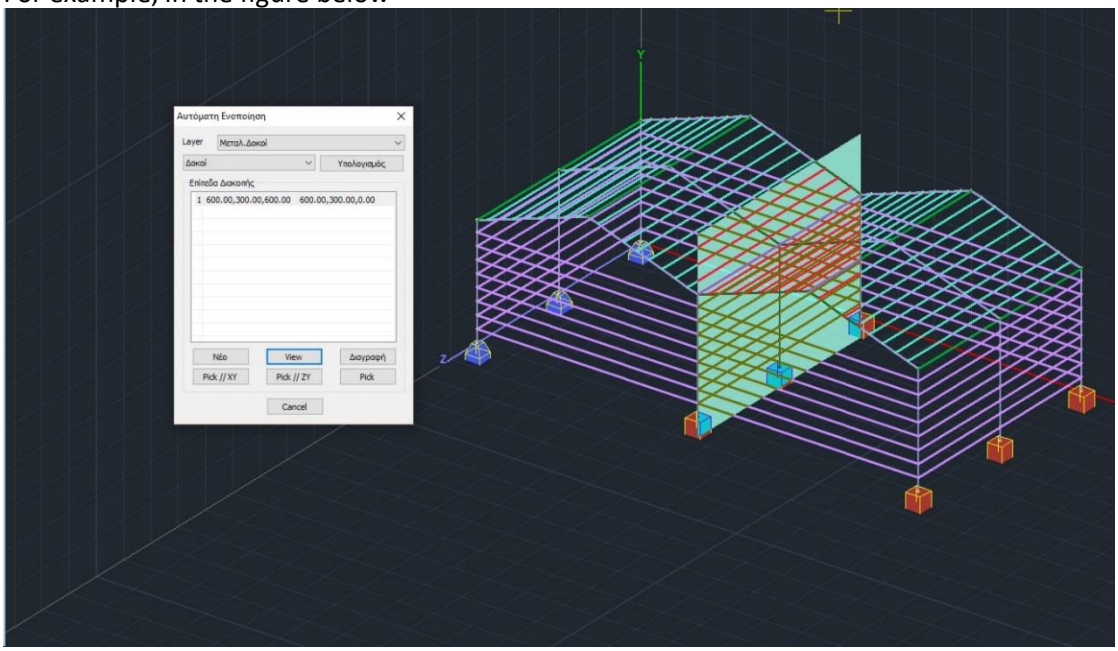


To delete a **discontinuity level,** select it from the list and press "**Delete**".

- For the **BEAMS**:

The definition of vertical **discontinuity levels**, but now through "**Pick**" you define two points, that is, a line that defines a vertical **discontinuity level**.

For example, in the figure below



discontinuity level of the front and back griders is the limit of the two buildings.

⚠ Especially for the beams, and when the **discontinuity level** you want to set, is parallel to the Global XY or ZY, press the corresponding command and point only one point.

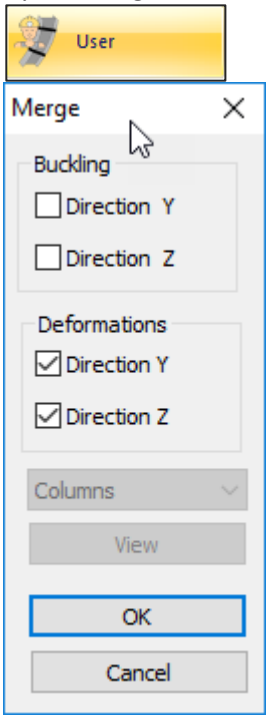
For Beam's and Column's **discontinuity levels,** *editing* can be achieved in two ways:

1. Either by deleting and defining a new one,
2. or by selecting the corresponding level and re-defining by pointing a point or points.

1.4.2 Users merging

Select the command, and then point the start point and the endpoint of the members to merge.

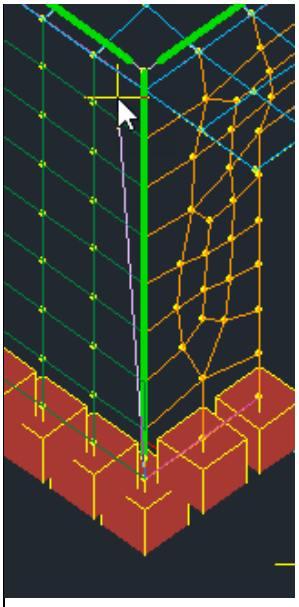
By selecting the second point (endpoint), the following dialog box appears:



where you define the direction of merging for Buckling and Deformations.

1.4.2.1 Merge concrete columns

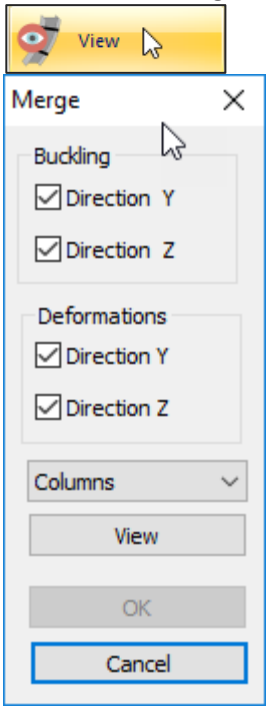
This command is mainly used in masonry buildings with vertical reinforced concrete elements which connect the nodes of the surface elements and which, in order to be designed, must be merged.



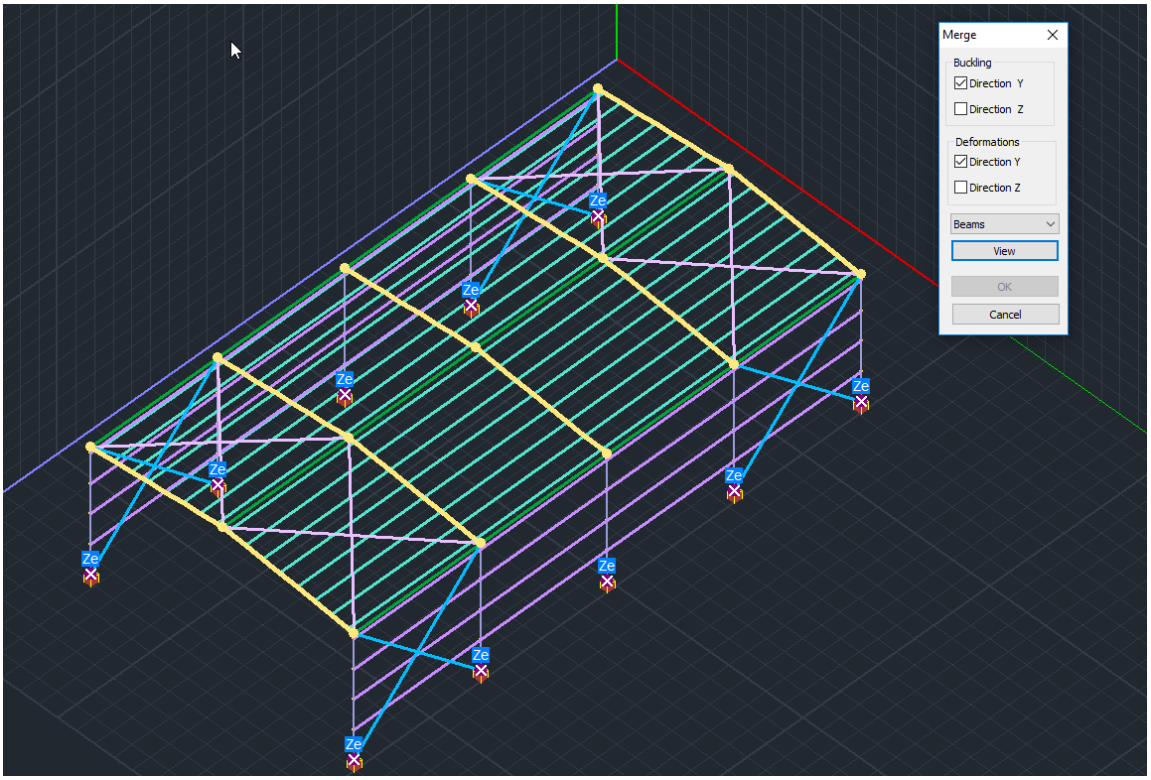
You select the command, and then you point the starting point and the end point of the members you want to merge.

1.4.3 View

Using View command, you can see the merged elements colored, accordi tong merge direction. Also, the following dialog box appears:



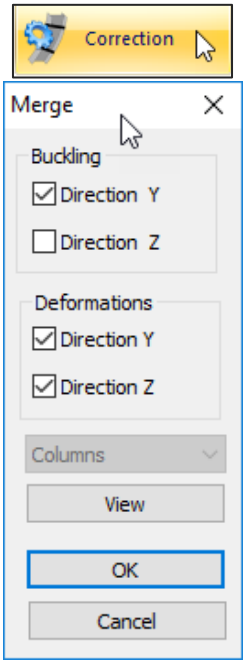
- Select element's type from the list, then
- check in Buckling / Deformations the direction of merging in order to see the corresponding merged elements.



1.4.4 Correction

“Correction” command offers the opportunity to correct elements which are already merged.

Select the command and then a merged element to display the following dialog box:



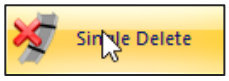
in which checks shows the merging direction.

Here you can modify the selections of the directions in Buckling and Deformations. Press View to see the member with the corresponding merged color.

ATTENTION

This command works only for the merged element, otherwise, the dialog box does not appear.

1.4.5 Single Delete

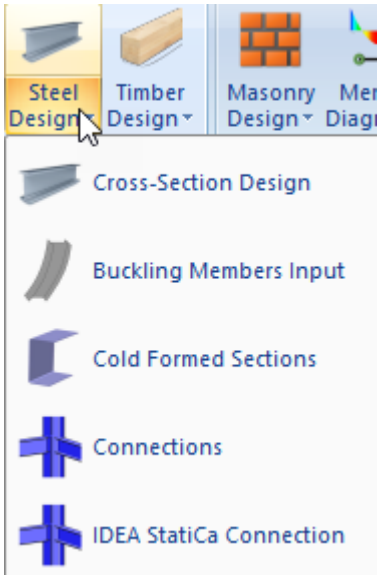


Select this command in order to delete a single merging.
Select the command and click a merged element. Right click to delete merging.

1.4.6 Overall Delete

Select this command to delete all merging. Select to delete all merging from all the merged elements.

2. Steel



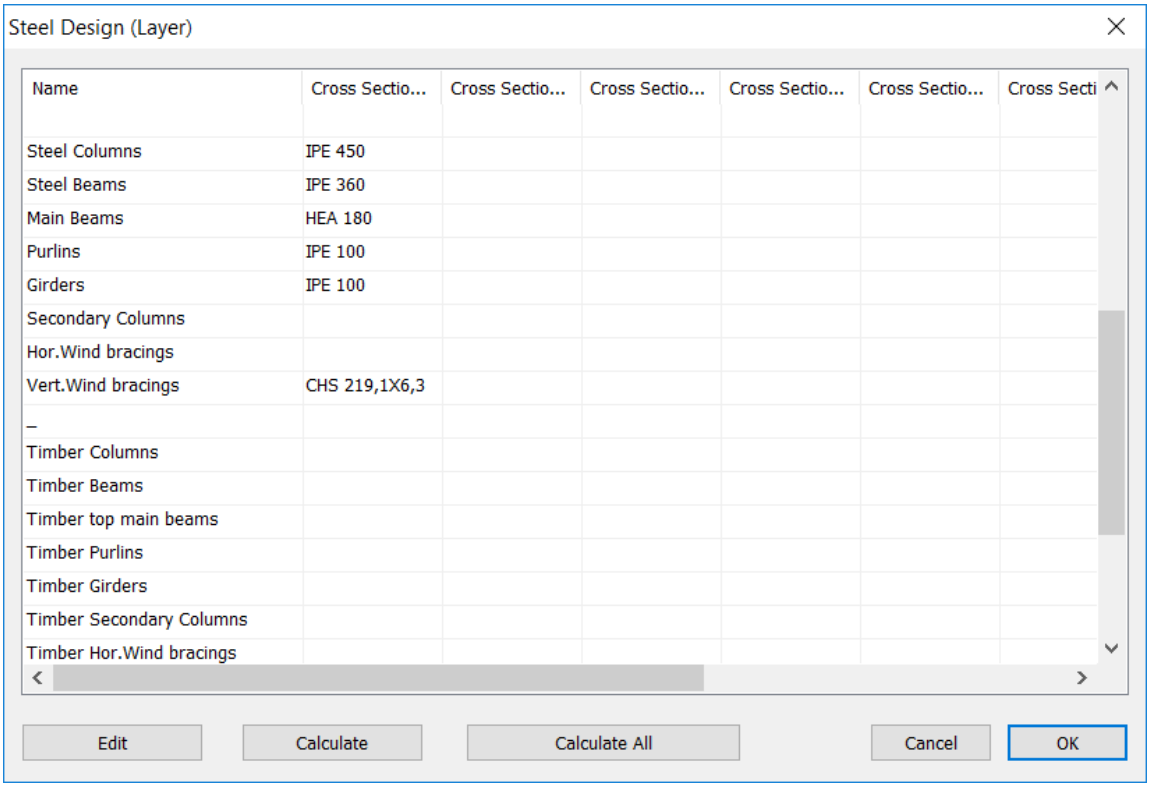
"Steel Design" command group contains commands for the cross-sections design, the buckling resistance, and the steel connections design.

⚠ Always remember to calculate the corresponding load combinations in the parameters dialog box.

2.1.1 Cross Section Design

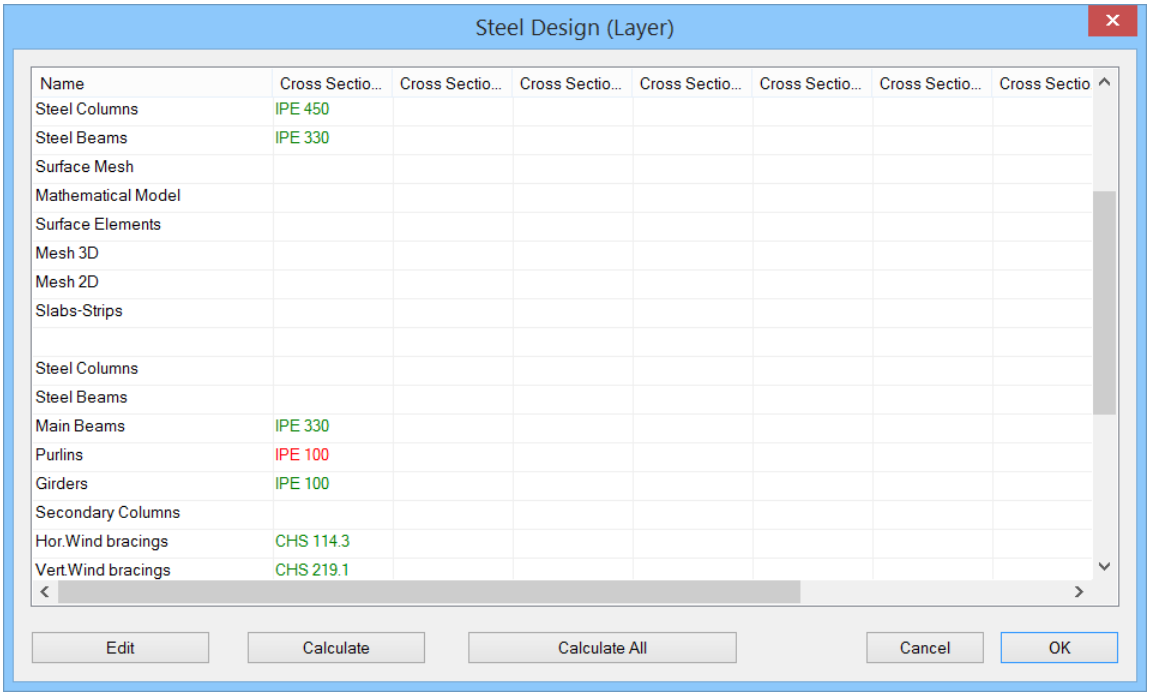
This command is used to check the adequacy of the steel cross-section.

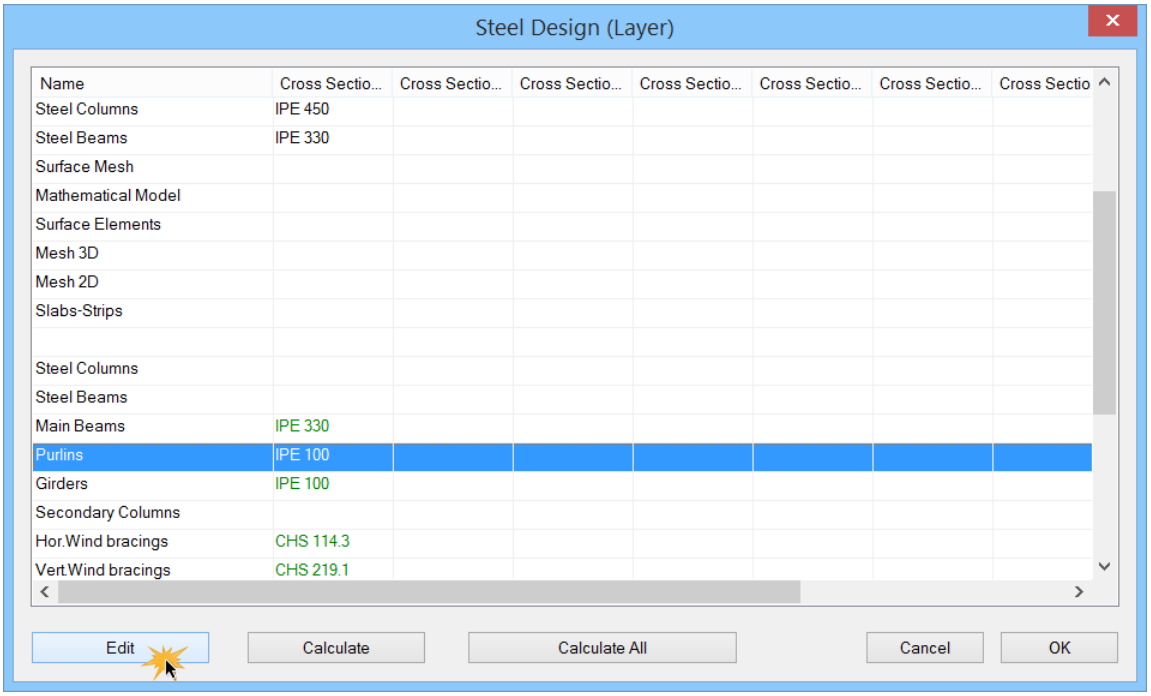
Select this command to open the following dialog box:



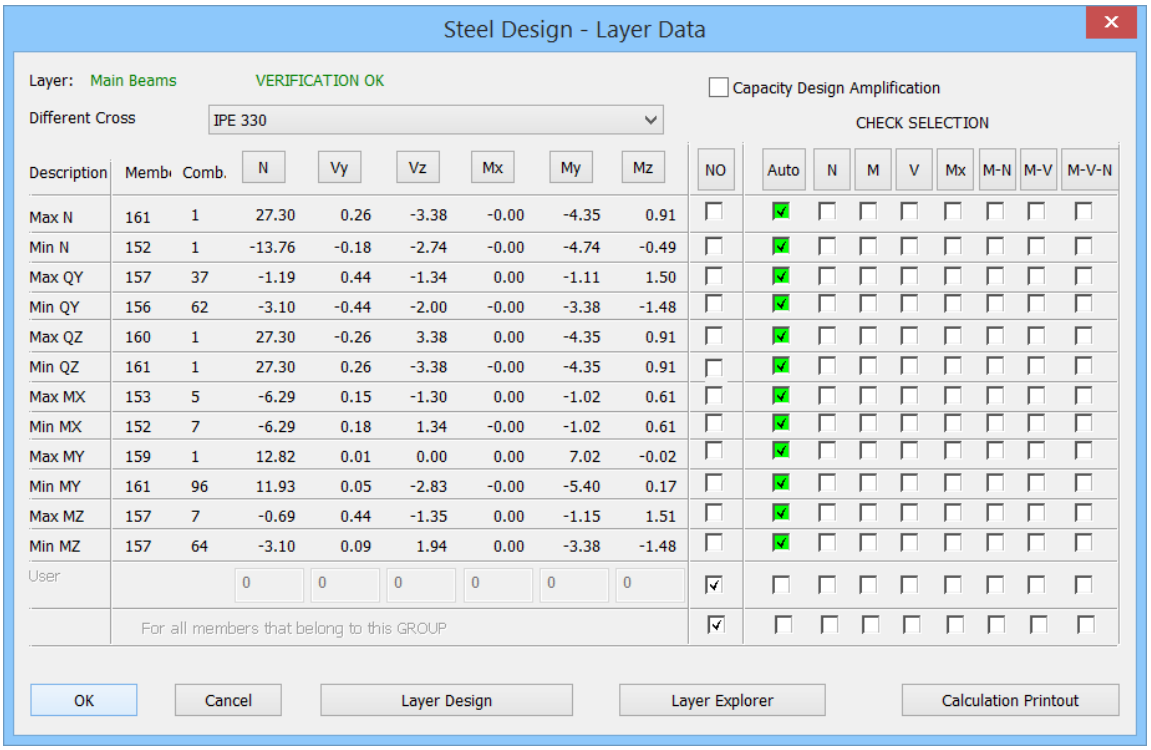
The first column contains the layers of the current project and the other columns the cross-sections that belong to each layer.

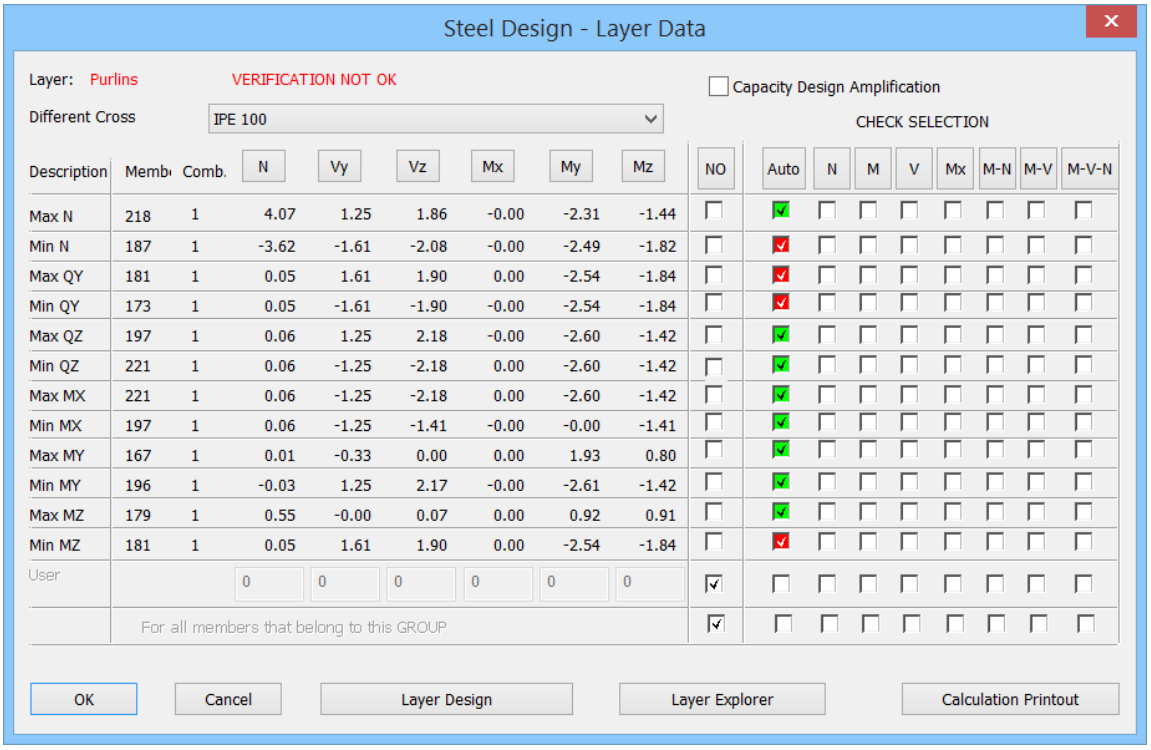
- Select the button "Calculate All" for the calculation of all sections.
- Alternatively, select the layers one by one and then click the button "Calculate".





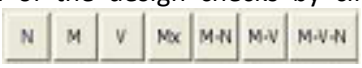
Green color indicates that all sections of this layer satisfy the design criteria (stress/resistance ≤1) red color that they don't.
In order to locate the inadequate members or just see the check results, select the layer and click "Edit".





Getting the mouse indicator over a green cell, a value lower than 1.0 is displayed (adequacy), while on a red cell a value greater than 1.0 (failure) is displayed.

All design checks' results are displayed, in the dialog box above, for all cross-sections of the current layer.

Apart from the automatic process, the user can follow his own check process. Select the combination of the design checks by clicking on the corresponding button from "CHECK SELECTION" 

and then the button "Layer Design".

Checking one by one you could notice that for some members, for example, the M & M-V check fail (red color). This happens because in this case the program uses only the values of MY, VZ and ignores N value (worst case).

You can also type your own values in "User" line and do your own checks.

To read the main results (automatic procedure, manually or "user") click on the button "Calculation Printout" or "Layer Explorer" for all results. The displayed TXT files are those generated by the program for the printout.

MORE DETAILS:

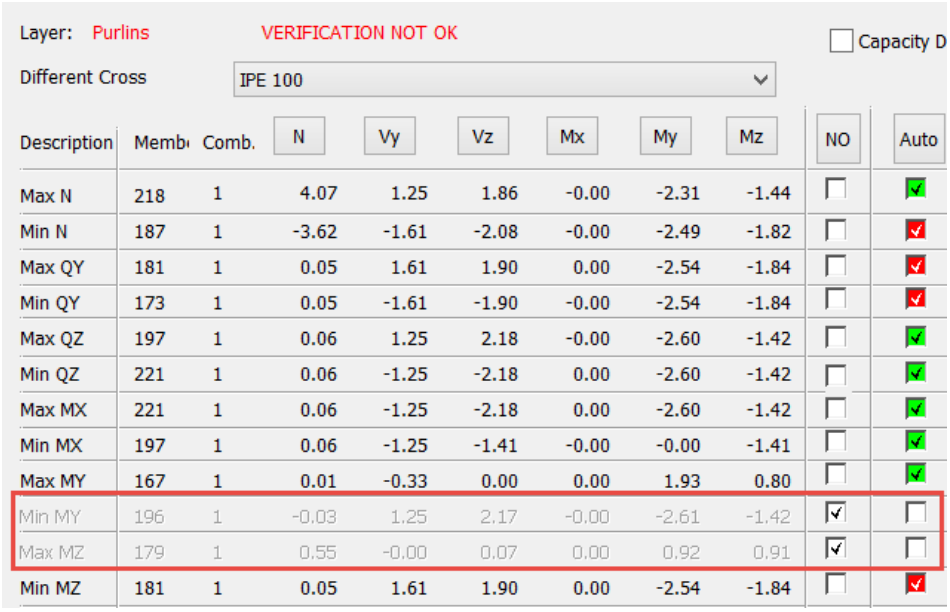
For each section of each layer, the program calculates, for each load combination, the maximum and minimum value of all stress resultants (N, Mx, My, Mz, Qx, Qy, Qz). The load combination that gives, for example, the maximum value of the axial force N and the corresponding structural member stressed with the N, is identified. The other cells of the same line are filled in with the corresponding values obtained for the same member and the same load combination.
In this way a table is created with 12 lines (maximum and minimum value) and 6 columns (6 stress resultants).
-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

The "Member" column contains the number of the structural member with the maximum or minimum value of the resultant stress.
The "Comb." column contains the number of the load combination that corresponds to the maximum and minimum values.

IMPORTANT NOTES:

1. The **sign convention** used by the program:
   Axial force with NEGATIVE sign => TENSION
   Axial force with POSITIVE sign => COMPRESSION
   But in TXT files the condition is the opposite: (+) TENSION, (-) COMPRESSION.

2. The **column** "**NO**" allows excluding one or more maximum or minimum obtained values. To exclude, for example, max Mz and min Mz, activate the checkboxes "NO" in the relative lines. So, for these checks, Mz max and Mz min will be excluded.

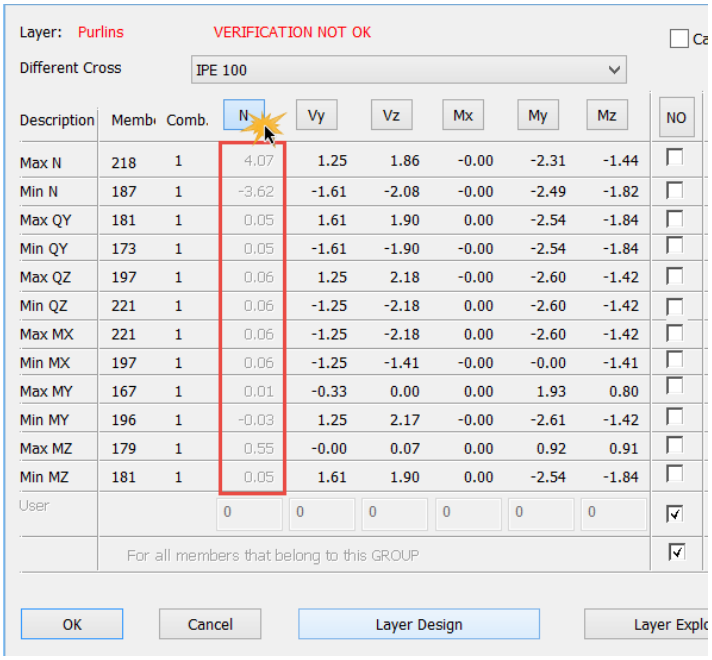


Layer: Purlins VERIFICATION NOT OK Capacity D

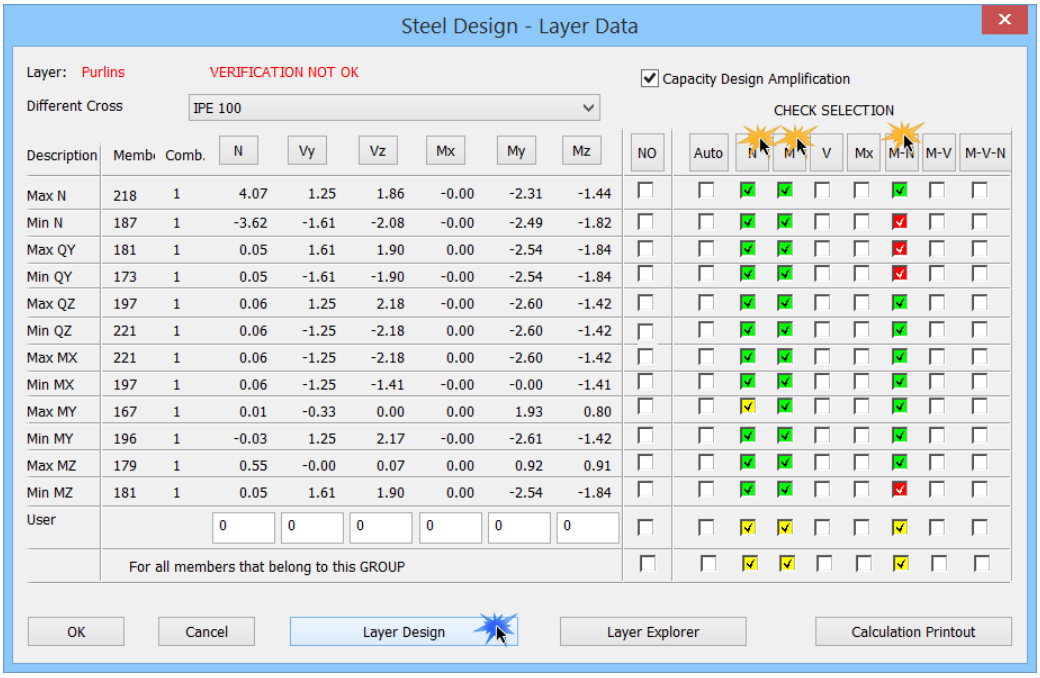
Different Cross IPE 100

Description	Memb	Comb.	N	Vy	Vz	Mx	My	Mz	NO	Auto
Max N	218	1	4.07	1.25	1.86	-0.00	-2.31	-1.44	☐	☑
Min N	187	1	-3.62	-1.61	-2.08	-0.00	-2.49	-1.82	☐	☑
Max QY	181	1	0.05	1.61	1.90	0.00	-2.54	-1.84	☐	☑
Min QY	173	1	0.05	-1.61	-1.90	-0.00	-2.54	-1.84	☐	☑
Max QZ	197	1	0.06	1.25	2.18	-0.00	-2.60	-1.42	☐	☑
Min QZ	221	1	0.06	-1.25	-2.18	0.00	-2.60	-1.42	☐	☑
Max MX	221	1	0.06	-1.25	-2.18	0.00	-2.60	-1.42	☐	☑
Min MX	197	1	0.06	-1.25	-1.41	-0.00	-0.00	-1.41	☐	☑
Max MY	167	1	0.01	-0.33	0.00	0.00	1.93	0.80	☐	☑
Min MY	196	1	-0.03	1.25	2.17	-0.00	-2.61	-1.42	☑	☐
Max MZ	179	1	0.55	-0.00	0.07	0.00	0.92	0.91	☑	☐
Min MZ	181	1	0.05	1.61	1.90	0.00	-2.54	-1.84	☐	☑

3. **How to exclude one or more intensive forces from the layer's design**
   If for some reason you want to exclude one or more intensive forces from the layer's design, press the corresponding Intensive Force column (for example N) and again "Layer Design" to receive the new results without considering the axial forces.



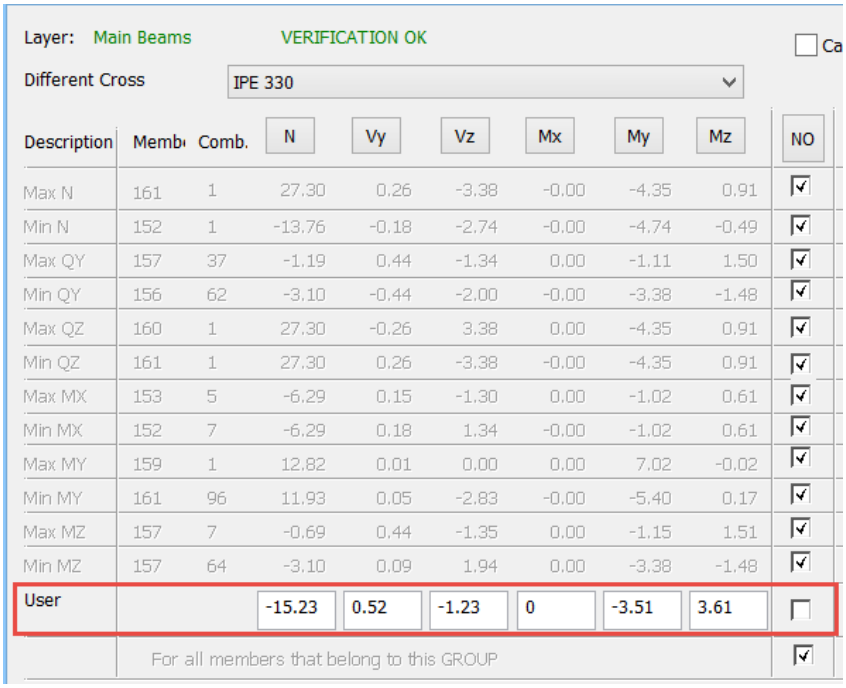
4. The "**AUTO**" **column** offers an automatic process through which the program calculates for each line of internal forces whose check should be done based on the values corresponding to each intensive force. This means that in case of considering N, My, Mz and Mx=Qy=Qz=0 the program performs Bending, Bending with Axial, Compression & Tension checks only and doesn't perform Torsion and Shear checks.
5. Choosing the **manual process** the user is free to check which checks to perform and then click "Layers Design" to see the results:



- a green check means stress/resistance ≤1
- red check means: stress / resistance >1
- yellow check means: not required

⚠ Getting closer the mouse indicator over a green cell, a value lower than 1.0 is displayed (adequacy), while on a red cell a value greater than 1.0 (failure) is displayed.

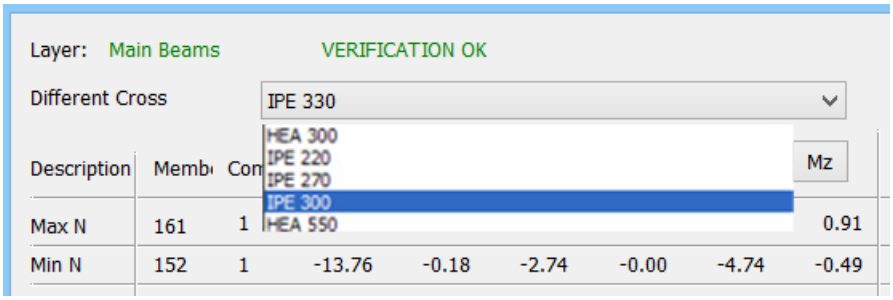
6. Activating "**User**" the user can type his own values of the intensive forces in order to check the sections. In the next dialog box:



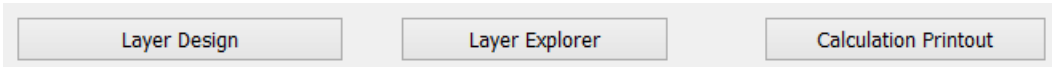
intensive forces are given by the user and those estimated by the program analysis are disabled.

⚠ **Attention to the Convention on the sign of the axial force!!!!**

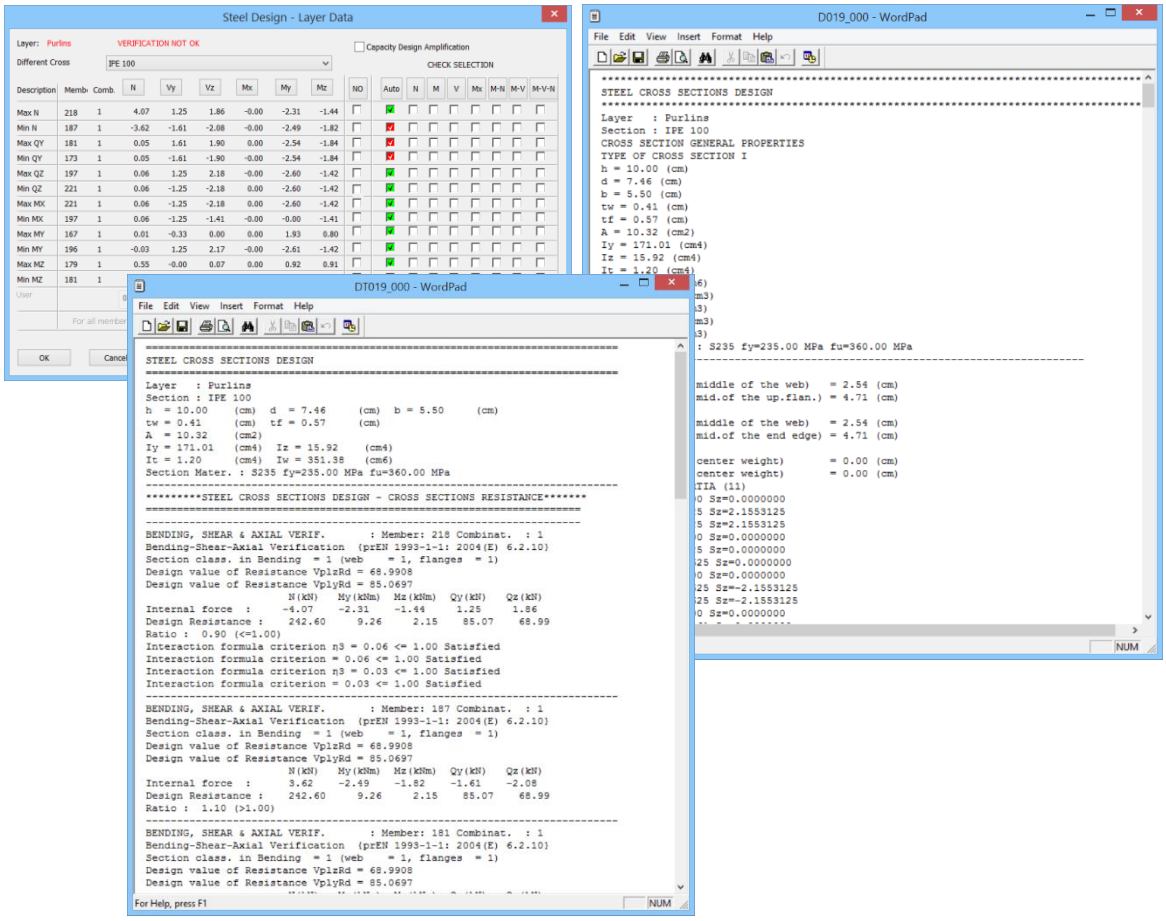
7. “**Different Cross Sections**” contains the different sections included in the "Steel Beams" layer.



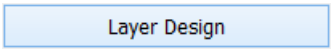
Follow the same procedures described above in order to design manually the other sections or to see the results:



- In table - Analytically - As Printout



8. Activate 

and press 

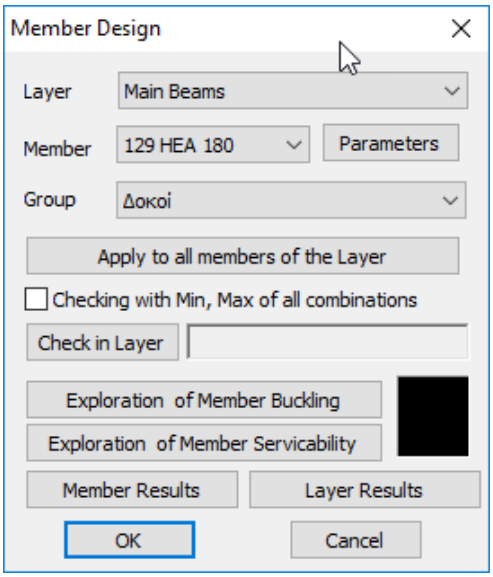
if you want to apply the capacity design in your checks.

2.1.2 Buckling Members Input

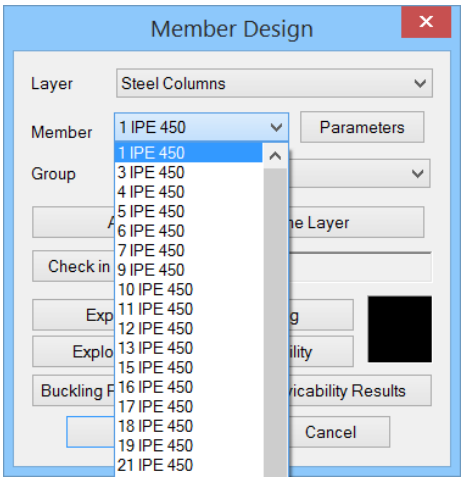
The buckling resistance check is one of the main design checks for steel structural members. Select the command "Buckling Members Input", to apply on each member of each layer the following resistance checks:

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	

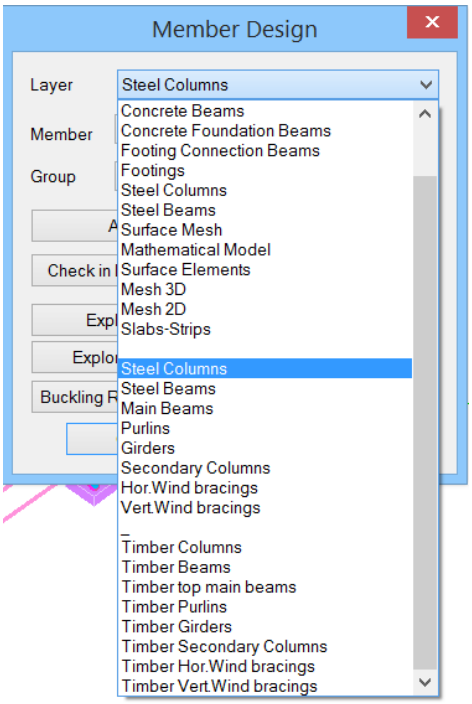
By selecting the command the following window opens:



Checking is performed by layer. So first select the layer from the drop down list and the "Member" list loads all members of this layer and its cross sections.

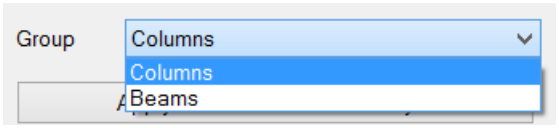


EXAMPLE:



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 for some of them, you can create different "Groups" in the same layer.

The program has two default Groups: "Beams" and "Columns".

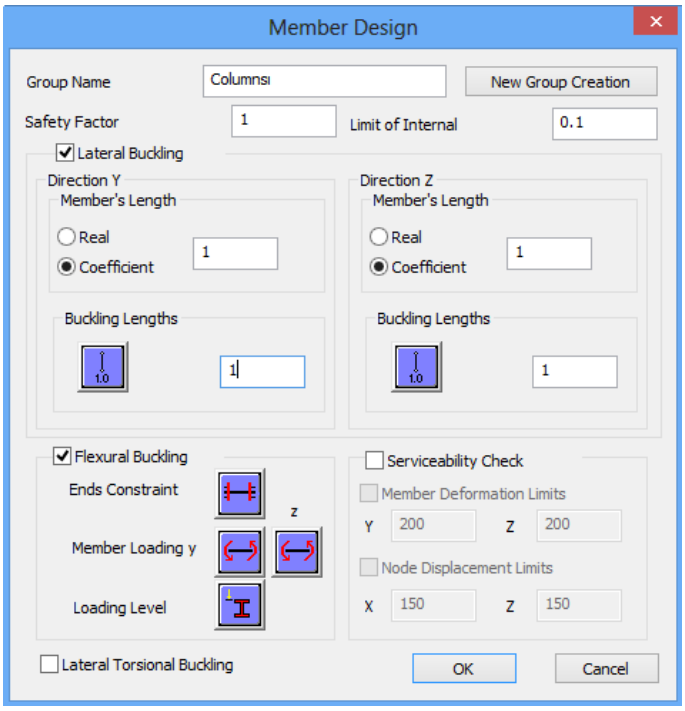


If you want to apply the same parameters to all members of the layer, then set the parameters once, keep the default name "Columns" and press the "Apply to all members of the layer".

Calculations will consider the same parameters for all members of the layer.

Otherwise, in order 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.

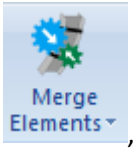
Select a "Layer" and click on the "Parameters", and the following dialog box opens:



In the "Group Name," you see the name of the parameter group. If you want to create your own 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.

NOTE

In older SCADA Pro versions as well as before the command was created 

, the user was asked to specify the length of the member and the buckling length along both directions Y and Z respectively, following the procedure:
In "**Member's Length**":

- By choosing "*Real*", you have to fill in the real length of the member (in m)
- By choosing "*Coefficient*", you have to type in a coefficient by which the different lengths of the members which belong to the particular parameters' group will be multiplied.

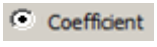
In case you want the program to take into account the real members' lengths, during buckling check, choose "Coefficient" with 1 value.
In case you have some members with different or equal lengths that are laterally secured at the same distance (eg 1/3), then you define the value of 0.33 and of course, you create a separate group of parameters to which these members will belong.

In the new version of SCADA Pro, the buckling length is defined by using the command "Merge Elements " and so no action is required in this field. Having followed the procedure of Merge Elements, in the Parameters field and specifically in the Member's Length, you leave it as it is and proceed with the definition of the remaining parameters.

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 4 parts, one for each check:

- Lateral Buckling resistance check: Activate the corresponding checkbox. Set the length of the structural member and the buckling lengths for both Y and Z directions. On the field "Member's Length" activate the label "Real" 

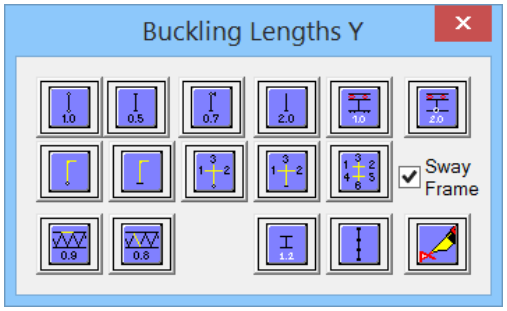
and type the real length in m, or activate the label "Coefficient" 

and type a factor ("1" means the real length).

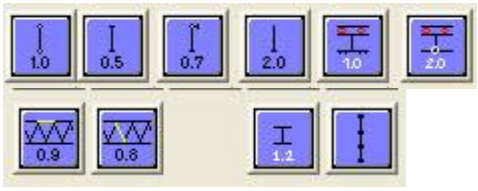
The parameter "Buckling Lengths" depends on the support conditions of the structural member.

Click on the following button 

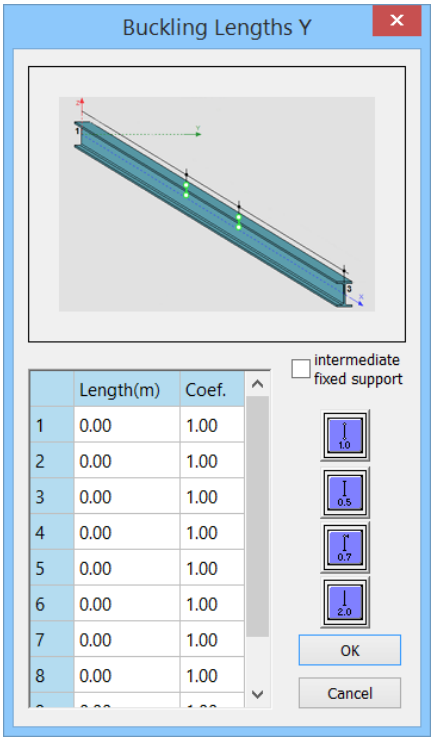
to open the following list and select the appropriate conditions so that the program automatically inserts the corresponding factor.



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



By choosing  you can define the positions of lateral blocks if there are any in order to calculate the corresponding reduced buckling lengths.

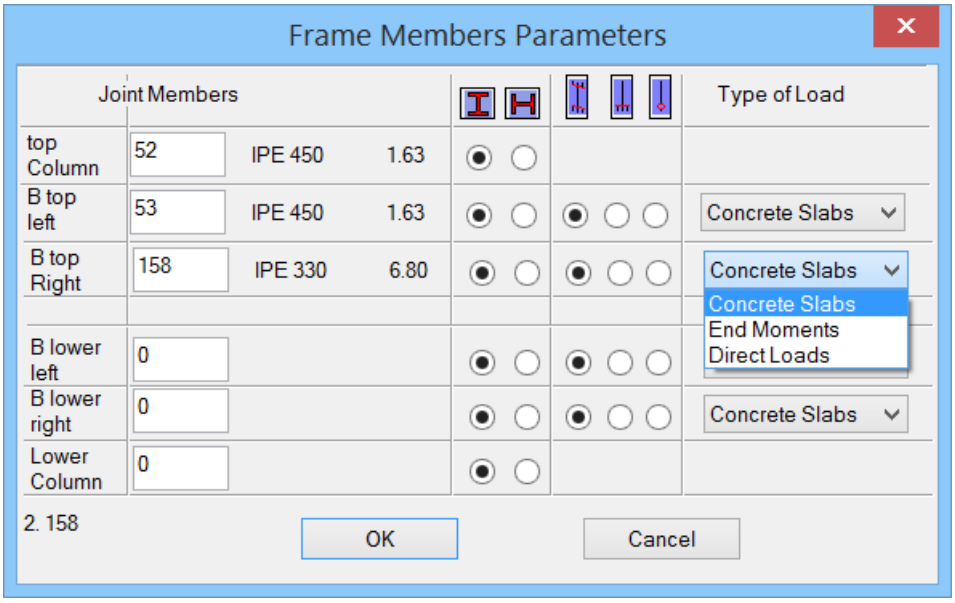


The second group

includes cases of members in multi-storey steel structures and allows setting the concurrent to the node members.

By choosing  (the most complex case) the user sets for the vertical member the 6 Members (2 vertical and 4 horizontal) that offers succor to it (3 on the top and 3 at the end).

By selecting the icon the following dialog box is displayed:



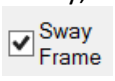
where for the respective fields the respective members that offer succor to the top and the bottom nodes of the member that specifies the buckling length are indicated graphically.

Every time you click a member, in the corresponding field, the number of the cross-section and the length is automatically filled in. To select the members to follow the indications on the left (Top column, Beam top Left, ecc).

Then indicate their orientation an especially for beams the type of support on the other end, and the type of load imposed on them.

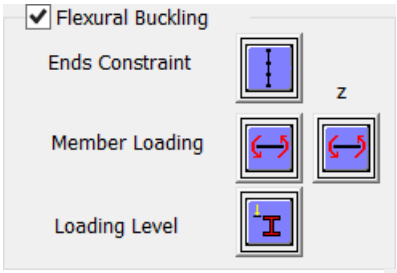
Pressing "OK", on buckling length the corresponding icon and the factor -1 appears, which generally means that the program based on the data you entered automatically calculates the buckling length for this member.

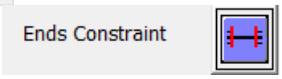
Finally, choosing/user can type his own buckling length.



set if the frame to which the member belongs is transposable or irremovable.

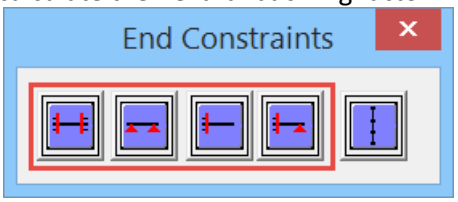
➢ Flexural Buckling resistance check:

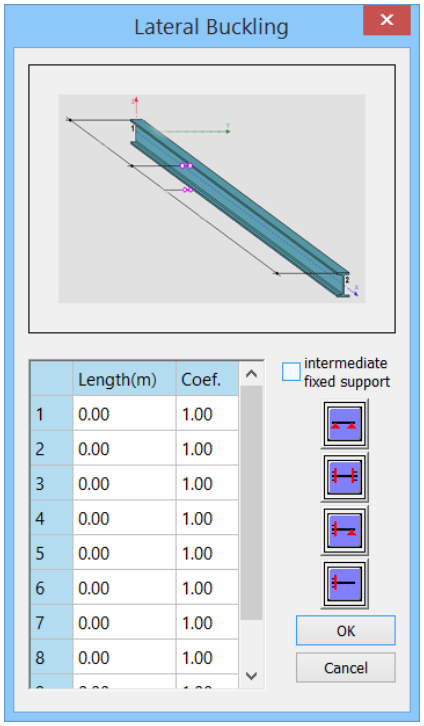


Activate the checkbox and press 

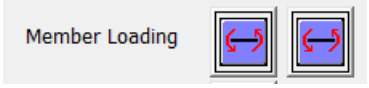
The "End Constraints" window, containing the various types of constraints opens.

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

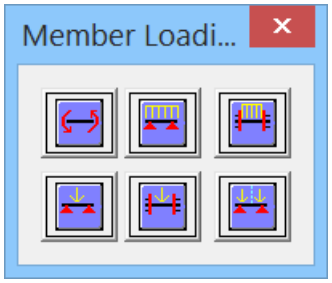




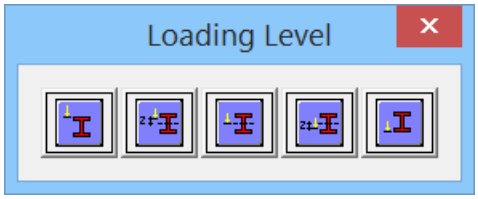
The last one  gives the user the opportunity to consider different constrains along the same member.

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:



In which you choose the type of Member Loading.
Finally, the last parameter/concerns the



determination of the Loading level of the member. The following five options are displayed by selecting the icon.

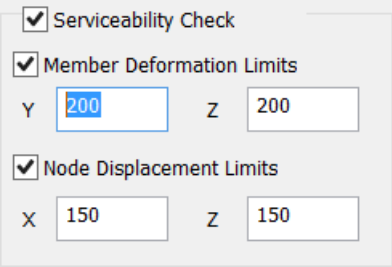
Loading levels for each icon:
1st icon: on the upper flange of the element
2nd icon: near and upward from the axis of symmetry of the element
3rd icon: on the axis of symmetry of the element

4th icon: near and below the axis of symmetry of the element
5th icon: on the lower flange element.

➢ For Lateral Torsion Buckling resistance check: activate the corresponding checkbox.

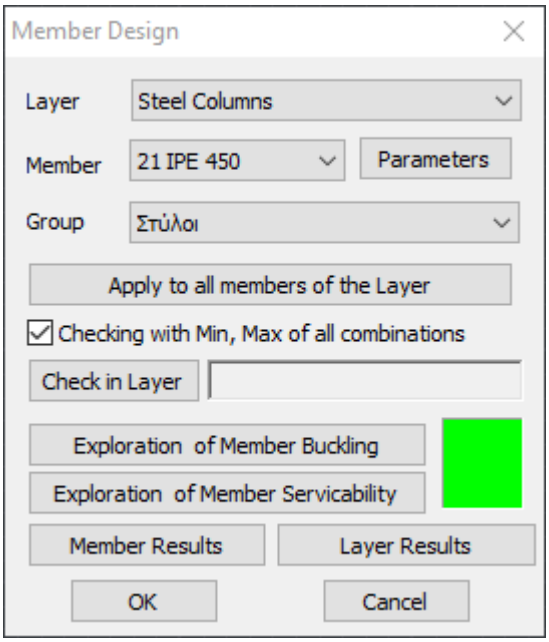
⚠ **NOTE**: For the lateral buckling and the lateral torsion buckling resistance check, the parameters are the same.

➢ 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.



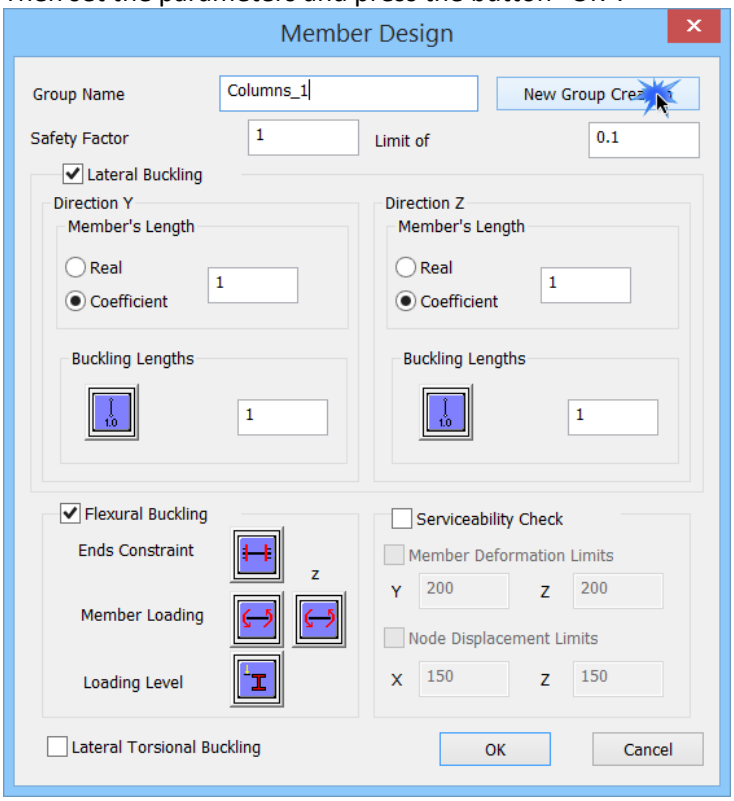
In order to apply the parameters you set to all members of a layer, select the command "Apply to all members of the Layer".
Click the button "Check in Layer" to check every member of the current layer, for every load combination. The results of the design checks are displayed in the black window that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

By activating the option ☑ Checking with Min, Max of all combinations, in checks, only the maximum and minimum values of the intensive forces resulting from all combinations, excluding the intermediate values, will be taken into account, so the process will be completed at noticeably shorter times.

EXAMPLE:

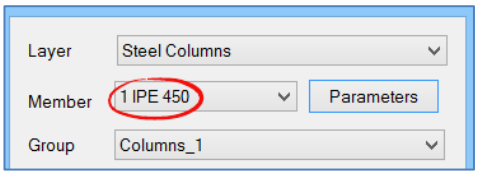
To define another set of parameters for some of the members of the layer follow these steps:

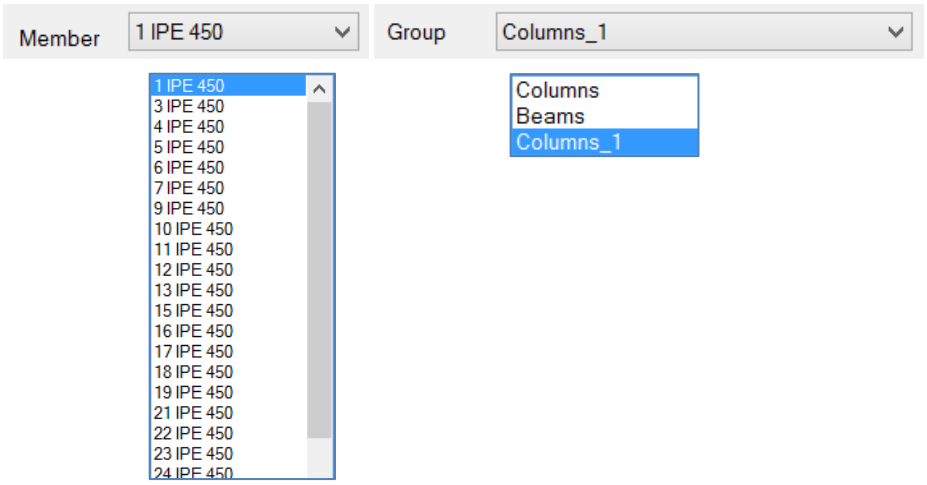
1st step: Press the button "Parameters" and open again the parameters dialog box.
Type the "Group Name" for the new set of parameters that will be created and press the button "New Group Creation".
Then set the parameters and press the button "OK".



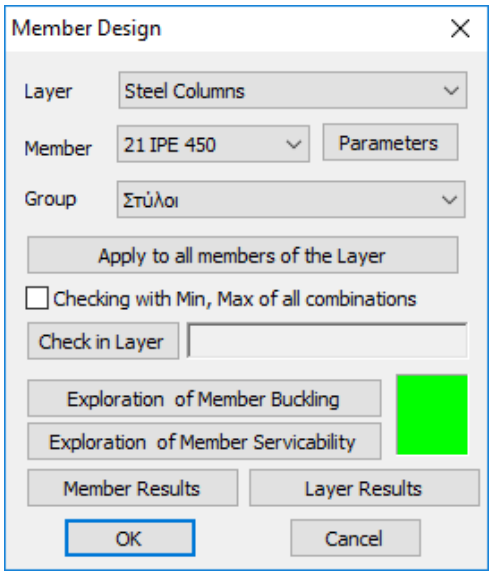
2nd step: define the members of the layer that will belong in “Columns_1” group.

Returning to the original dialog, the only member that gets the parameters automatically, is the current member in the list of members.
All other members have the parameters of the group “Columns”
To move the members from a group to another, select each one from the Members list and change the Group.



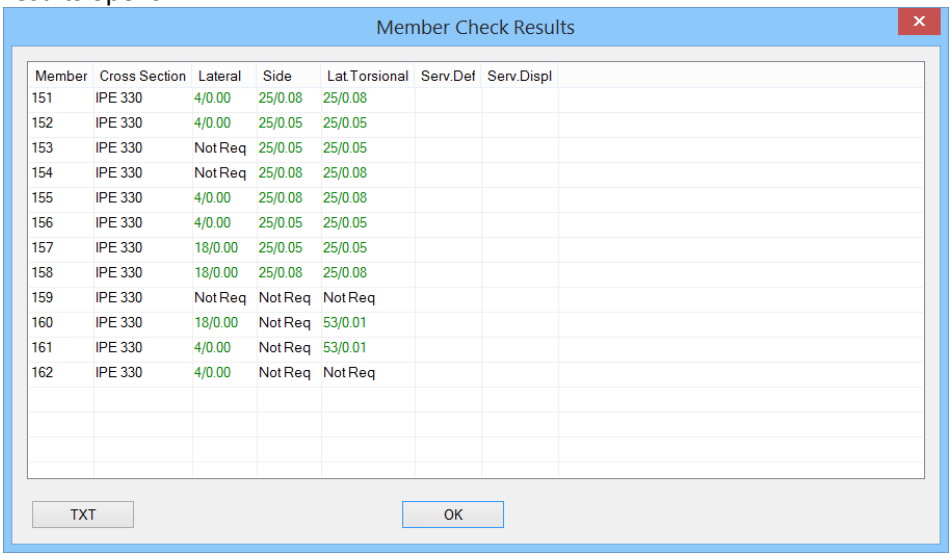


As soon as the layer check is completed, the color of the window on the right gives the information:



Red: if there is a failure
Green: if there is no failure.

By double clicking on the colored window, the dialog box containing members check summary results opens:



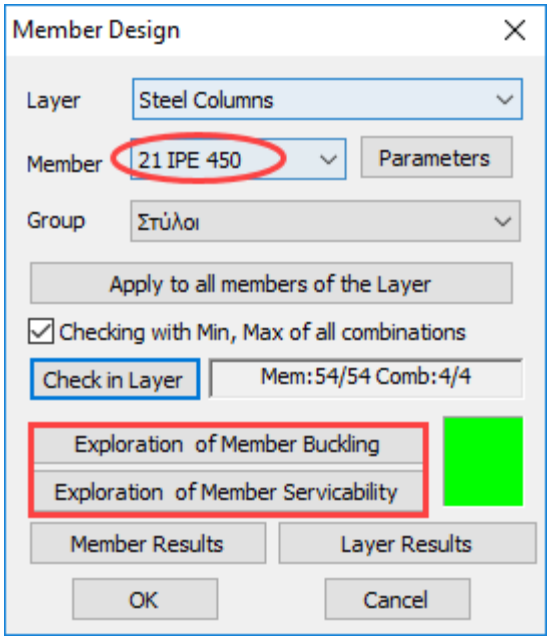
The first column indicates the number of the member, the second column indicates the cross section and in the next five columns, the least favorable ratio of strength and the combination number from which this ratio resulted is displayed.
Greens are the ratios below unity and red the ratios above it.
"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 for all combinations.

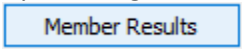
For each group of (N, My and Mz) the checks are made 4 times based on the following combinations:
N with min My and min Mz
N with min My and max Mz
N with max My and min Mz
N with max My and max Mz

⚠ That's why in the output results and in 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) opens the files containing the analytical results of all checks for all combinations for the active member.

By selecting Results the files that include the summary results of the checks on the active member



and all members of the active layer/opens.

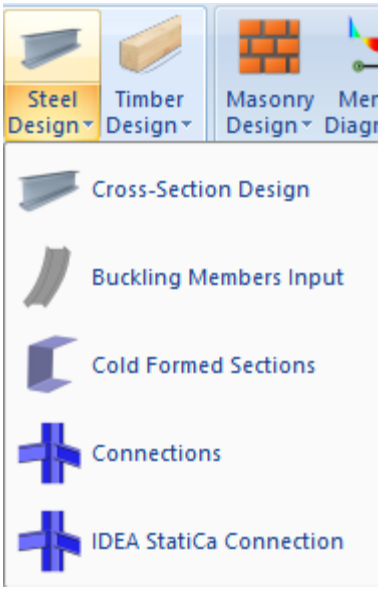
2.1.3 Cold Formed Sections

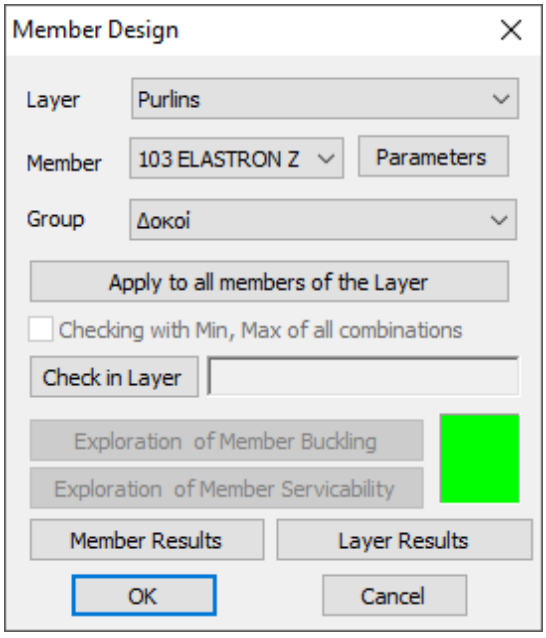
This command concerns the checks of cold formed sections.
The design of cold formed sections concerns the:

- **Resistance check in cross sections**
- **Resistance check in members**
- **Serviceability check**

The procedure of selecting the members and the checks that are going to follow is similar to the warm formed sections' buckling.

The main difference between the warm formed and the cold forms sections elements are that cross-sections' and members' checks are now done with **a common command** (see figure) rather than separately. An important feature is that all members and their cross sections are checked **for all combinations**.





For the rest, the steps to design are the same as those for warm formed elements (per layer, members' merge, buckling parameters, etc.).

DESIGN PRINTOUT

The design results are displayed either per member or per layer. In the second and more general case, the printout's form is the following:

1. **Page 1: General cross-section 1 data**
   Information about dimensions and properties of the initial and virtual cross-section
2. **Page 2: Active cross-section 1 (A part)**
   Information about N, My, and Mz ratios of active cross section dimensions
3. **Page 3: Active cross-section 1 (B part)**

Information about N, My and Mz ratios of active cross-section properties

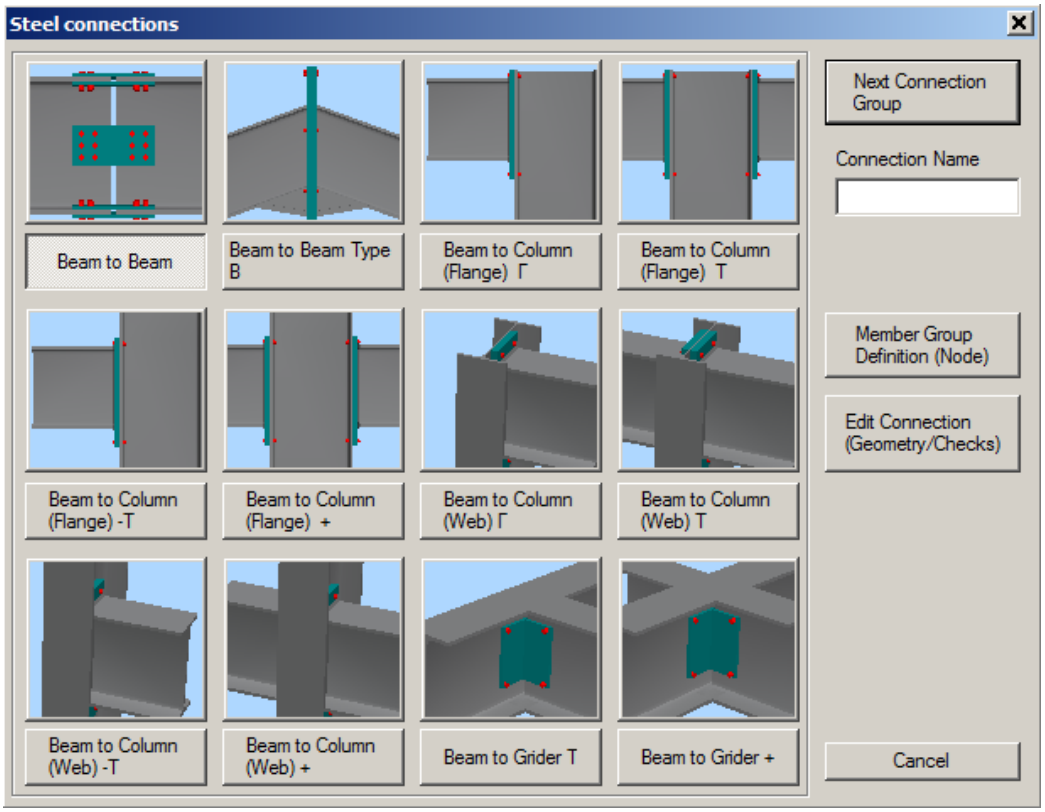
4. **Page 4: Cross-Section check for the 1st member with cross-section 1**
   Resistance checks based on §6.1
5. **Page 5: Member check for the 1st member with cross-section1**
   Resistance checks based on §6.2 & 6.3 and serviceability check based on §7
6. **Repeat steps 4 & 5 :**
   If multiple members have the same cross section within the layer.
7. **Repeat steps 1 to 6:**
   In case of multiple cross sections within the layer.

The printout per layer can also be extracted while creating the **Study Printout**.
(See "Cold Formed sections" in the User's manual)

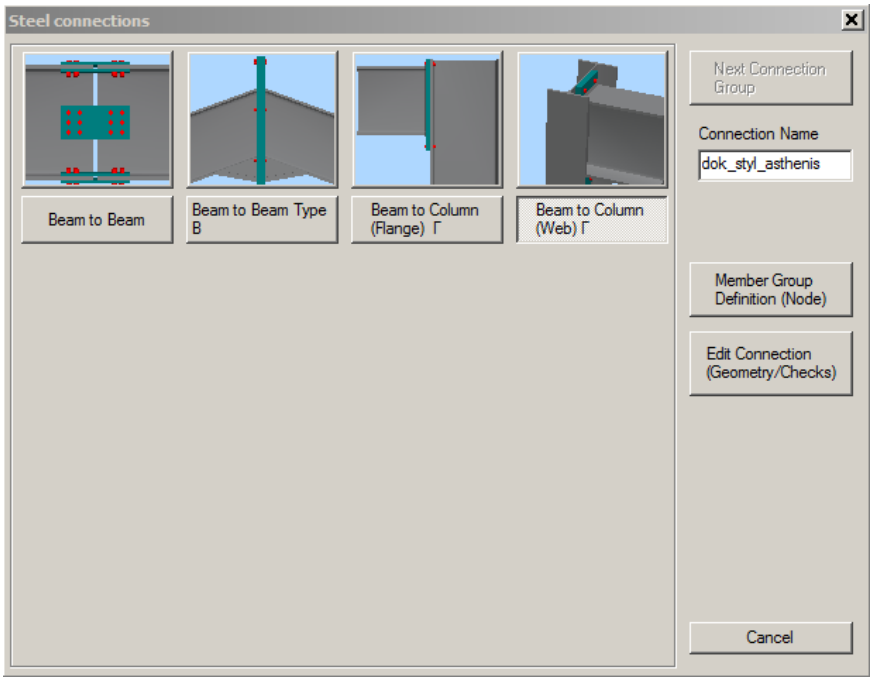
7.1.4 Connections

The last command of the group command "Steel Members Design" is the "Steel Connections", used for the steel connections' design. Select the command and choose one of the following steps:

A) Right click on the screen to open the library that contains all the available steel connections and select the appropriate one. Click on the button "Next Connection Group" to see more connections.



B) Select with left click the members that you want to connect together. Then right click to open a library that contains only the suitable connections for the selected members.



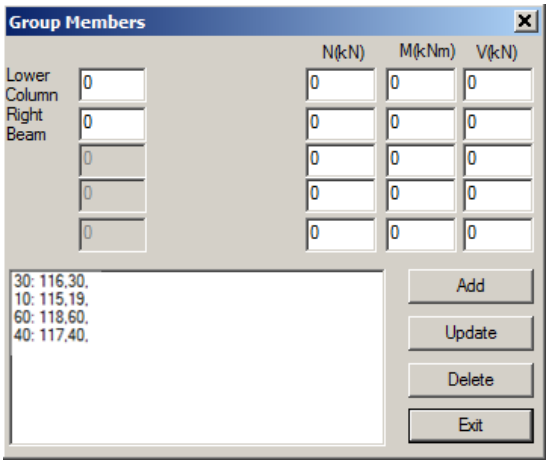
EXAMPLE:

Left click to select member 30 (column) and member 116 (beam) and right click to open the library with the 4 possible types of connection. Select the last one “Beam to Column (Web) Γ”.

Fist, type a name (e.g. dok_styl_asthenis).

⚠ No space between words.

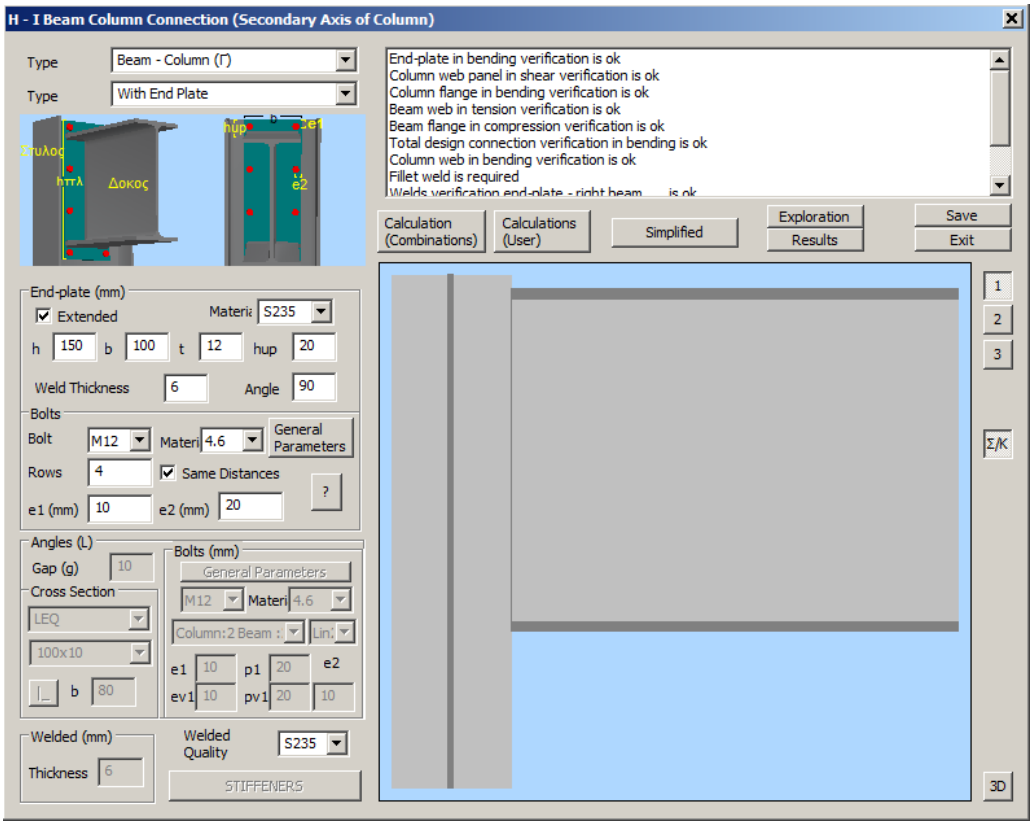
Then, select the “Member Group Definition (Node)” command and in the dialog box you can add more groups of members with the same connection features (i.e. column – beam) or type your own values for the stress resultants N, M, V for the existing groups.
To add groups of members, click into the field “Lower Column” and pick the column 19. Then click into the field “Right Beam” and pick the beam 115 (or just enter the numbers in the corresponding fields) and then click the button “Add”.



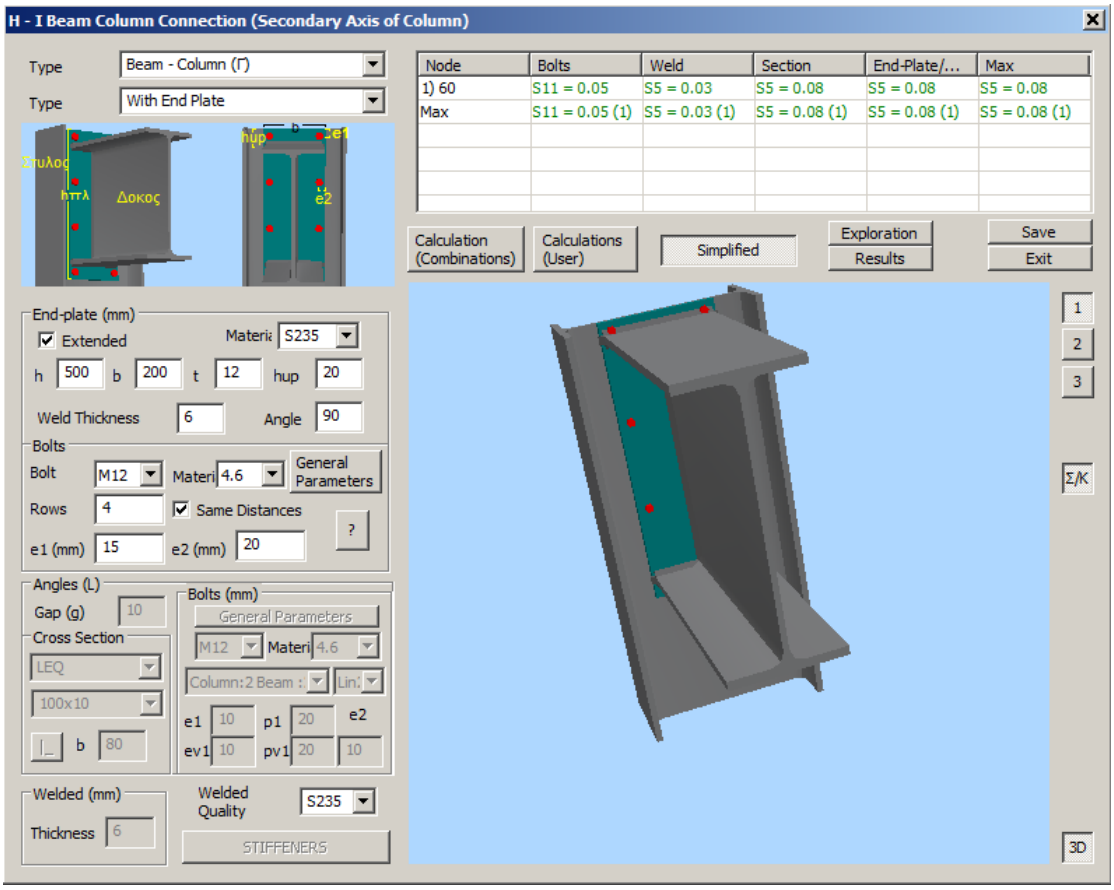
Use this dialog box for the design of steel connections with the same type and the same cross-sections in total (i.e. column IPE 450 - beam IPE 330).

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. Then select the material and define the bolts’ parameters. In each type of connection, the relative parameter fields are active. Then for the design calculations by using the analysis’ combinations select the command “Calculation (Combinations)”.
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 14 to 15 cm and then click again the button "Calculation (Combinations)".



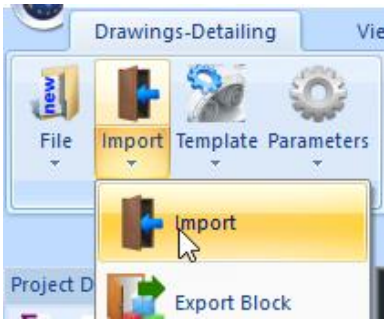
Click the button "3D" to see a three-dimensional representation of the connection that is updated as you change the parameters.
The buttons "1", "2", "3" are used for the display of the two side views (1 & 2) and the plan view (3). The button "Σ/K" is used for the display of the three-dimensional representation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



If the geometrical checks are satisfied, the program calculates and displays all Eurocode 3 design checks for the connection. Click "Simplified" to see the results. Green fonts mean adequacy and red failure. If all checks are satisfied the program will be able to save the connection and generate the drawings automatically. Otherwise, the procedure will stop and you need to change some values of the connection to continue. To read the main results click the button "Results" and for all the results, click the button "Exploration". The displayed *txt files are those generated by the program for the printout.
Click "Save" and then "Exit" to return to the connections' window.

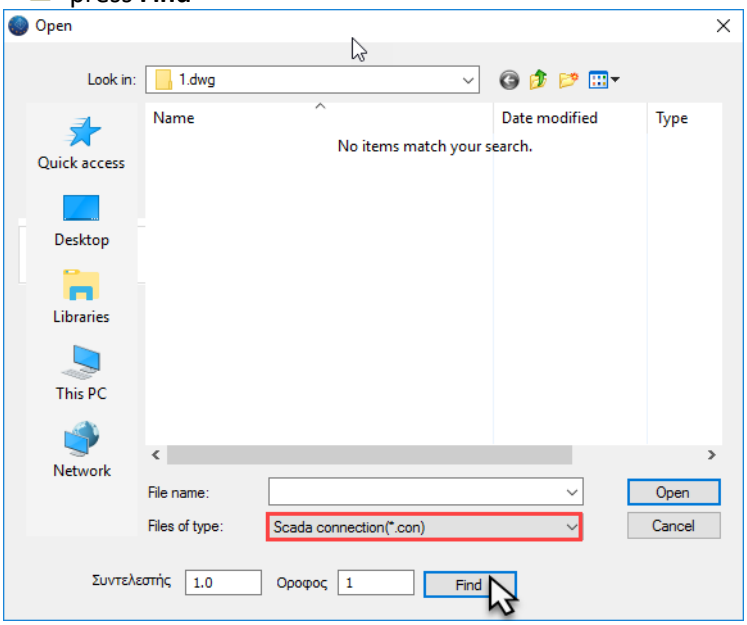
Connections' drawings are located in the folder of the project "sxedia":
C:\SCADApro\ "project name" \scades_Synd\sxedia

You can import them using:



And in the dialog box:

- in Files of Type select **SCADA Connection**
- press **Find**



In the Search File in the window that opens, select the connection to import the designs, views and section, and the detailed table of the link elements.