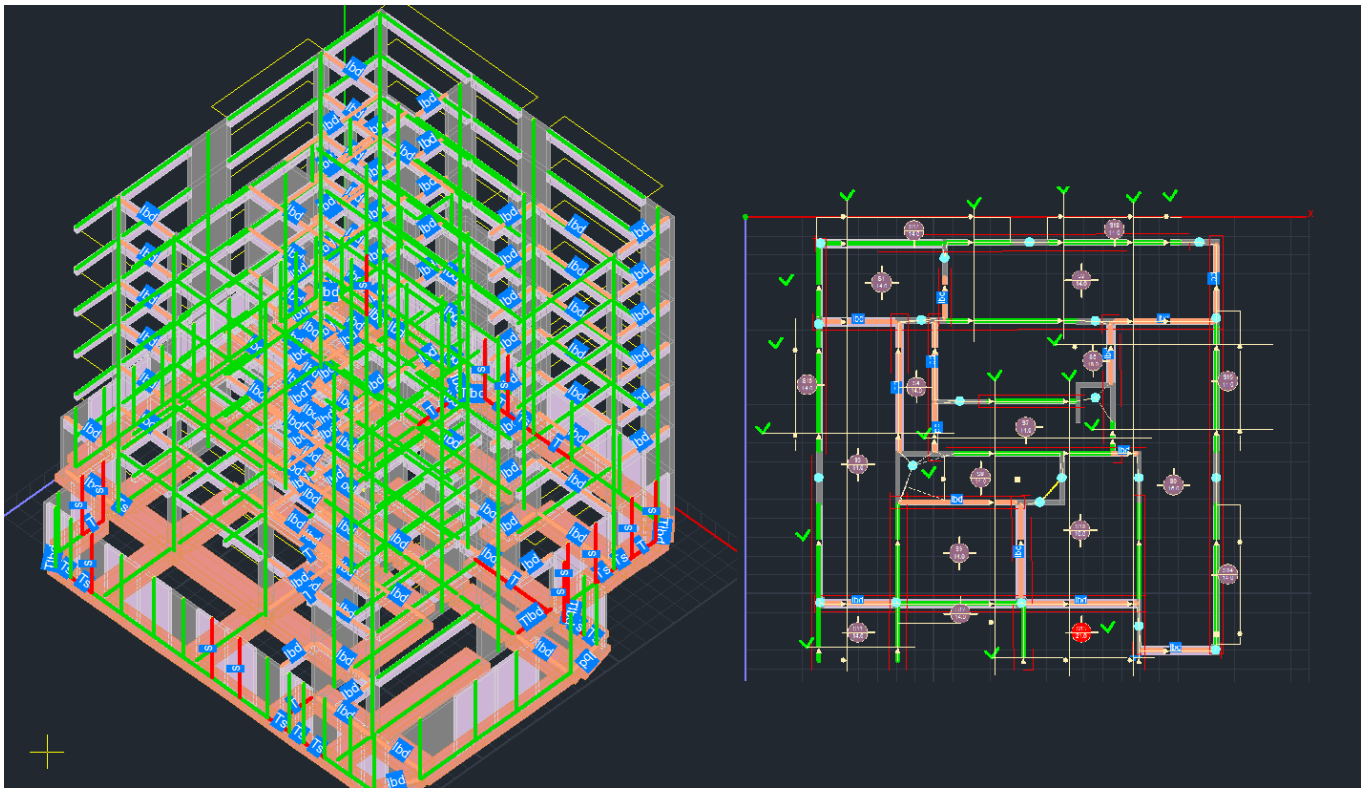
SCADA Pro™
Structural Analysis & Design

User's Manual

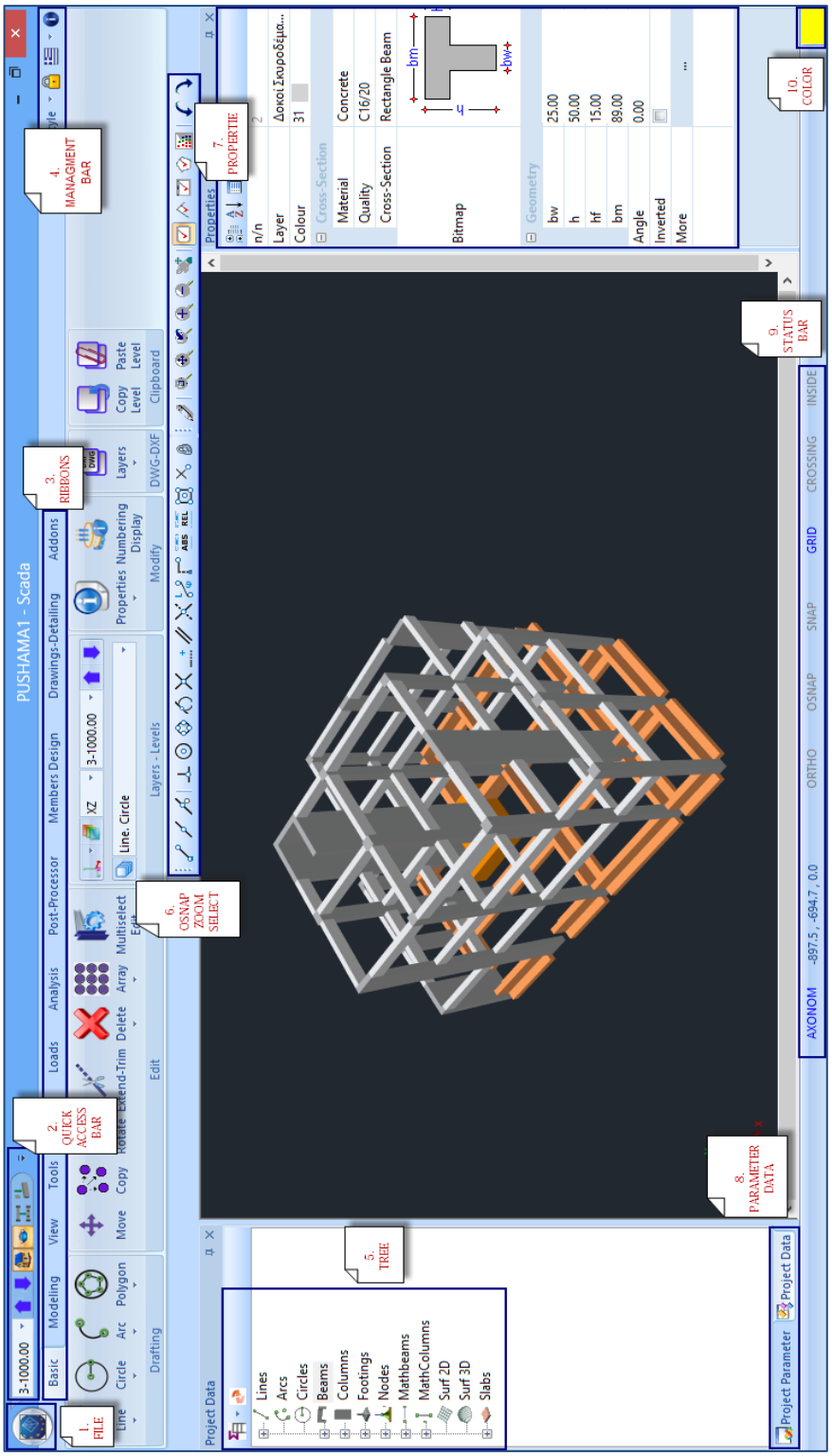
9.MEMBERS DESIGN



ACE HELLAS
INTEGRATED SOLUTIONS

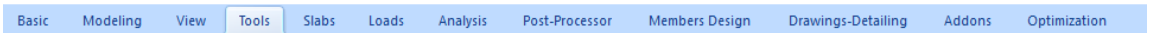
CONTENTS

I. THE NEW UPGRADED INTERFACE of SCADA Pro

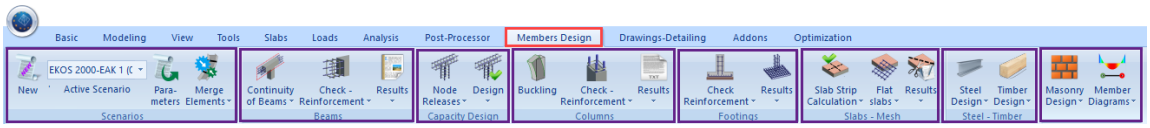


II. DETAILED DESCRIPTION OF THE NEW INTERFACE

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



Members Design

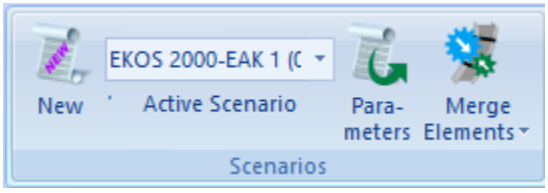


The 9th Unit entitled "Members Design" contains the following eight groups of commands:

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.

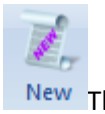
1. Scenarios

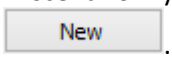


The “Scenarios” command group contains the commands for the creation of a new scenario as well as the editing of the parameters of the design checks and reinforcement in every type of structural elements.

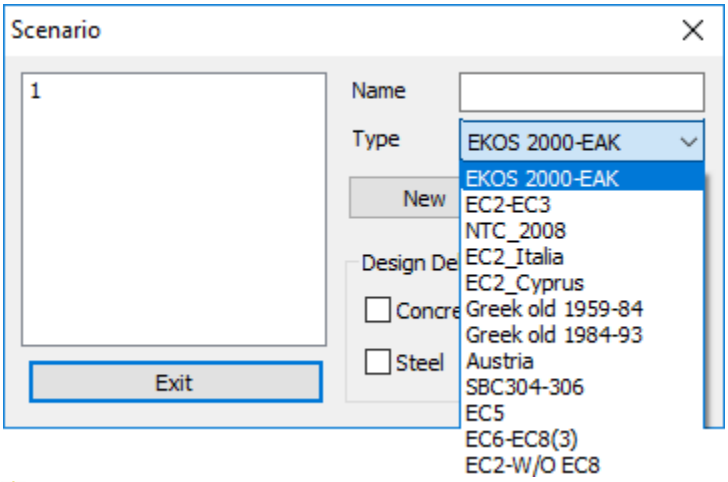
Furthermore, a new group of commands is added, concerning merging of elements (steel and timber)

1.1 New



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

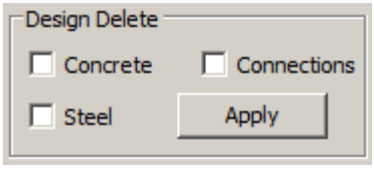
.



⚠ **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.

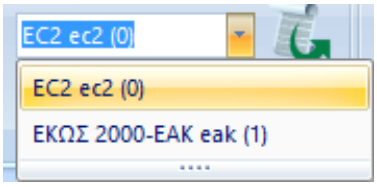
To modify an existing scenario press the button “Update”**/.**



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.

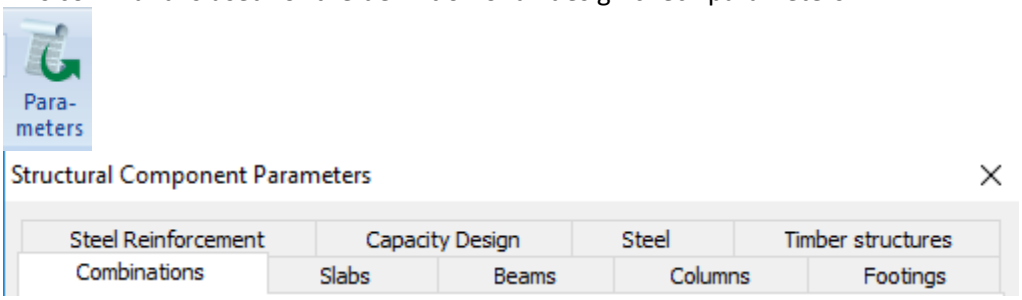
1.2 List

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.



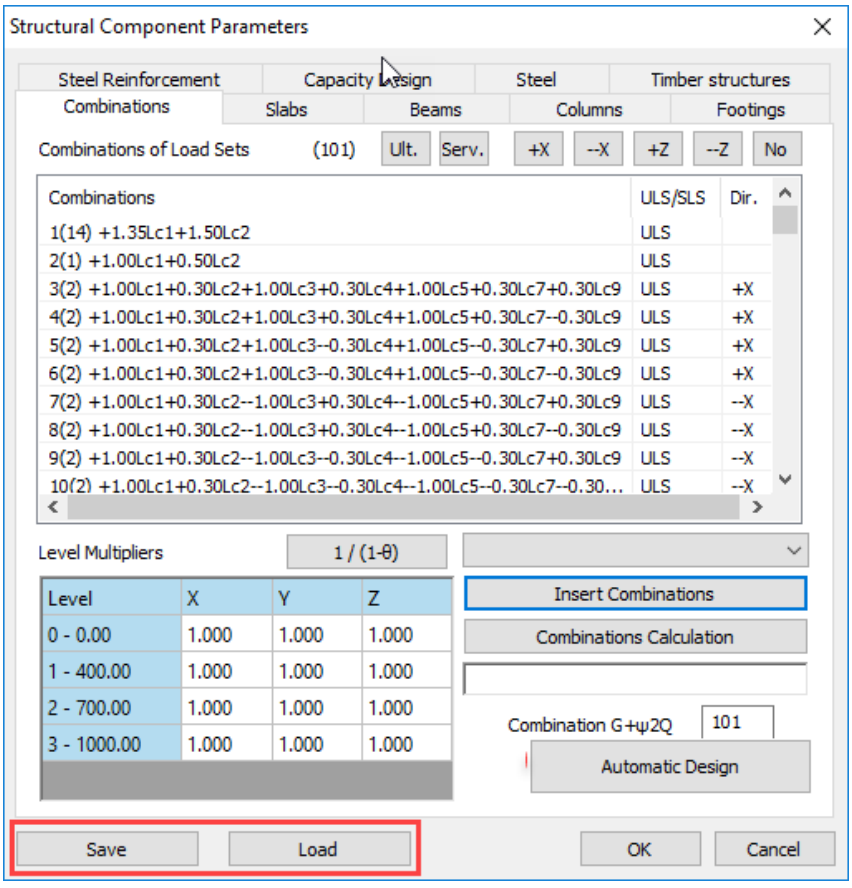
1.3 Parameters

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



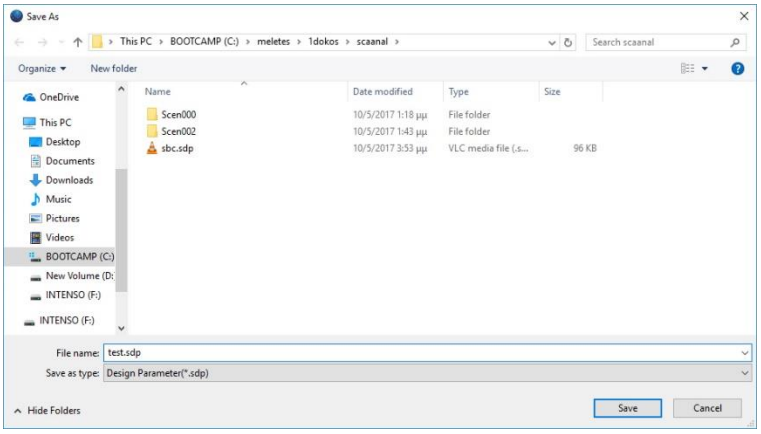
NOTE

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.

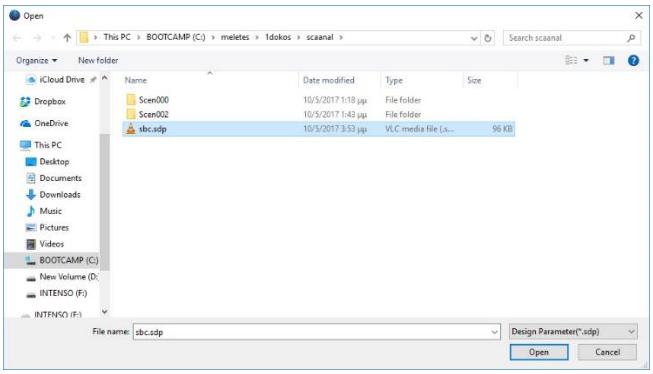


Once you configure the dimensioning parameters, you can now save them to a file to use them in your next projects.

Press “Save” and type a name

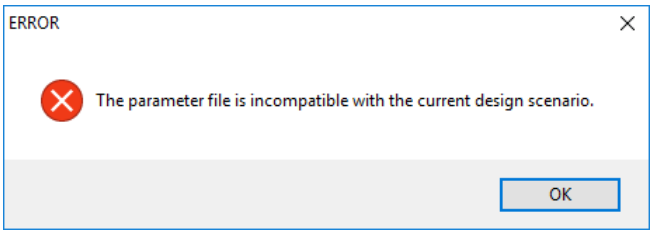


The file extension is sdp scenery design parameters.
Use “Load” command to apply the parameters that are already saved.



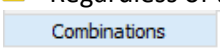
ATTENTION

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:

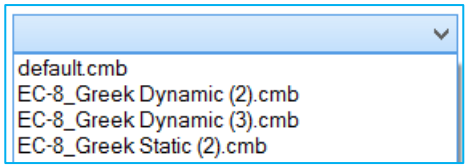


1.3.1 Combinations

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



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

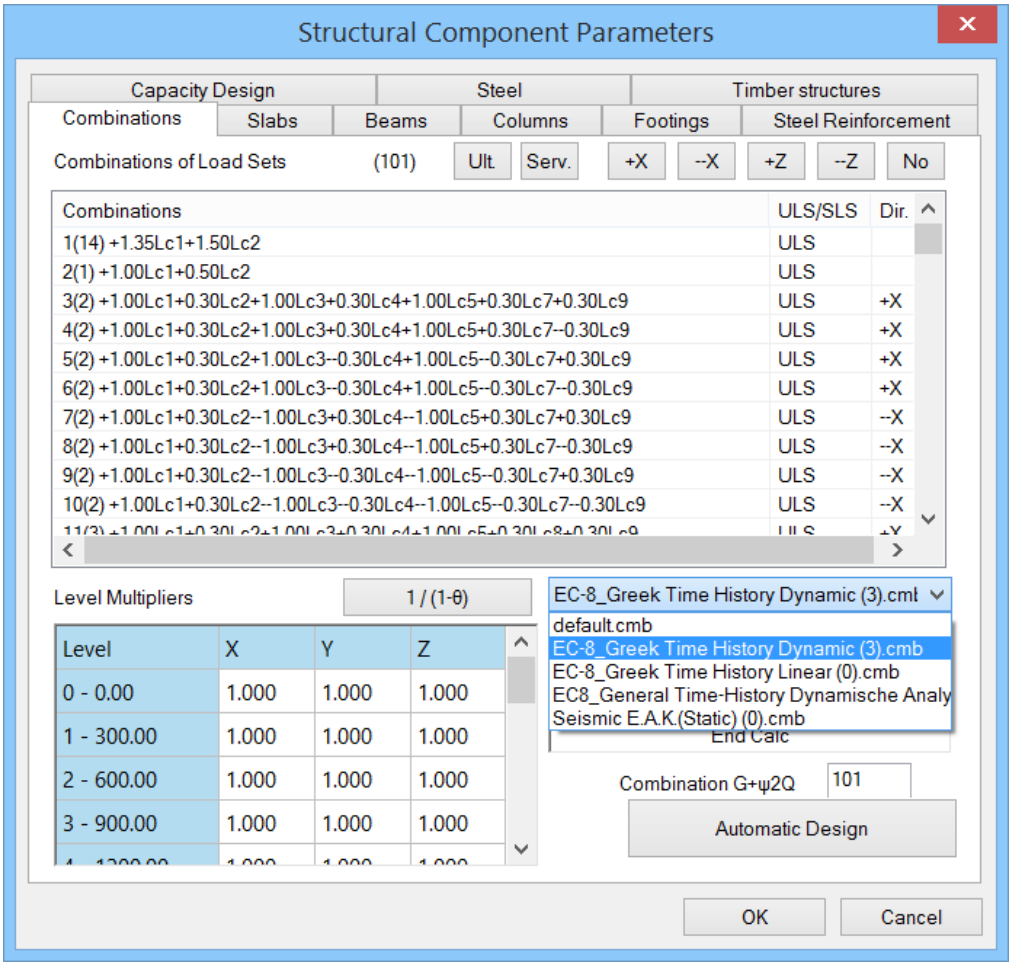
-from the dropdown list 

with automatic calculation or

- Through the command 

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

.



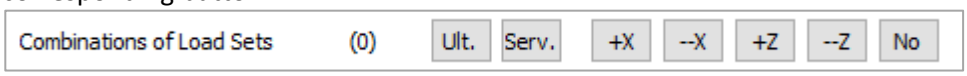
⚠ **NOTES:**

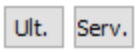
Depending on the case and the fulfilled conditions, you can use either the static or dynamic combination for design. You can also select combinations from different analysis scenarios to check the deviations, on the designing members, between them.

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 

.

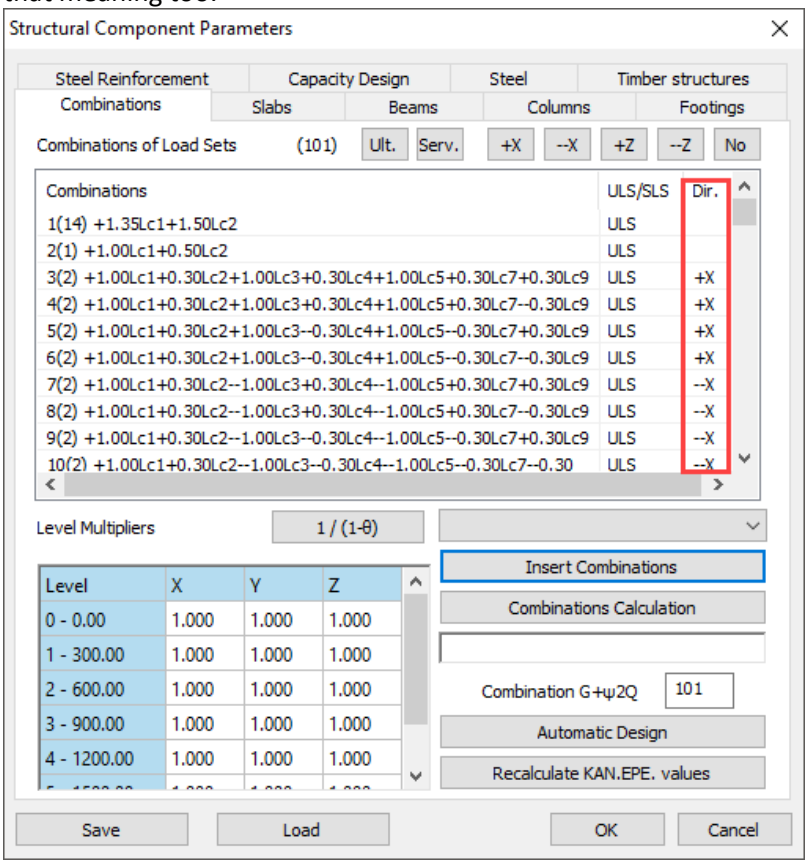
In the column “Dir.” That indicates the direction of the participation for the specific capacity design combination, by selecting the corresponding button 

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.

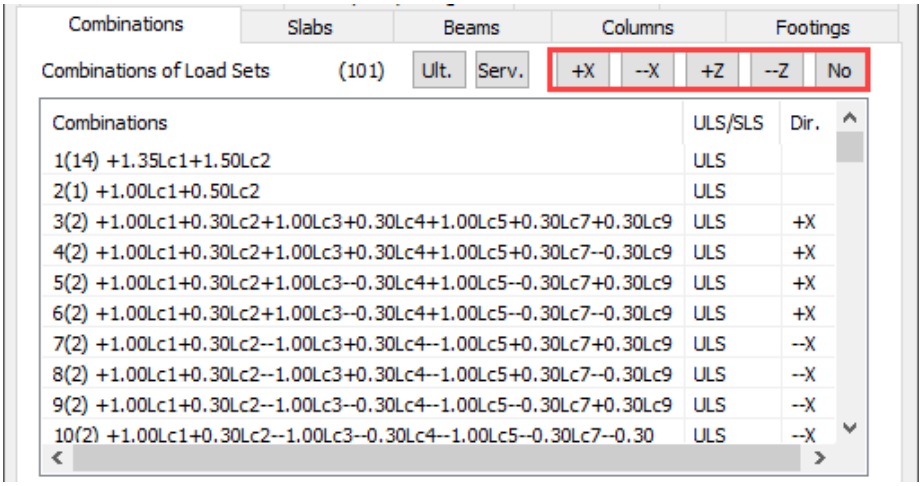
In conclusion, we would assume that in case we want to exclude one direction from one column to avoid the capacity design check, we move to the definition and choose the direction of the local axis that is parallel to the direction of the earthquake we want to exclude.
In case that either the column 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

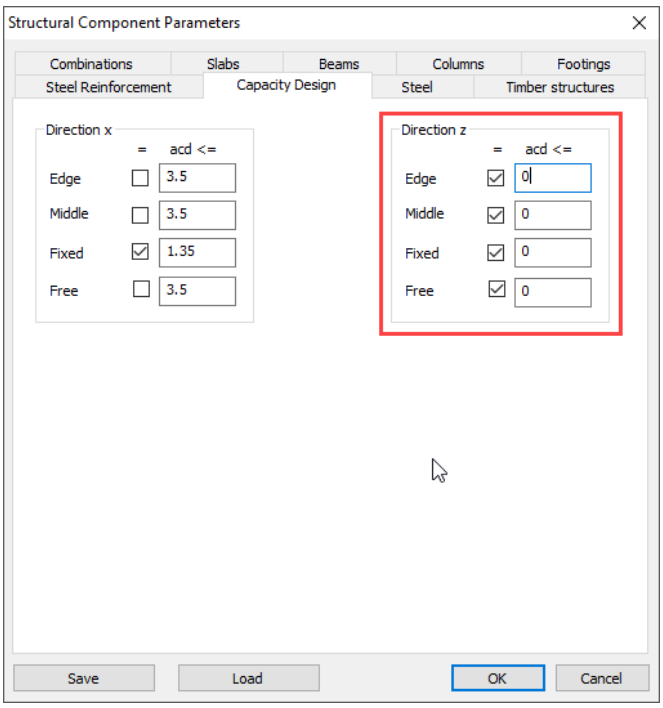
COMB.	SMRby	SMEby	acdy calc	acdy	SMRbz	SMEbz	acdz calc	acdz
3	134.000	15.876	10.973	4.000	134.000	2.907	59.929	4.000
4	134.000	15.876	10.973	4.000	134.000	2.907	59.929	4.000
5	134.000	15.569	11.189	4.000	144.800	4.605	40.880	4.000
6	134.000	15.569	11.189	4.000	144.800	4.605	40.880	4.000
7	144.800	15.569	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

“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 

to take into account the P-Delta effect during the design check. The stress resultants will be increased automatically at the corresponding levels, in which 0.1 < θ <0.2.

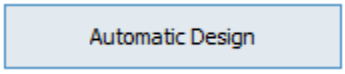
ATTENTION:

For modification purposes, press the following button 

.

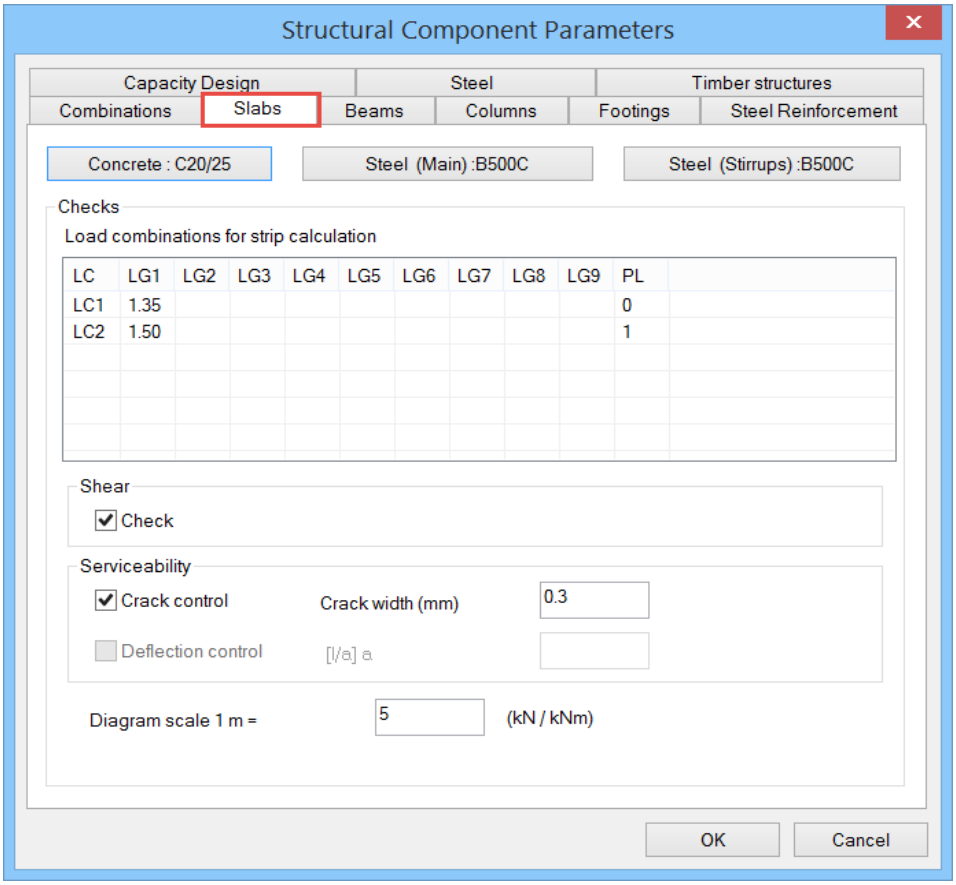
The following field/concerns only the Greek EKOS.

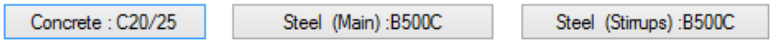
Automatic Design:



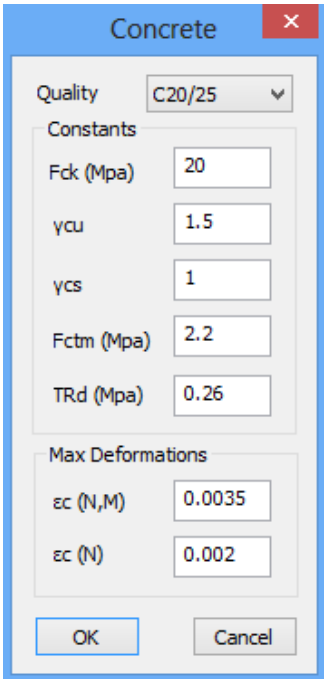
This command offers the possibility for an automatic application of the appropriate design checks and the automatic designing of all structural elements, just by pressing the corresponding button. Set the parameters in the various tabs and then press the button “Automatic Design” or follow step by step the procedure to design the structural elements concerning the fulfillment of the design checks.

1.3.2 Slabs



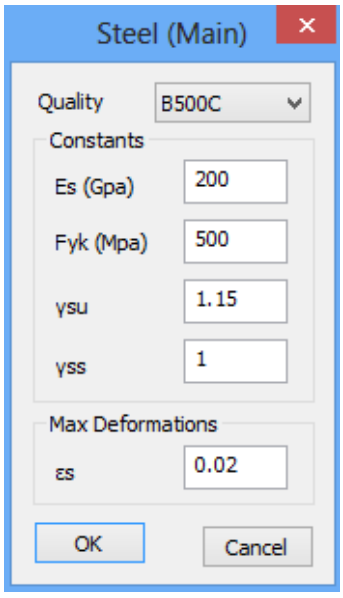
Press the following buttons 

to change the material of the slabs: concrete and the steel rebar’s type.



By choosing a different concrete type, the corresponding parameters are updated automatically.

⚠ More specifically,
"**F_{ck}**": the characteristic cylinder strength in MPa.
"**γ_{cu}**": the partial factors for concrete in the ultimate limit state
"**γ_{cs}**": the partial factors for concrete in the serviceability limit state.
Fctm: the tensile strength of concrete in MPa
Trd: the shear strength of concrete in MPa
In the field "Max Deformations" the maximum deformation of concrete is defined for the simultaneous action of bending (M) and axial load (N) or just for the axial load.
You can also type a different value from the default one.



In Steel (Main & Stirrups) dialog box there are the parameters of the steel reinforcement.

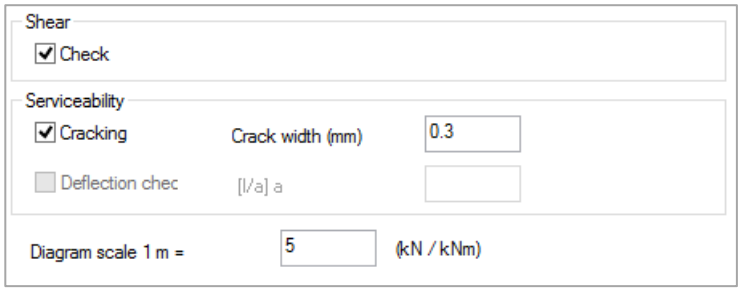
⚠ More specifically,
"**E_s**": the design value of modulus of elasticity of the steel reinforcement in GPa.
"**F_{yk}**": the characteristic yield strength of the steel reinforcement in MPa.
"**γ_{su}**": the partial factors for steel reinforcement in ultimate limit state
"**γ_{ss}**": the partial factors for the steel reinforcement in serviceability limit state.
In the "Max Deformations" field the maximum deformations of the steel reinforcement are indicated.

In the "Checks" field the following dialog box can be edited as appropriate.

Loads combination for strip calculation

LC	LG1	LG2	LG3	LG4	LG5	LG6	LG7	LG8	LG9	PL
LC1	1.35	0.00								0
LC2	1.50									1

- The live and dead loads’ factors considered during the strip slab calculation are displayed and can be modified as appropriate.
- PL column refers to the **adverse loads of slabs**. 0 for not considering the corresponding line’s loads and 1 to consider them.

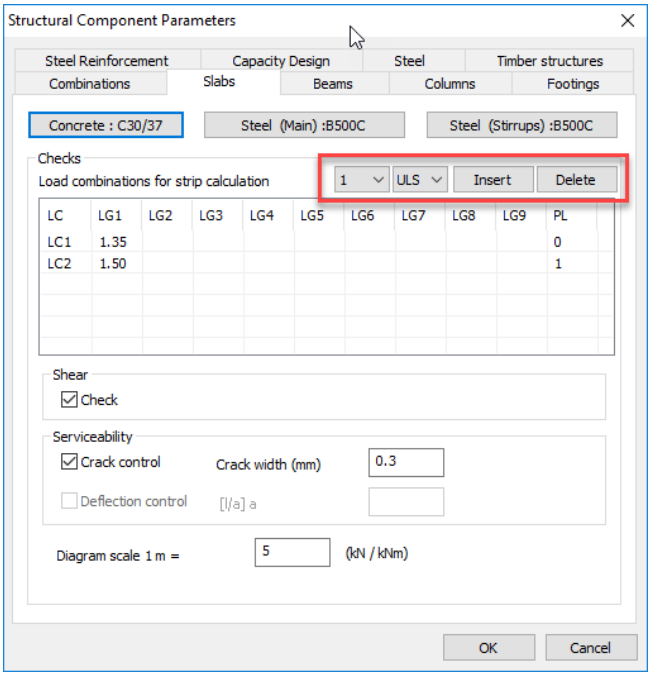


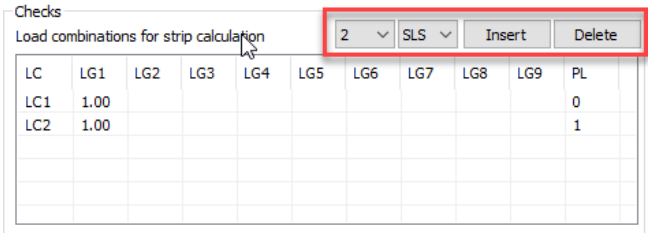
Activate the relative checks for the slab verification.
Define the value in the field “Crack width” for the concrete cracking check.
Define scale for diagrams’ display.

NEW FEATURES

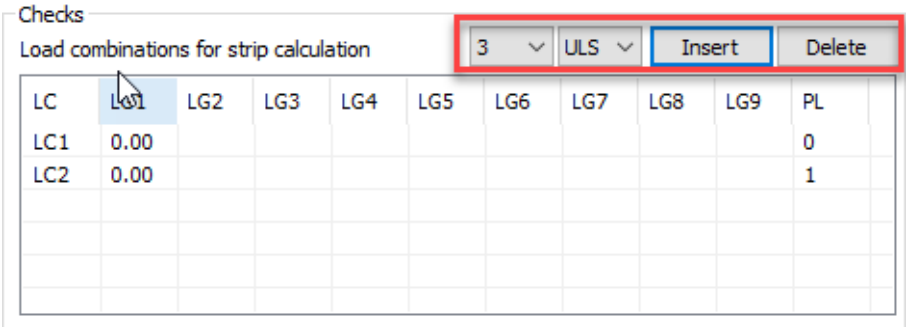
§ More combinations for the slab’s design

In the new version of SCADA Pro, there is the possibility of introducing more combinations for the slab’s design. In the design parameters window, inside Slabs:





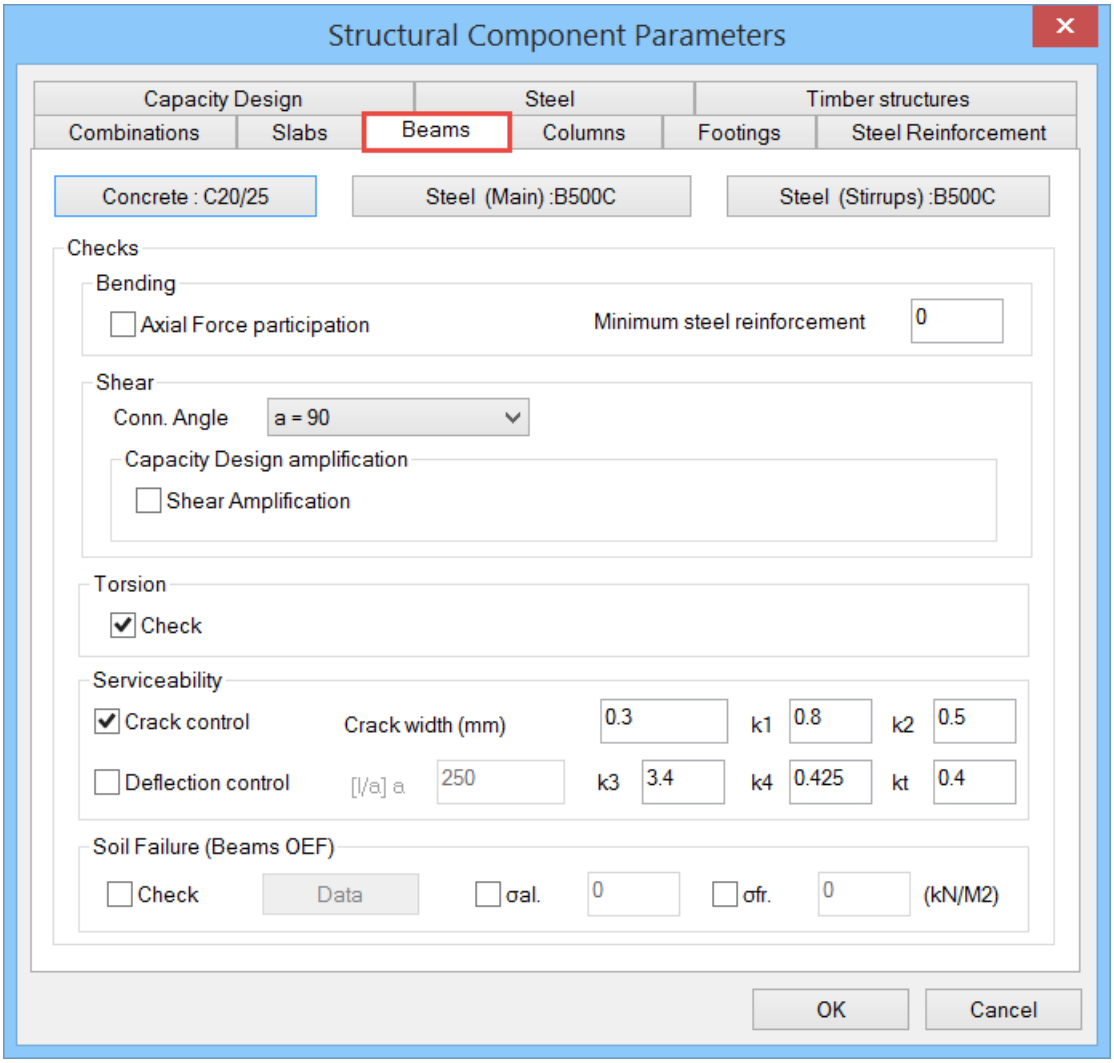
There are two default combinations, one for the Ultimate and one for the Serviceability Limit States.
To create a new combination, press, “Insert”. The new combination is the combination number 3 and the coefficients are all 0.



Change the coefficients and select the Limit State.
In the same way, you can define as many combinations as you wish, or modify the ones you have already created. The program will use the combination with the worst moment regarding the USL combinations and correspondingly will make deformations checks by the functionality combinations. By using “Delete” button you can delete the created combinations. The 1 & 2 default combinations cannot be deleted.

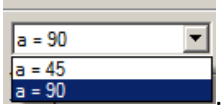
1.3.3 Beams

This field contains the design checks for the beams verifications.

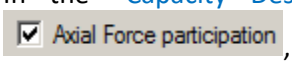


In the “Bending” field activate the checkbox ”Axial Force participation” for considering the axial force in the bending design checks.

In the “Shear” field define in the drop-down list “Conn. Angle” as the angle of the stirrups



.

In the “Capacity Design amplification” field keep activated the following checkbox 

, in case that it is necessary to perform capacity design (e.g. DCM and DCH categories, “Capacity Design” is necessary) according to §5.4.2.2, EC8.

IMPORTANT NOTE!!!

γ_{Rd} strength factor receives the value that corresponds to the ductility class automatically:
γ_{Rd}=1,0 for DCM
γ_{Rd}=1,2 for DCH

In the "Priority" field activate the checkbox "Check". Then, the program will consider the shear resistance V_{cd} = 0 and will calculate the stirrups.

In the "Serviceability" field activate the checks to be considered in the serviceability limit state (i.e. Crack and Deflection control) and modify the values in the "Crack width" and " k1", "k2", "k3", "k4", "kt" (§7.3.4, EC2) fields.

k1: is a coefficient which takes into account the bond properties of the bonded reinforcement:
k1=0.8 for high bond bars
k1=1.6 for bars with an effectively plain surface (e.g. prestressed tendons)
k2: is a coefficient which takes into account the distribution of the strains
k2=0.5 for bending
k2=1.0 for pure tension
k3=3.4
k4=0.425

The "Deflection control" In the beams according to EC2 is made by selecting the corresponding check in the parameters of the beam.

Define the upper limit (l/a) of the deformation.

The results appear at the end of the exploration file

DEFLECTION CONTROL

BEAM 1 5 b=0.25 h=0.50 c=0.03

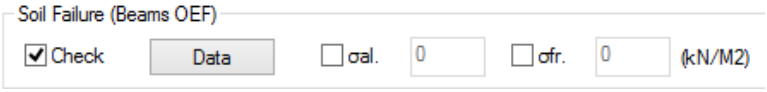
COMB	M	N	Du1	Du2		
100	-21.356	-0.000	0.00443	0.00011	0.00454	0.01500
101	-20.315	-0.000	0.00432	0.00012	0.00443	0.01500
102	-23.958	-0.000	0.00471	0.00010	0.00481	0.01500

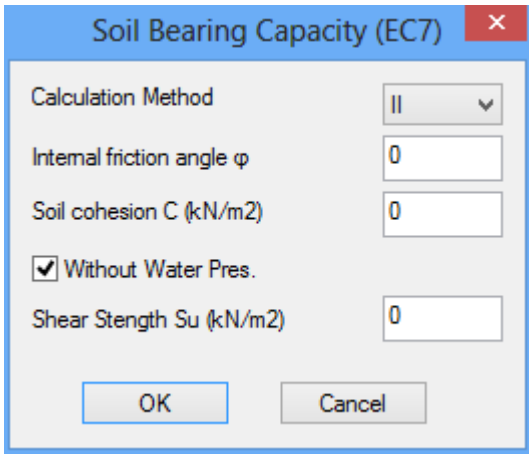
EXAMPLE:

In the above beam, a check considering the three combinations of functionality (100, 101, 102) is done and the size Du1 is the maximum deformation of the element, as it results from the calculation of the elastic cord.
Du2 is the deformation as calculated based on the EC2 relationship 7.18.
Then comes the sum (Du1+Du2) and the last column is the upper limit l/a.

It should be (Du1+Du2) < l/ a, so as the check to be achieved.

Soil Failure (Beams OEF): In this field, there are design checks related to beams on elastic foundation.





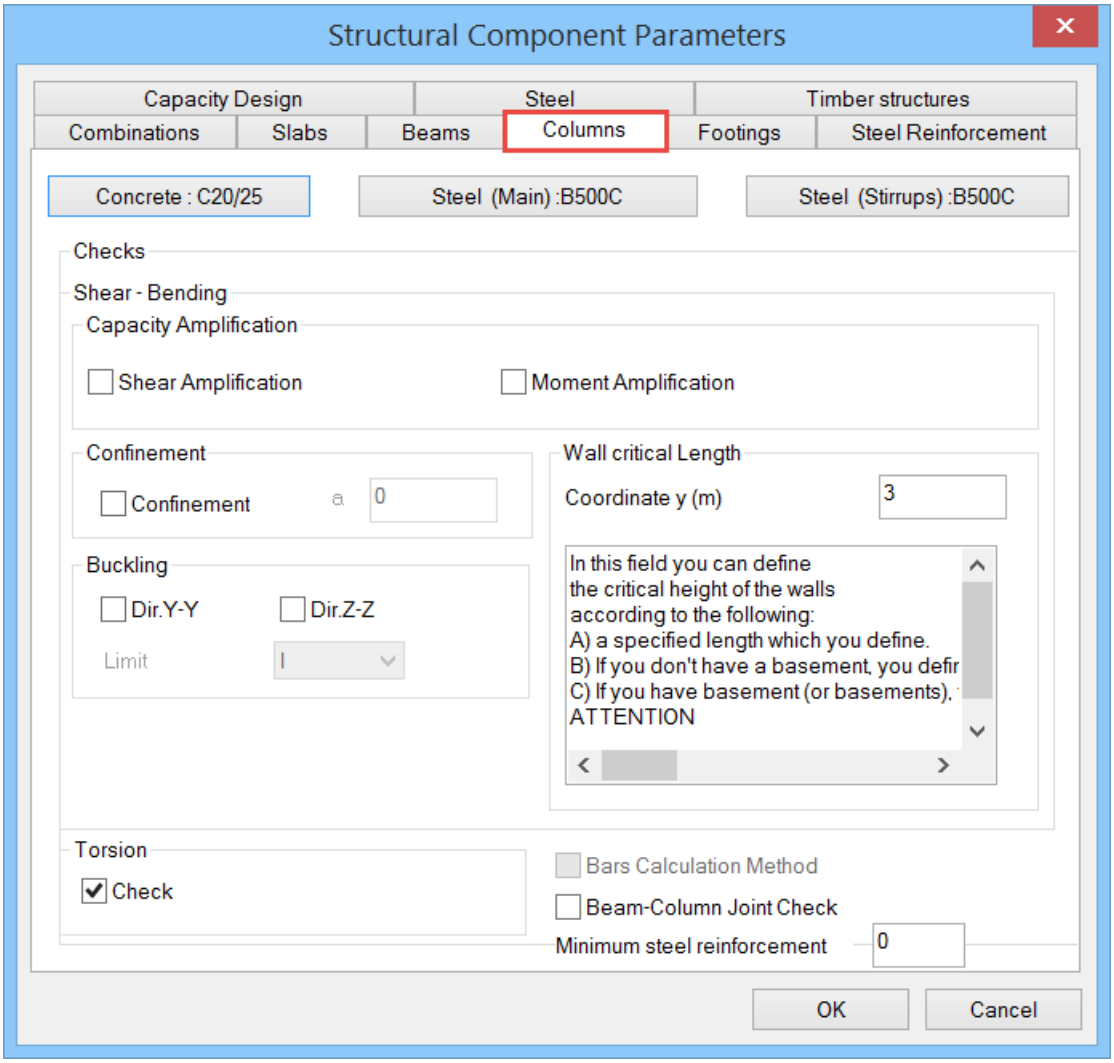
Activate the checkbox “Check” and click the button “Data”. A dialog box opens entitled “Soil Bearing Capacity (EC7)” where you enter the soil data, in case you have a geotechnical study. Select a calculation method from the drop-down list and define an internal friction angle ϕ, the soil cohesion and the shear strength Su. Then, the program will calculate the stresses.
Alternatively, activate the checkbox next to each stress and enter a value.
Soil Failure calculation for Beams OEF, according to the selected calculation method, is based on the EC7 provisions.

The parameters in the field “Soil Failure” regarding the soil resistance for beams OEF are calculated using EC7 methods. Otherwise, you can activate the checkbox near the relative stresses and type the value for the nominal stress σ_{al} (KN/m^2) and the stress fracture σ_{fr} (KN/m^2).

The soil failure check based on EC7 is shown in the results and the most unfavorable reason is displayed.
Analytical results of the checks for each combination are also shown in the Exploration file.

1.3.4 Columns

In this field, there are the design checks related to columns and shear walls verification.



In the field "Capacity Amplification" activate the appropriate checkbox, in case that the capacity design is necessary (e.g. According to EC8, in §4.4.2.3 and §5.4.2.2, for DCM and DCH, "Capacity Design" is necessary).

γ_{Rd} overstrength factor receives the related to the ductility class value automatically:
γ_{Rd}=1.1 for DCM
γ_{Rd}=1.3 for DCH

In the field "Wall Critical Length" type a value according to the cases in the list below.
A) Type a specific value, regardless of the Design Regulation considered in the calculations.
B) If there is no basement, indicate the height H of the first level. The program compares this value with the parameter l_w=H/6 and keeps the greater one.
Γ) If there is a basement, define the height H of the level above the basement. The program compares this value to the l_w and H/6 and keeps the greater one.
The definition of a value in this field is mandatory for the calculation of the walls critical length.
Activate the checkbox "Confinement" and type a value in the "a" field. Otherwise, keep the checkbox inactive and the program will calculate the stirrups section and the distribution according to the paragraphs §5.4.3.2.2 and §5.4.3.4.2 of EC8*[1].

*1

For Columns (*§ 5.4.3.2.2, EC8)*

$$\alpha\omega_{wd} \geq 30\mu_{\varphi}\nu_{d} \cdot \varepsilon_{sy,d} \cdot \frac{b_c}{b_o} - 0.035 \qquad (5.15)$$

Where

ω_{wd} is the mechanical volumetric ratio of confining hoops within the critical regions; $\omega_{wd} = \frac{volume\ of\ confining\ hoops}{volume\ of\ concrete\ core} \cdot \frac{f_{yd}}{f_{cd}}$

μ_{ϕ} is the required value of the curvature ductility factor;

ν_d is the normalized design axial force ($\nu_d = N_{Ed} / A_c \cdot f_{cd}$);

$\varepsilon_{sy,d}$ is the design value of tension steel strain at yield

For Walls (*§ 5.4.3.4.2, EC8)*

For walls of rectangular cross-section, the mechanical volumetric ratio of the required confining reinforcement ω_{wd} in boundary elements should satisfy the following expression, with the values of μ_{φ} as specified in (2) of this sub clause:

$$\alpha\omega_{wd} \geq 30 \cdot \mu_{\varphi} \cdot (\nu_d + \omega_v) \cdot \varepsilon_{sy,d} \cdot \frac{b_c}{b_o} - 0.035 \qquad (5.20)$$

Where the parameters are defined in 5.4.3.2.2(8), except ω_v, which is the mechanical ratio of vertical web reinforcement ($\omega_v = \rho_v \cdot f_{yd,v} / f_{cd}$).

In the field "Buckling" activate the checkbox  referred to the Y or Z direction (along with the local axis Y or/and Z).
(NOTE: View of local axes: Menu>>"View">>"Switches">>"Local Axis")

In the field "Short Columns" activate the checkbox "Check" to perform the required check in DCH cases *2.

*2

Beam-column joints *(§5.5.2.3)*

(1)P The horizontal shear acting around the core of a joint between primary seismic beams and columns shall be determined to take into account the most adverse conditions under seismic loading, i.e. capacity design conditions for the beams framing into the joint and the lowest compatible values of shear forces in the framing elements.

(2) Simplified expressions for the horizontal shear force acting on the concrete core of the joints may be used as follows:

a) for interior beam – column joints

$$V_{jhd} = \gamma_{Rd} \cdot (A_{s1} + A_{s2}) \cdot f_{yd} - V_C \qquad (5.22)$$

b) for exterior beam – column joints:

$$V_{jhd} = \gamma_{Rd} \cdot A_{s1} \cdot f_{yd} - V_C \qquad (5.23)$$

Where

A_{s1} *is the area of the beam top reinforcement;*

A_{s2} *is the area of the beam bottom reinforcement;*

V_C is the column shear force, from the analysis in the seismic design situation;
γ_{Rd} is a factor to account for overstrength due to steel strain-hardening and should be not less than 1.2.

(3) The shear forces acting on the joints shall correspond to the most adverse direction of the seismic action influencing the values As1, As2, and Vc to be used in expressions (5.22) and (5.23).

***Beam-column joints** (§5.5.3.3)*

(1)P The diagonal compression included in the joint by the diagonal strut mechanism shall not exceed the compressive strength of concrete in the presence of transverse tensile strains.

(2) In the absence of a more precise model, the requirement of (1)P of this subclause may be satisfied using the subsequent rules.

a) At interior beam – column joints the following expression should be satisfied:

$$V_{jhd} \le \eta \cdot f_{cd} \cdot \sqrt{1-\frac{\nu_d}{\eta}} \cdot b_j \cdot h_c \qquad (5.33)$$

Where
$\eta=0.6\cdot(1-f_{ck}/250)$;
ν_d is the normalized axial force in the column above the joint; and
f_{ck} is given in MPa

b) At exterior beam – column joints:

V_{jhd} should be less than 80% of the value given by the right-hand-side of expression (5.33) where:

V_{jhd} is given by expressions (5.22) and (5.23) respectively;

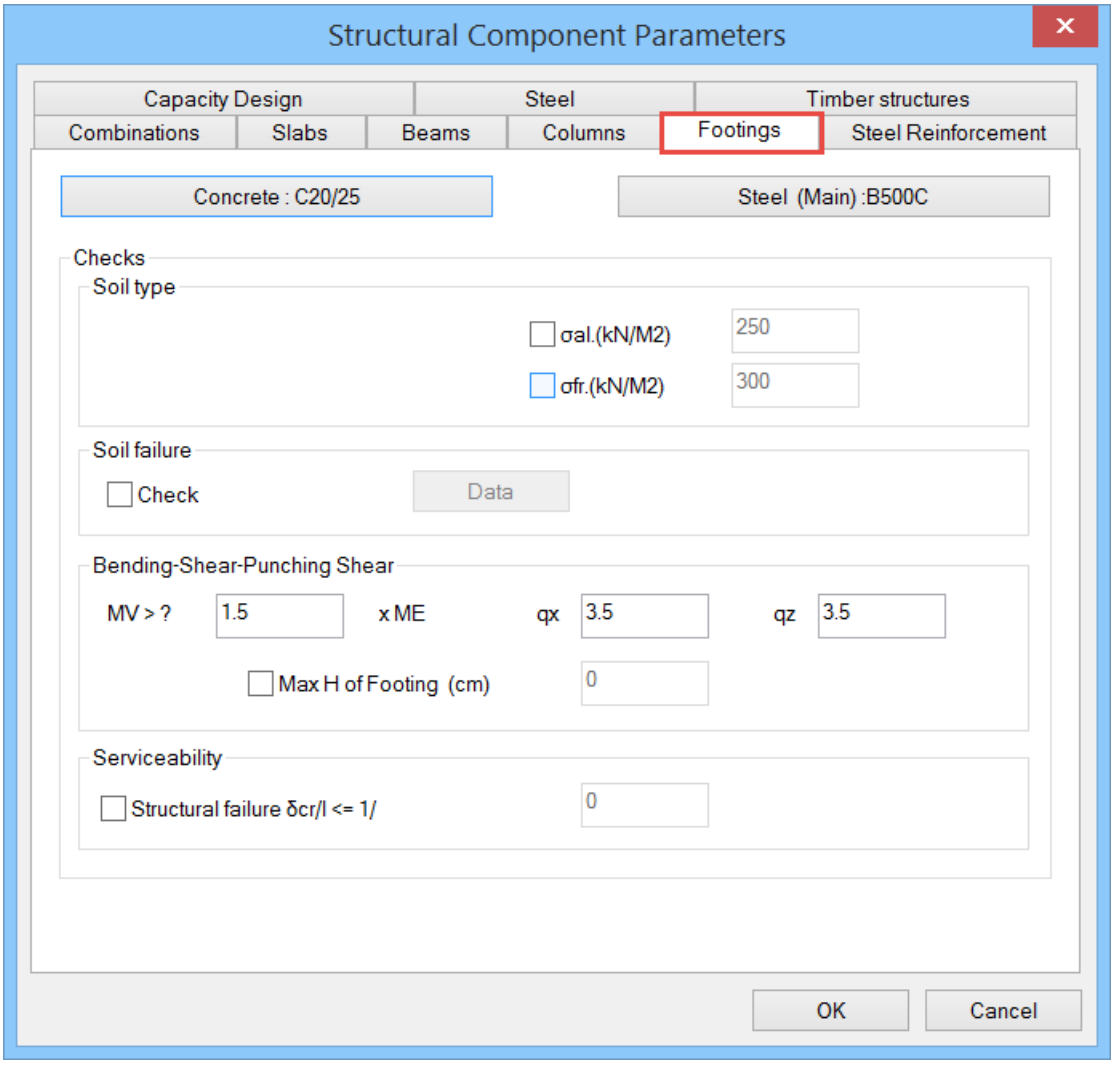
And the effective joint width b_j is:

a) if $b_c > b_w$ then $b_c > b_w : b_j = \min\{b_c; (b_w + 0.5\cdot h_c)\}$ *(5.34a)*

b) if $b_c < b_w$ then $b_c > b_w : b_j = \min\{b_w; (b_c + 0.5\cdot h_c)\}$ *(5.34b)*

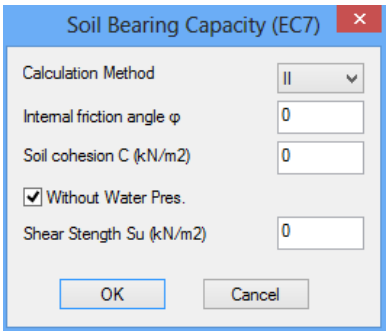
1.3.5 Footings

In this field, the relative checks for the footings verification are located



In the "Soil Type" field you can activate the checkbox next to the relative stresses and type the value for the nominal stress σ_{al} (KN/m2) and stress fracture σ_{fr} (KN/m2).

In the "Soil failure" field activate the checkbox "Check" and then press the button "Data", to enter the soil data, in case you have a geotechnical study. Select a calculation method and define an internal fiction angle ϕ, the soil cohesion and the shear strength Su. The program will calculate the stresses. Otherwise, activate the checkboxes next to the stresses and enter a value.
Soil Failure calculation for Beams OEF, according to the selected calculation method, is based on the EC7 provisions.



"Bending-Shear-Punching shear": In this field activate the checkbox "Max H of Footing" and type a value. The activated checkbox means that the program will perform the design check against punching shear. If the original height does not satisfy the punching shear design check, the program will calculate the height that satisfies the check. If this is higher than the limit you have set, a message is displayed that informs you that a higher footing is necessary.

In the following fields "qx" and "qz" 

type the values of the coefficients used in the analysis.

In the "Serviceability" field the design checks that correspond to the serviceability limit state are included. Activate the checkbox which allows the user to define the limit value of the considered ratio δ_{cr}/l.

1.3.6 Steel Reinforcement

On the first field "Available Bars", which is common for all structural elements, specify the diameters of the reinforcement bars.

From the diameters list 

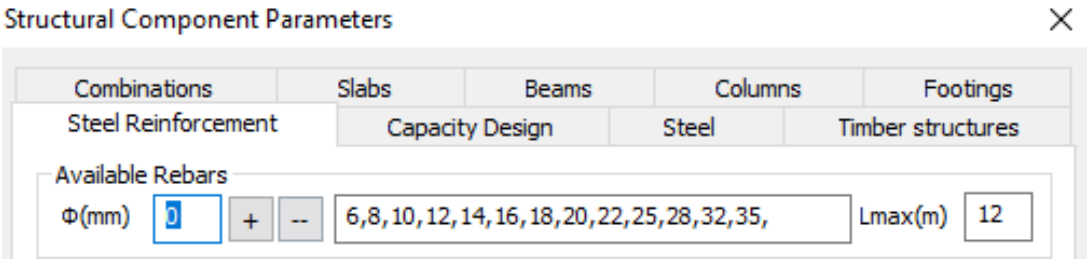
add or remove a new or an existing diameter, respectively. Type a value in the following field 

and press the button 

to add the new diameter to the list. To remove an existing diameter, select the value from the list and press the following button 

.

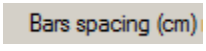
In the Lmax field, you type the maximum bar length to be used in the reinforcement of the structural elements.



1.3.6.1 Slabs

In "**Slabs**" the rebar of the slabs is defined:

In the field "Bar Cover" type the rebar cover value according to the climatic conditions (mm).

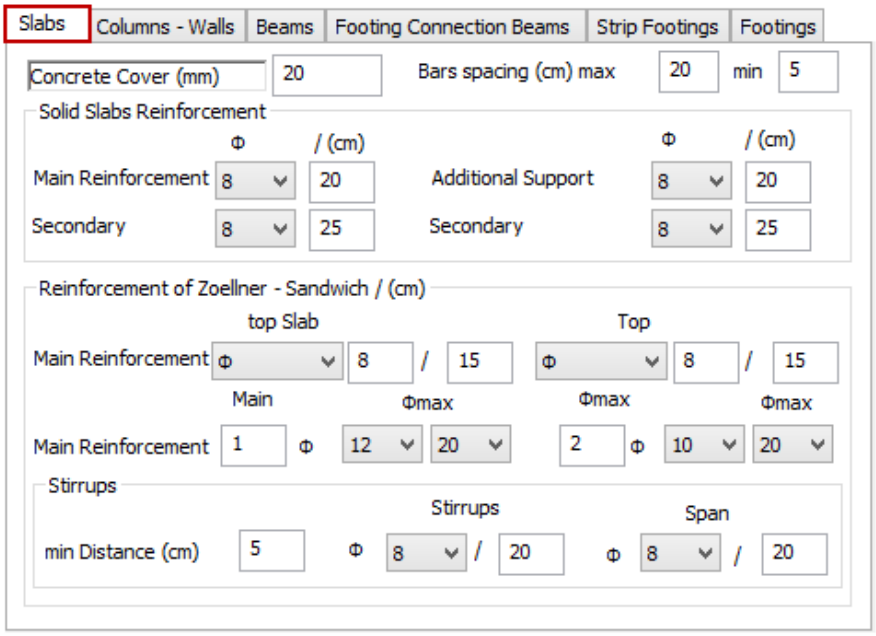
In the "Bar spacing" field 

type the maximum and minimum distance (cm) between the reinforcement bars inside the slab.

In the field "Solid Slabs Reinforcement" specify the minimum reinforcement (main, secondary, additional) and the minimum relative distances.

In the "Reinforcement of Zoellner-Sandwich" field specify the upper and lower, minimum main reinforcement for the solid part. For the beams inside the slab, define the bars' quantity and the upper and lower maximum and minimum diameter.

In the field “Stirrups” specify the minimum distance between the stirrups (cm), diameter/distance in support and span.

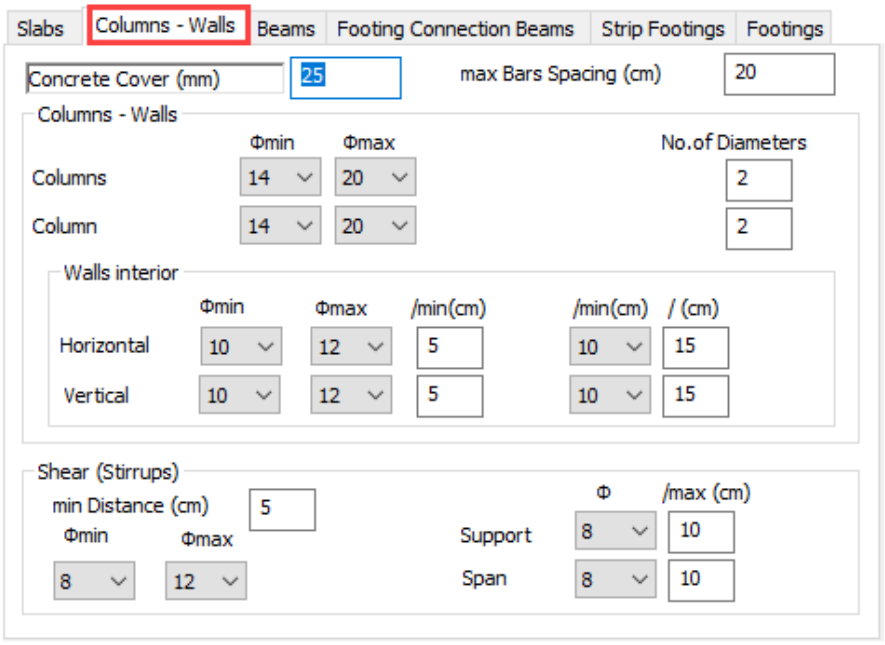


1.3.6.2 Columns-Walls

In “**Columns-Walls**” the reinforcement of the columns-walls is defined:

Type the bar cover value according to the climatic conditions (mm).

In Bars spacing (cm) type the maximum and minimum limits related to the distance (cm) between the bars.



In the “Columns-Walls” field specify the maximum and minimum limits of the diameter of the main steel reinforcement used for columns (“Columns” field) and columns inside the shear walls (“Column” field).
In the field “No. of Diameters” define how many differed diameters could be used in columns and shear walls, respectively.

EXAMPLE:
If you type 2, the program will consider two more diameters, i.e. 3 in total (i.e. Φ16-Φ18-Φ20). If you type 0, it will consider only one.

In the “Walls interior” field define the parameters of the steel reinforcement for the body of the shear walls. Type the maximum and minimum diameter for horizontal and vertical rebar and the minimum distance between them. For distances less than the minimum defined, the program will increase the diameter.

In the field “Shear (Stirrups)” specify maximum and minimum limits for the stirrups’ diameter and the minimum distance between them (for less than then minimum distance, the program increases the diameter).

In the following fields 

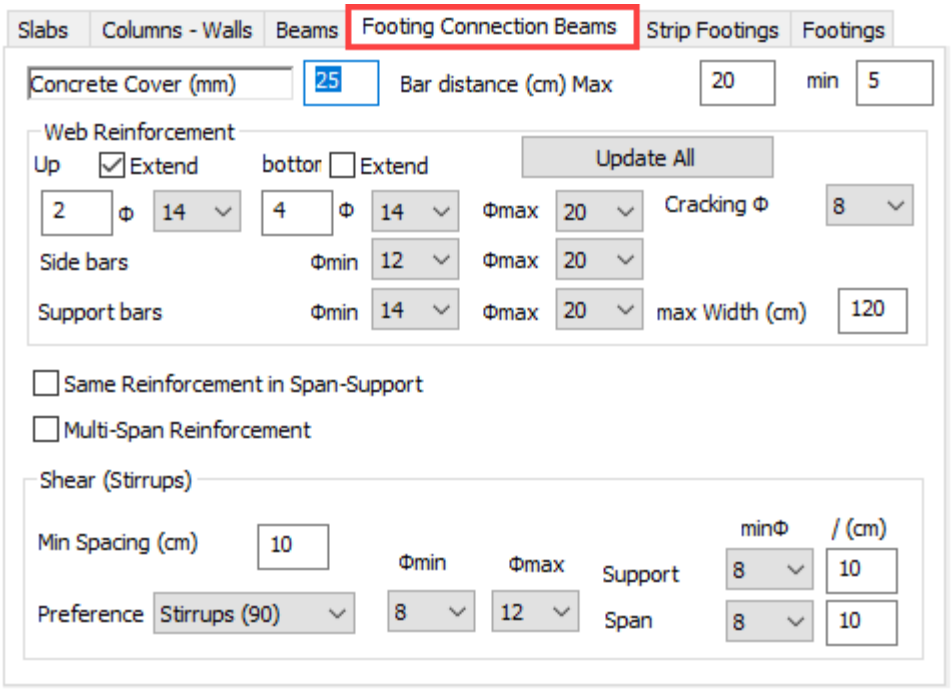
specify the steel reinforcement in the support and the span. If the defined values satisfy the design checks, they will be placed, otherwise, the program will take into account the limit values so as different steel reinforcement is placed.

1.3.6.3 Beams/Footing Beams

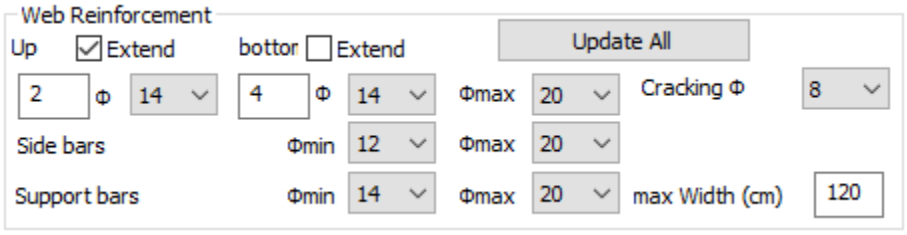
In “**Beams**” and “**Footing Beams**” define reinforcement:

Type bar cover value according to the climatic conditions (mm).

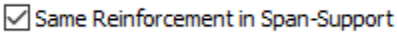
In Bar distance (cm) type the maximum and minimum limits for distance (cm) between the bars.



In the field “Web Reinforcement”, the checkboxes “Up” and “Bottom” next to “Extend” means that the beam supports’ steel reinforcement will pass through the span and will be added on this. Specify the maximum and minimum limits for the diameter of the main reinforcement in extending (upper and lower), for bars in the span and the support.



First, specify the minimum main reinforcement of the upper side. Type the number and select the diameter. Repeat for the lower side. Then define the maximum diameter.
Then specify the minimum and maximum diameter for the main reinforcement on side and supports. Finally, in the “Cracking Φ” field specify the minimum diameter of the reinforcement considered for crack control.
In the “max Width” field, determine the maximum distance to place a common bar. In case of bigger distance, the program will place two bars in the support.

Activate the following checkbox 

and the steel reinforcement will be the same for span and support.

IMPORTANT NOTE:

MULTI-SPAN REINFORCEMENT

Activate the following checkbox ☑ Multi-Span Reinforcement so that the reinforcement in the spans is common to the entire beam.

In the implementation process of single beams reinforcement with common rebars of a specific length, the program places the additional rebars in the supports based on certain criteria. There are two ways to place the additional supports reinforcement.

- The first way is additional rebars to come from each span on both sides and be positioned on the respective side of the span.
- The second way is to place a common support rebar.

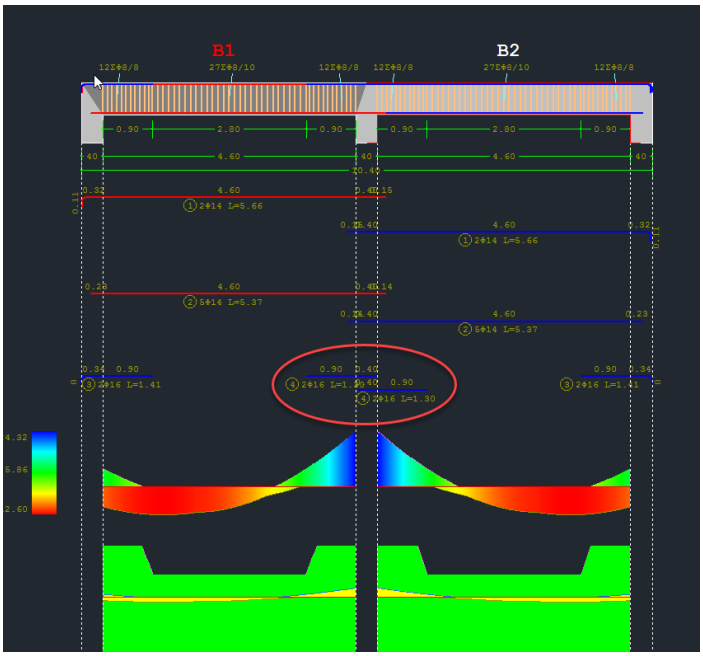


Figure 1

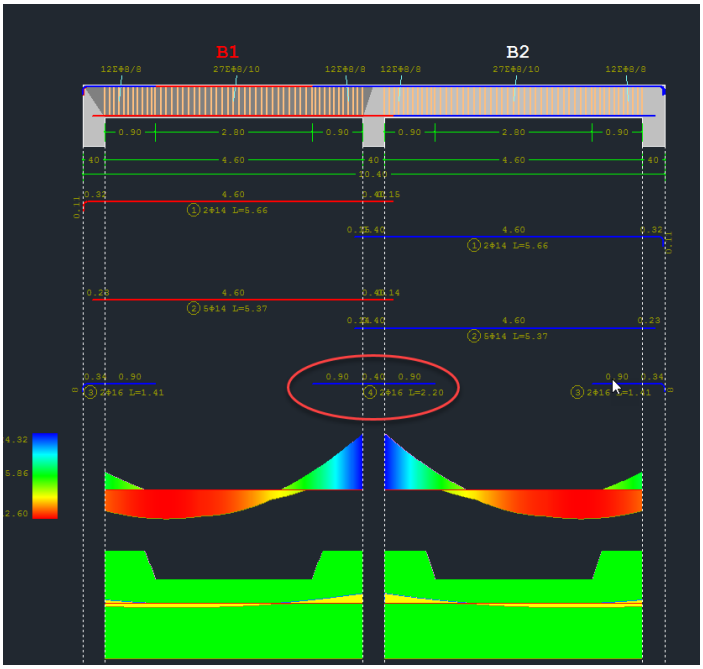
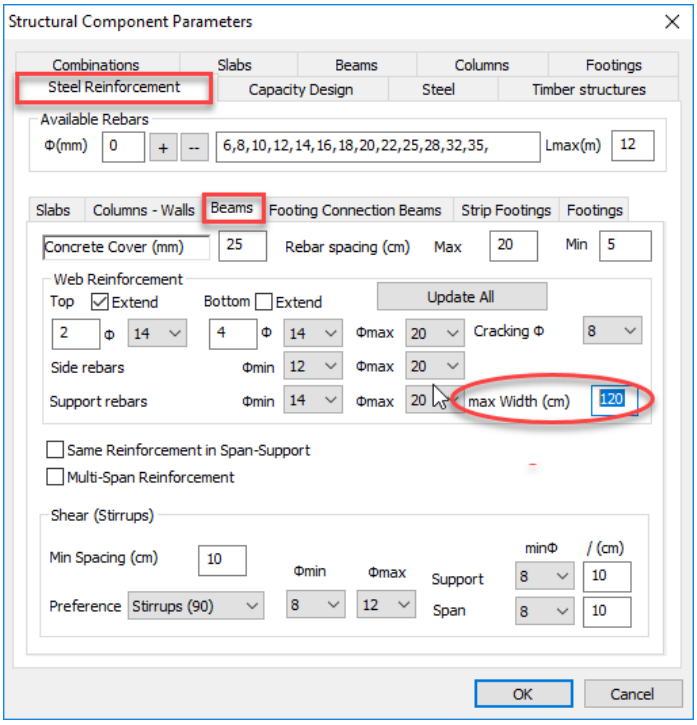


Figure 2

A. The first criterion is the width of the support, as determined by the parameters of beams reinforcement.



If it exceeds the max support width (see Figure 1), then support reinforcement per side is placed separately.
If the width of the support is less than max width, then the same rebar is placed all along the support (see Figure 2).

NOTE:

Changing this parameter post creation of the alignments of the beams requires deletion and re-creation.

B. The second criterion has to do with the width of the beams in the support. If this width is different for the two beams, then additional support rebars are placed separately. Otherwise, a common rebar is placed.

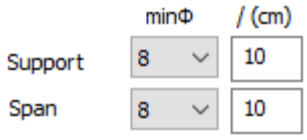
CONCLUSION:

Common rebar is placed only if both criteria mentioned above are satisfied:

A. Support width <max width in parameters
B. Same beams width

In the next section "Shear (Stirrups)" the parameters for the shear reinforcement of the beams are defined.

Specify,
the minimum distance between the stirrups
the angle (if they are placed perpendicularly or obliquely (45°))
the diameter limit values



In the following fields specify the diameter and the distance of the reinforcement bars in the support and the span. If these values satisfy the design checks, the rebar is placed as appropriate, otherwise, the program places a different reinforcement by taking into account the limit values of the diameter as well as the distance.

NOTE:

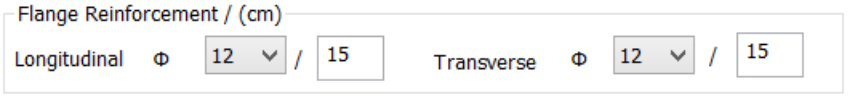
Update All: After completing the parameter selections and before closing the window, press the Update All button to update the parameters.

1.3.6.4 Strip Footing

In "**Strip Footing**" the parameters of the steel reinforcement in strip footings are defined:

For the common parameters mentioned in the previous tabs, follow the previous described procedure.

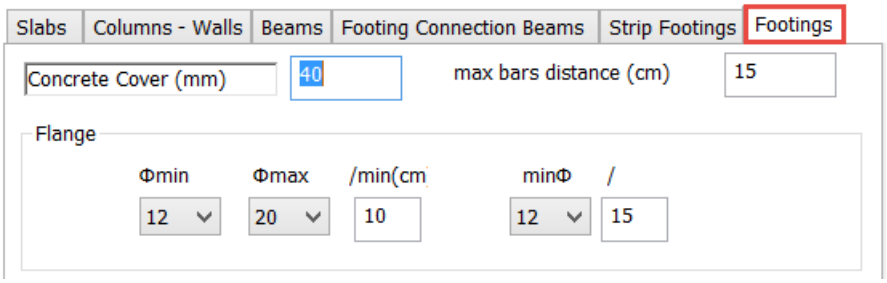
Also, in the "Range Reinforcement" field define the limits for longitudinal and transversal flange reinforcement.



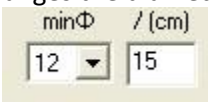
1.3.6.5 Footings

In the "Footings" tab, the parameters of the steel reinforcement in footings are defined:

Type the bar cover value by the climatic conditions (mm).
In the field "max bar distance" type the maximum and minimum limit values for the distance (cm) between the reinforcement bars.



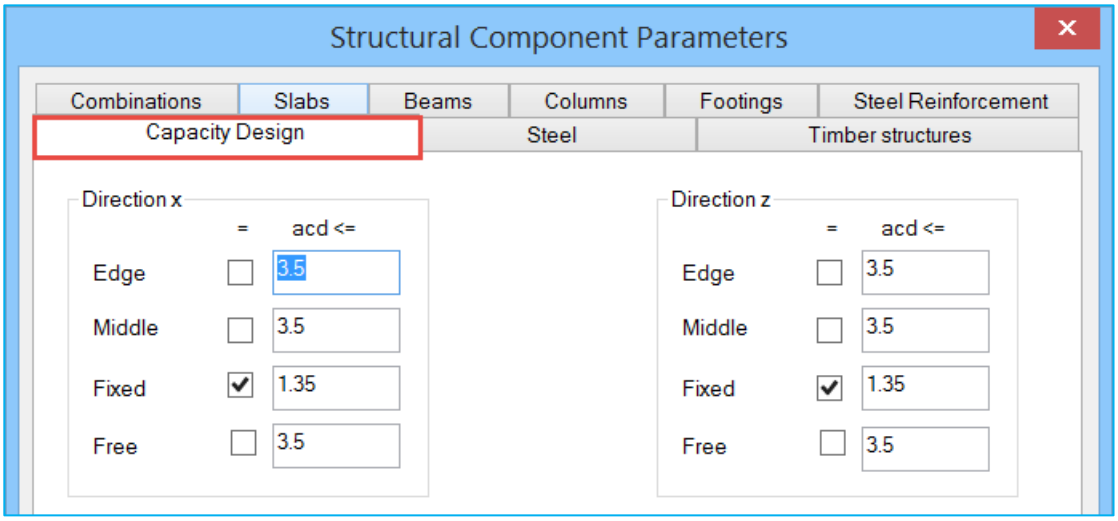
Specify the minimum and maximum diameter of the bars and the minimum distance. For lower distance, the program changes the diameter.

In the following fields 

determine the diameter and the distance of the reinforcing bars, you wish to place. If it is sufficient, then they are placed. Otherwise, the reinforcement, resulting from the program's design verification, is placed.

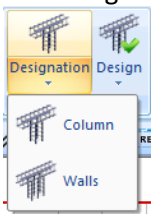
1.3.7 Capacity Design

The last tab "Capacity Design" concerns concrete structures capacity design checks:



Specify the upper bound of the factor “a_{cd}” for capacity design, in each direction.

Generally, the value of α_{cd} should be less or equal to the seismic behavior factor q.
For fixed columns, α_{cd} = 1.35 is taken.
Activate the corresponding checkbox and type the value you want.
Without ticks, the program takes into consideration the calculated values.
The designation of a node as free or fixed is performed by using the command "Designation".

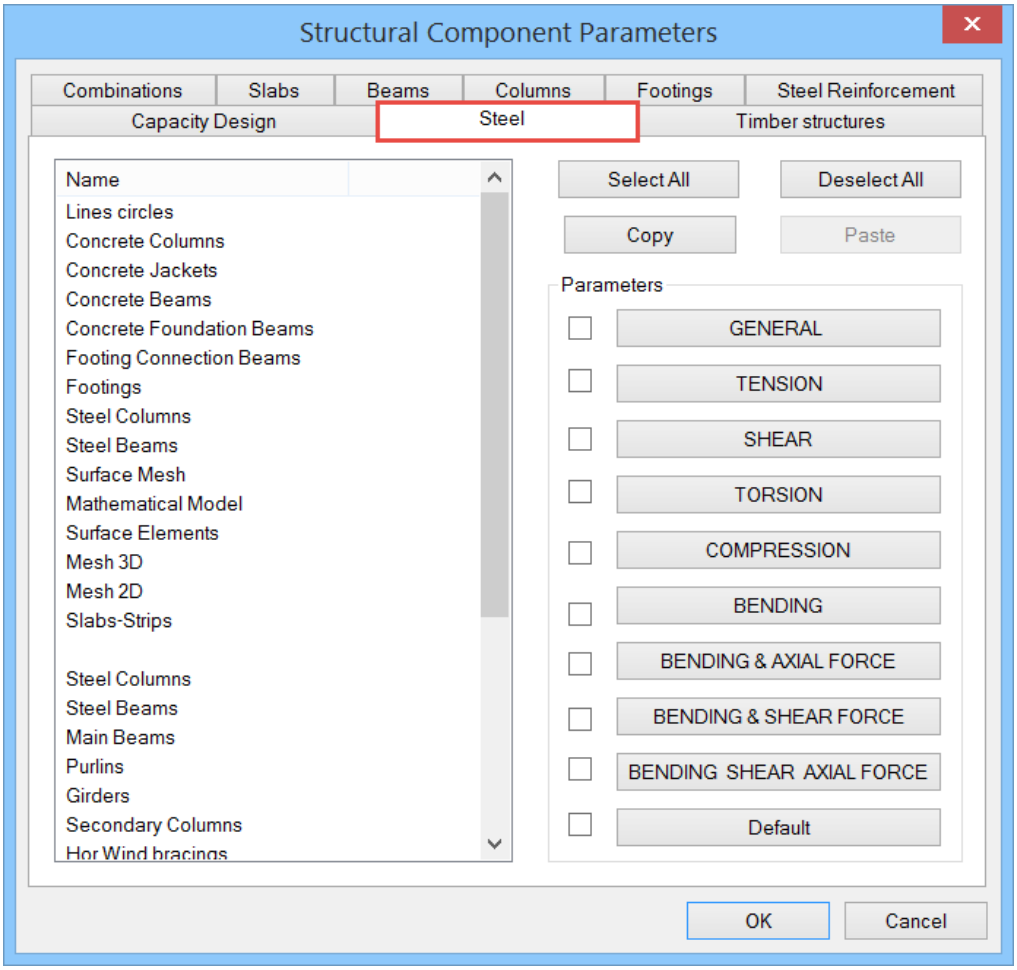


⚠ Nodes without Designation are considered free in both directions except for the fixed ones.

⚠ Finally, considering the exploration of the column, there are the results of the capacity design check (initial and incremented moments) for each combination. Of course, in the direction in which the capacity design check has not been taken into account, the initial moments are the same as the incremented ones.

1.3.8 Steel

The next section regards the **“Steel cross sections”** check parameters and displays the following dialog box:

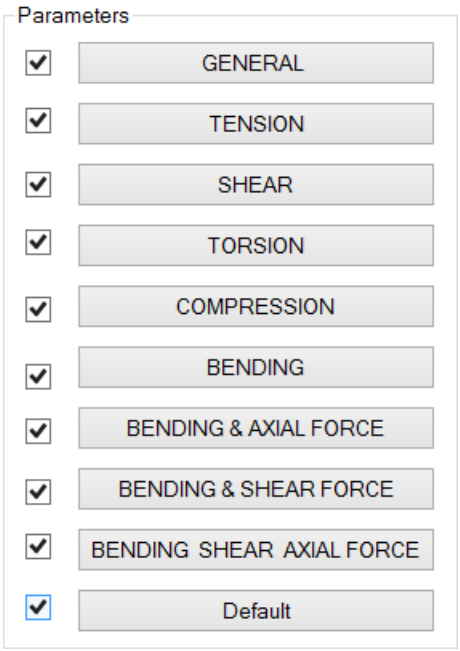


First, select a layer. Click one from the list, or more using “ctrl”, or all using “Select All”.
(By pressing the button “Deselect All” cancel the previous layers’ selection.)
Then activate one or more design checks by clicking on the corresponding checkbox and press the corresponding button to specify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

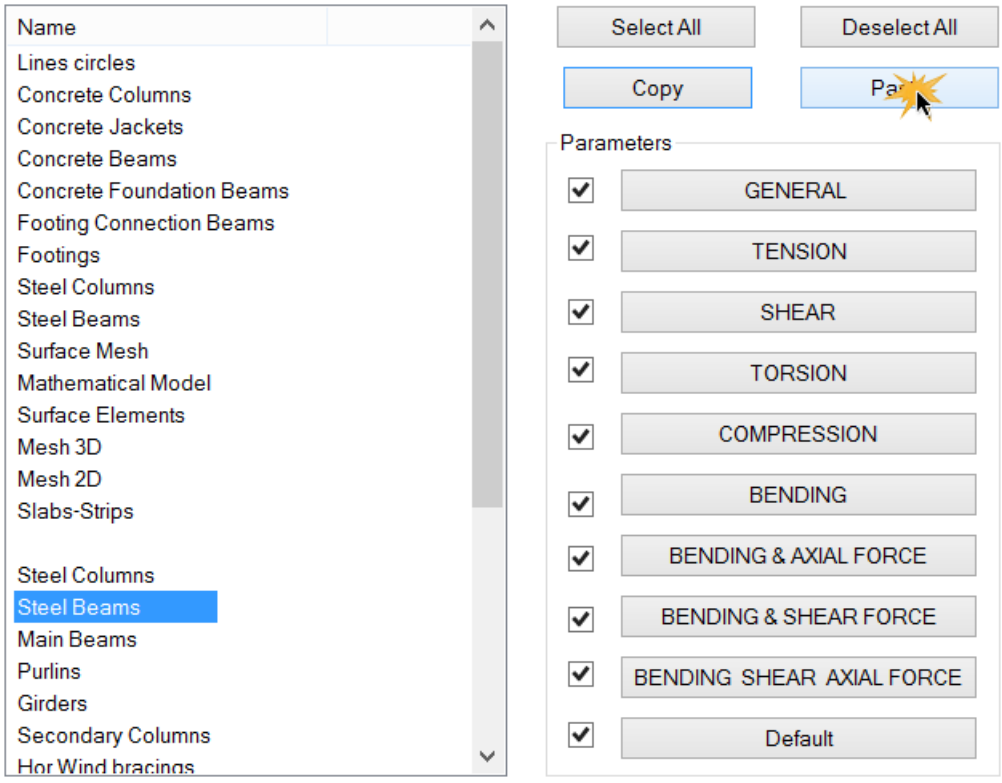
The parameters defined for one layer can be copied to other layers, using the command "Copy".
Select a layer → define the parameters → press “Copy” → select another layer → press “Paste”.

EXAMPLE:

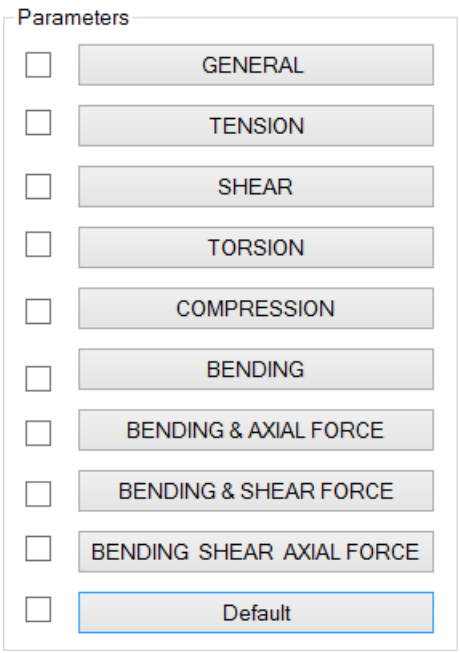
Suppose you have set all parameters for the layer Steel Columns and you want to pass these parameters to Steel Beams. Activate the check box next to "Default" and all parameters are selected automatically.

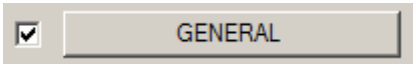


Then press “Copy”, select layer Steel Beams and press “Paste” (now activated).

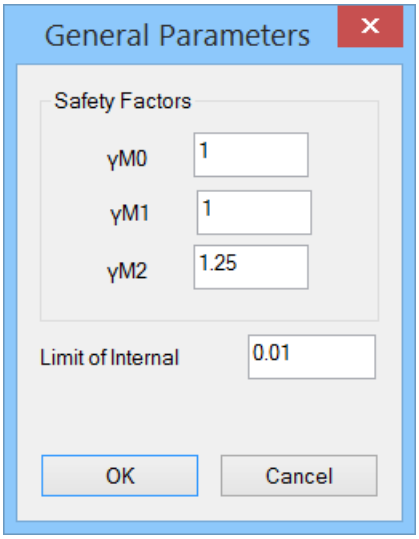


Now all the parameters defined for Steel Columns are defined also for the layer Steel Beams. An alternative method to set the same parameters to all layer including steel sections is to select all layers by pressing "Select all" button and set the parameters once for each check category. Note that at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



Press the button “GENERAL” 

to set the γ_{Mi} safety factors:



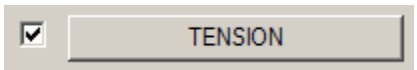
γ_{M0}: partial factor for cross-sections’ resistance whatever the class is

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

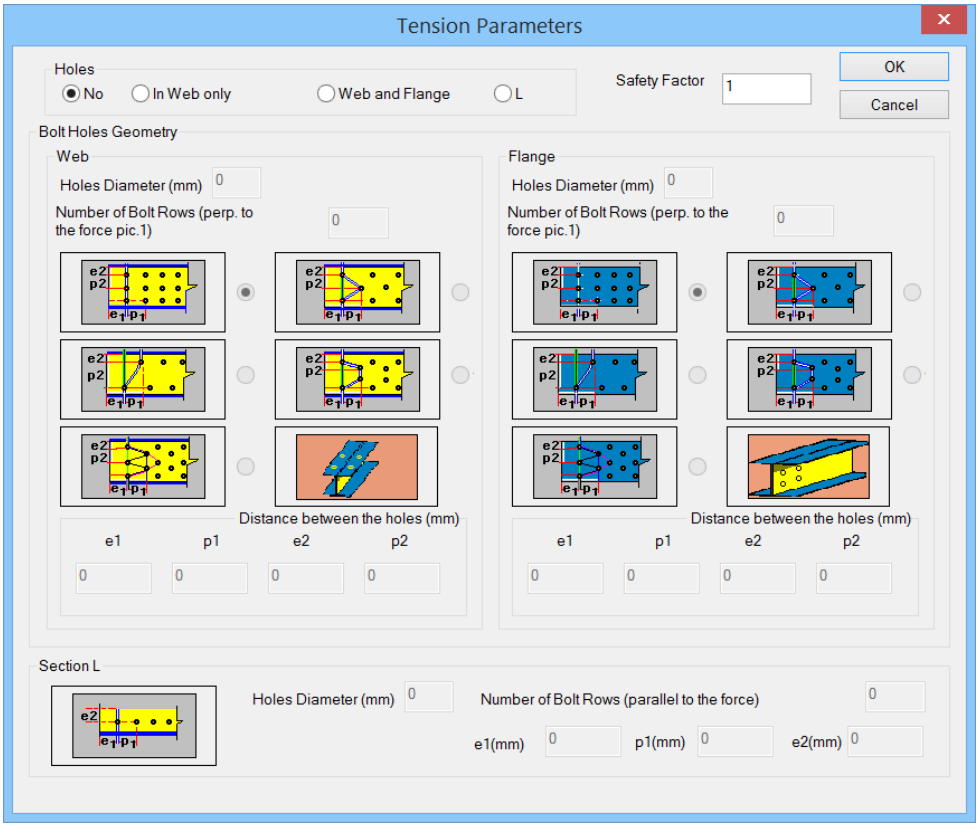
γ_{M2}: partial factor for resistance of cross-sections in tension to fracture

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” 

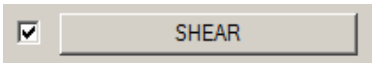
to define the parameters that correspond to the shear design check as well as the position of the hole check (EC3 §1.8 §3.5):



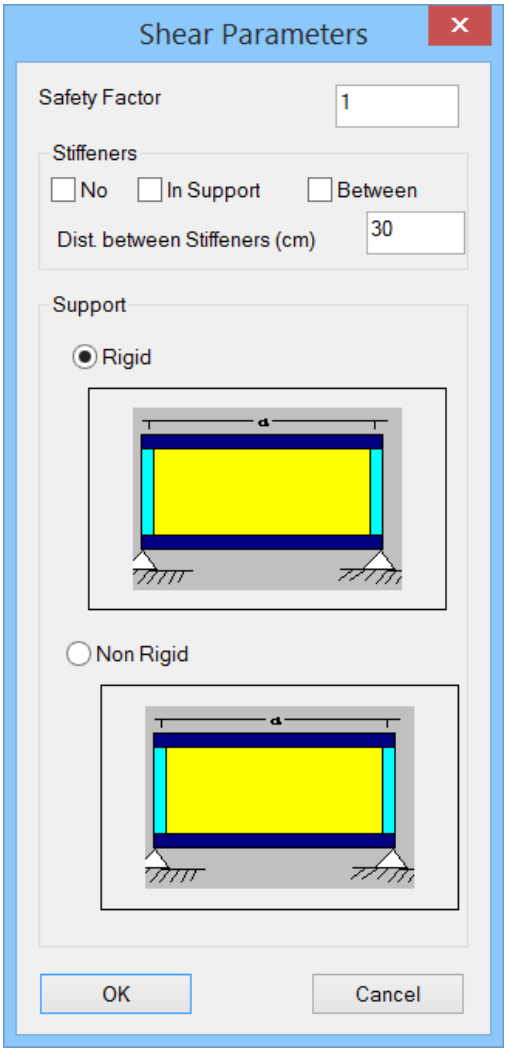
Specify the spacing of the centers of two consecutive holes, the holes diameter and the number of bolt rows.
In case of L section specify the parameters on the bottom of the dialog box in the field “Section L”.

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.

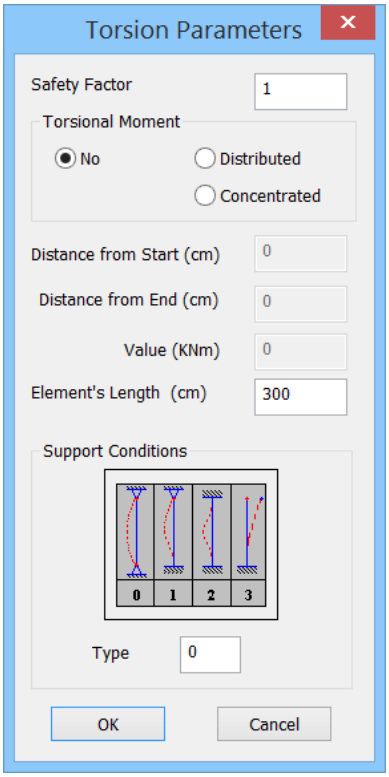
The safety factor for all design checks is fixed and equal to one, which means that the program calculates the ratio of the stress resultant versus the resistance. A value of the calculated ratio greater than 1.0 indicates failure.

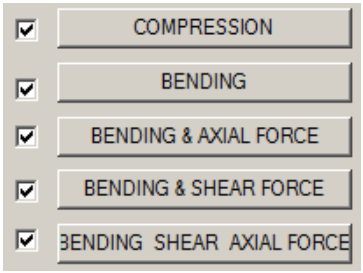
Press the button “SHEAR” 

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).

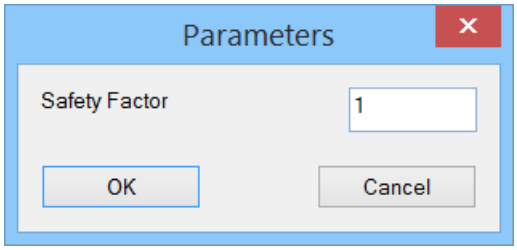


Click the button “TORSION”  to define whether the structural elements of the selected layer are loaded by a distributed or concentrated torsional moment, or not. In the following dialog box, you may as well define the support conditions based on the corresponding figures. Select the type of moment and 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.



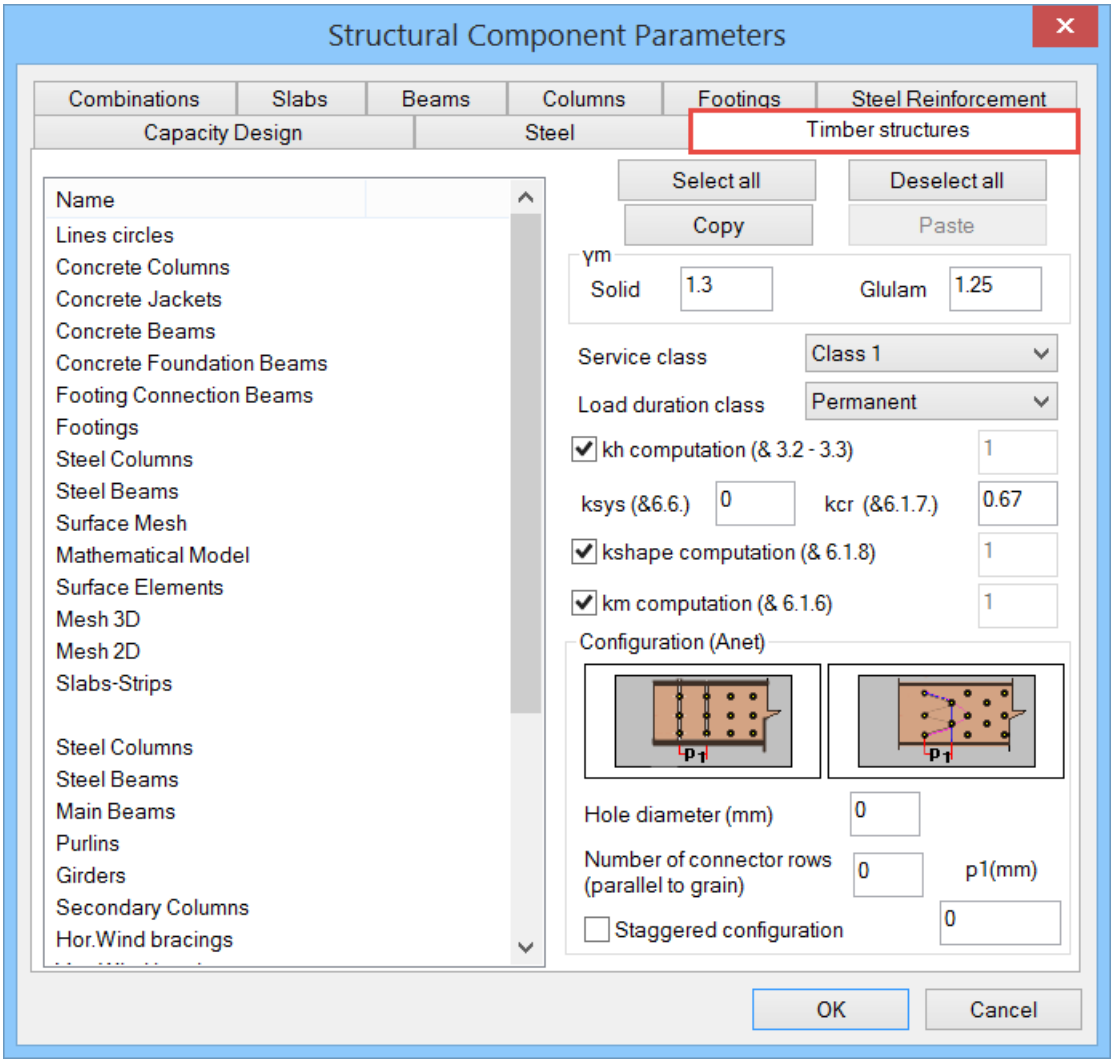


For all design checks presented in the figure on the left, define the safety factor in the dialog box that appears when you click one of the five buttons. The safety factor is the ratio of the resistance value versus the corresponding design value, which is set 1.0 by default.



1.3.9 Timber structures

The next section regards the **Timber cross sections** check parameters and displays the following dialog box:



The definition of design parameters for timber sections is made per layer. Select the layer for setting the parameters (ex. Timber Columns). Then the program enables you to Copy And Paste to other layers.

EXAMPLE:

Suppose you set all the parameters for the Timber Columns layer and you want to copy them to the Timber Beams layer.

Left click on Timber Columns layer in the list and press “Copy”, then select Timber Beams layer and press “Paste” which is now activated.

Those parameters are explained in detail below :

Partial Factors γM

Partial factors' value γ_M for the materials will be used in cases of ordinary or accidental loads (the accidental loads do not include the case of an earthquake, on which the paragraph 2.9.4) is applied.

Suggested values (EC5 – Table 2.3) are presented in the table below:

Recommended partial factors	**γ_M**	
Solid timber	1.3	Members
Glued laminated timber – Glulam	1.25	

Service classes – influence of moisture content

Humidity is associated with the environmental conditions on the structure (or the single member), i.e. the temperature and relative humidity. The regulation defines three service classes (EC5 – 2.3.1.3):

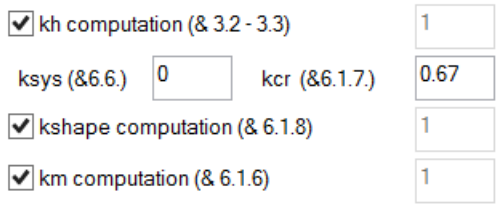
Service Classes	**Temperature and relative humidity**	**%**	**Examples**
1	temperature of 20°C and the relative humidity of the surrounding air only exceeding 65 % for a few weeks per year	(9±3) %	Closed constructions heated (hot roofs, interior floors, and interior walls)
2	temperature of 20°C and the relative humidity of the surrounding air only exceeding 85 % for a few weeks per year	(12±3) % (15±3) %	Closed construction unheated or heated periodically (ex. holiday homes) Open roofed construction, cold roofs, exterior walls and general structures that are not directly exposed to the elements
3	climatic conditions leading to higher moisture contents than in service class 2	> 19 %	Construction in damp areas or structures exposed to the elements (ex. direct wetting)

Load Duration Class

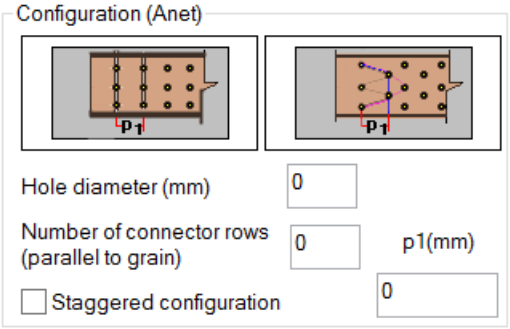
Since the strength of a member decreases during loading, the regulation divides load types into categories according to their duration (EC5 – 2.3.1.2):

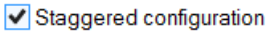
Load-duration class	Period of time	Examples of loading
Permanent	> 10 years	self weight
Long-term	Six months to 10 years	storage
Medium-term	One week to 6 months	imposed floor load
Short-term	to 1 week	snow, wind
Instantaneous		wind and accidental load

Kh, Kshape, Km factors are calculated automatically by the program by the respective chapters of EC5. The values can be defined by deactivating the checkbox and typing the respective value.



- **K_h** is a growth factor and depends on the type of wood, the size of the member and the type of load.
- K_{sys} is a growth factor that refers to continuous loads of sectional systems.
- K_{cr} is a reduction factor with a fixed value of 0.67 and concerns the shear
- **K_{shape}** is a factor depending on the shape of the cross section
- **K_m** is a factor considering redistribution of bending stresses in a cross-section



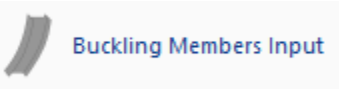
In "Configuration" field set an initial approximate order of the holes in the wooden member which will be used for members' predesigning and then it will be applied to the connections' design.
You set the diameter of the holes and the number of connector rows and the distance p1 according to the two figures.
To set a 

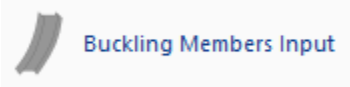
activate the corresponding Checkbox.

1.4 Merge Elements

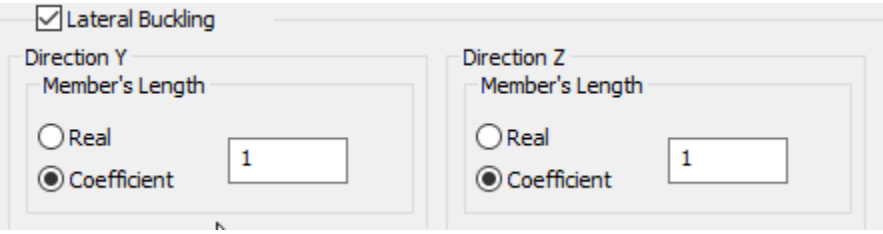
In the new version of the program, a new command group is added, which concerns merging of steel (and timber) members for the calculation as well as buckling and deformation checks display according to EC3.

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 

)



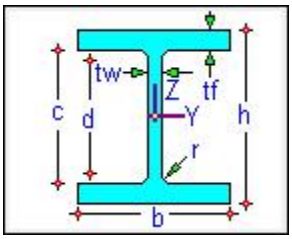
- Now, be using merging per direction, there is no need for the coefficient process, and merging will be achieved, in most cases, automatically.
- Also, note that through the merging process, the buckling length, is calculated correctly, and in the printouts of the results a merged element is printed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

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

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

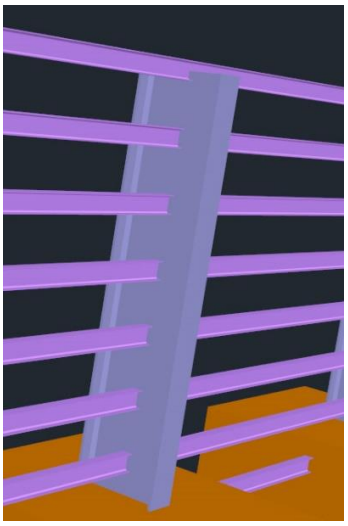
- **y-y** is the **major**, and
- **z-z** is the **minor**,

as in the figure below:

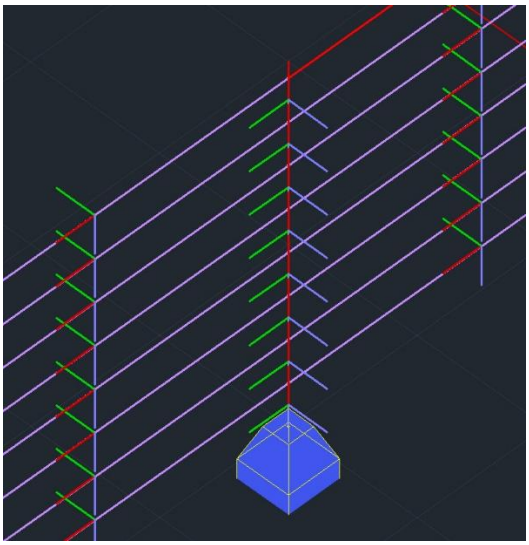


EXAMPLE:

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.

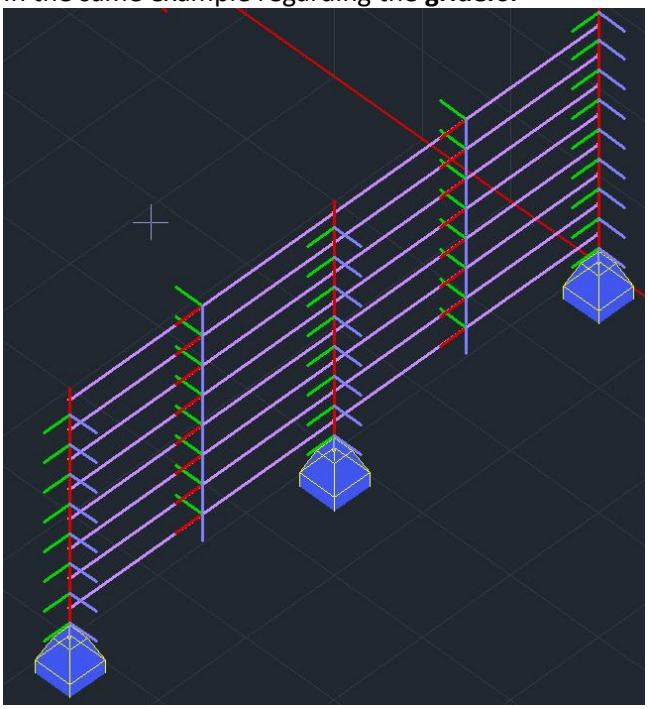
In the other direction, buckling along its **minor** axis **z-z** (blu) means:

- 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:

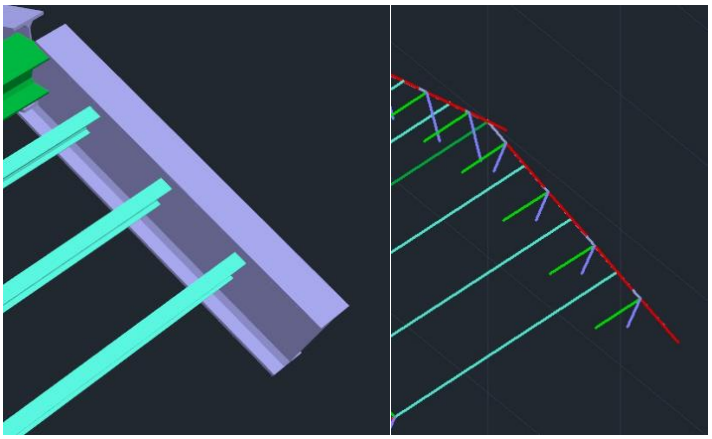
Generally, making a rule, we could say that, we consider the **merged length Ly** in the direction where the local axis y-y is parallel to the supporting elements. While in the other direction, if there are no supporting elements, **Lz** is **the length of each member**.

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



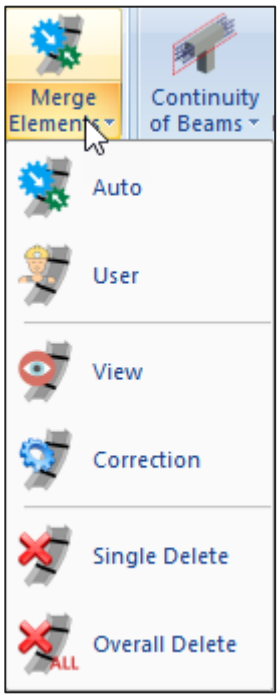
The supports from the columns are parallel to the local z-z axis (blue, out of plane) of the griders. So, merging will be in Lz (total length). While in the y-y direction (green, in the plate), Ly is the length of each member.

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.

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

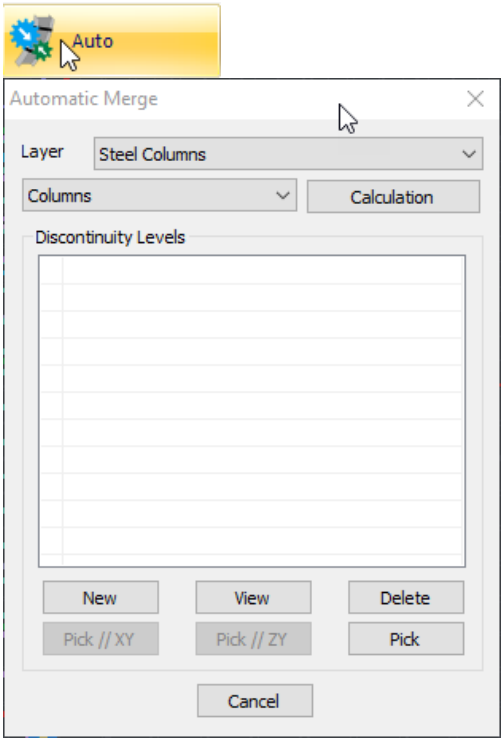
Finally, in automatic merging, there is the definition of discontinuity levels, horizontal or vertical, used as merging boundaries of a continuous element.

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:



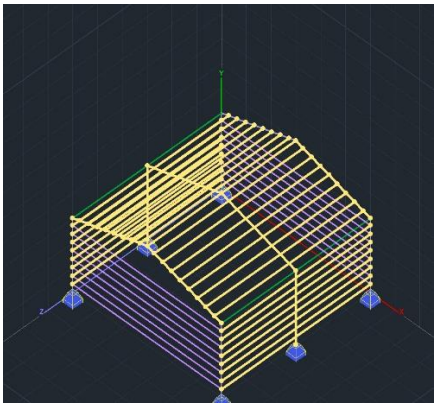
First, choose the layer of the elements to merge.

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.

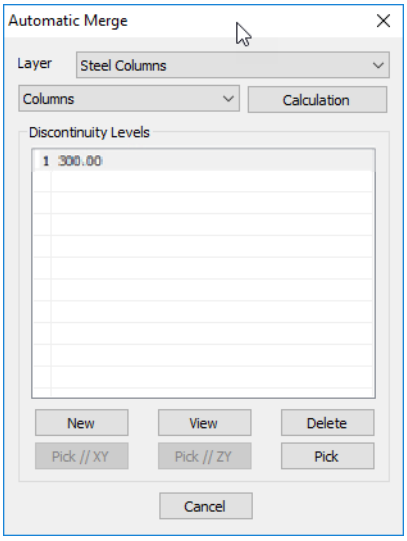
Predefined limits:

- 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:

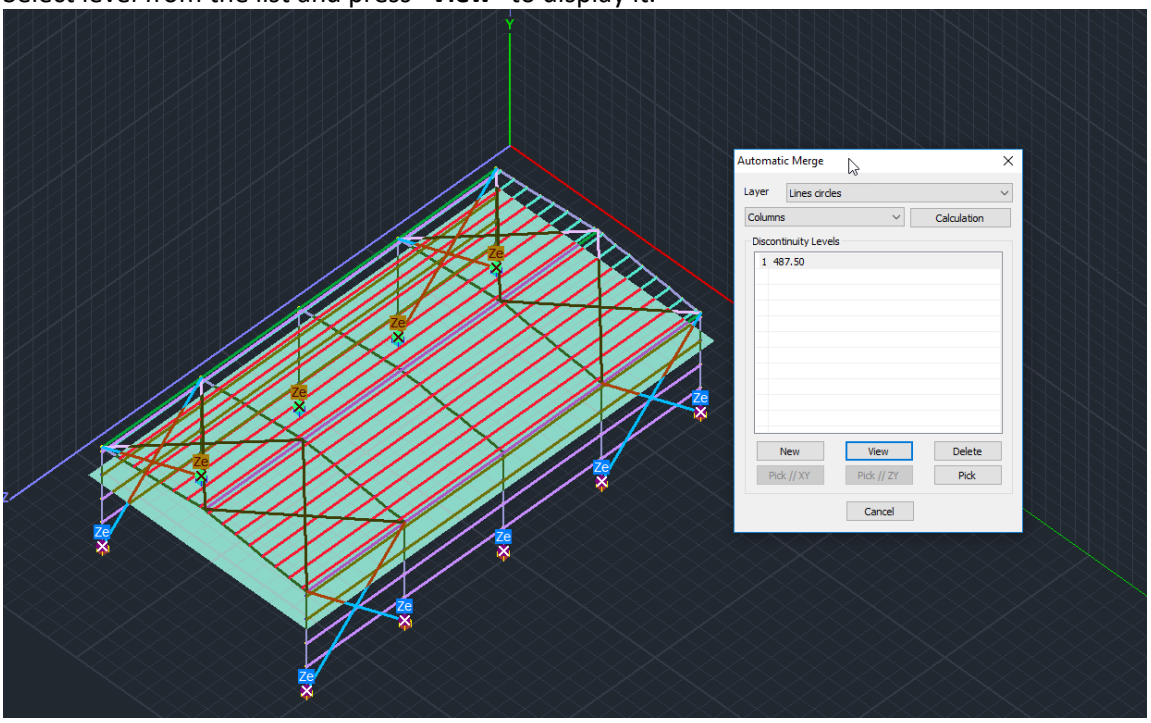
A three floor building with 0.00, 300.00 and 600.00 height levels, in **discontinuity levels** list of the columns, only the level 300.00 will be specified by default (that is, only the intermediate level without the limits)



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 **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.

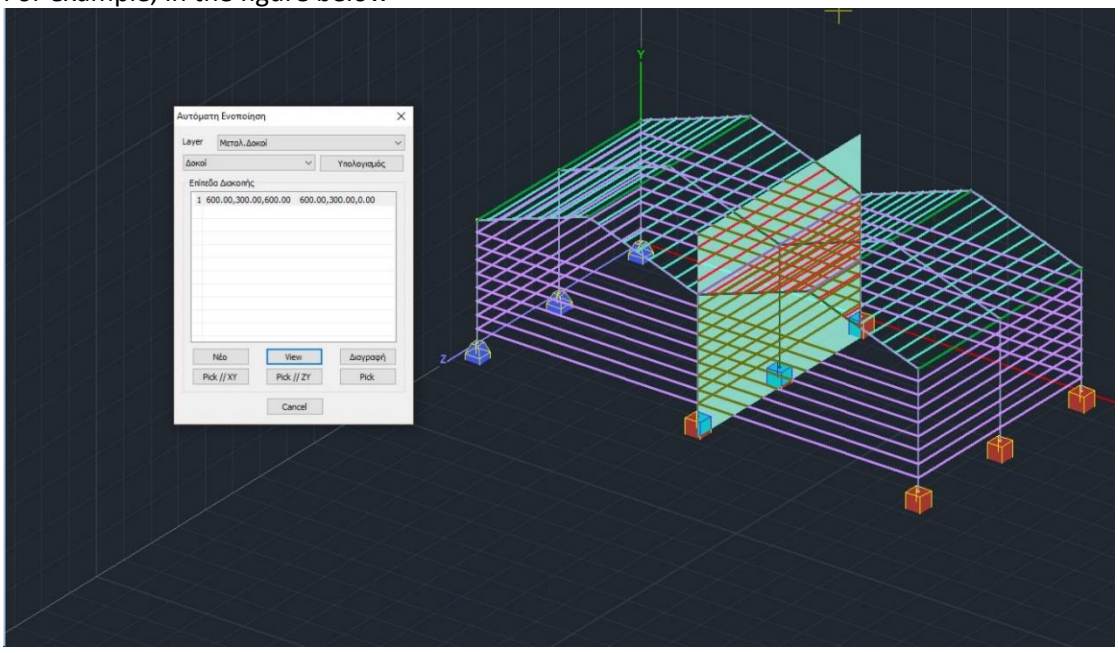


To delete a **discontinuity level,** select it from the list and press “**Delete**”.

- 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.

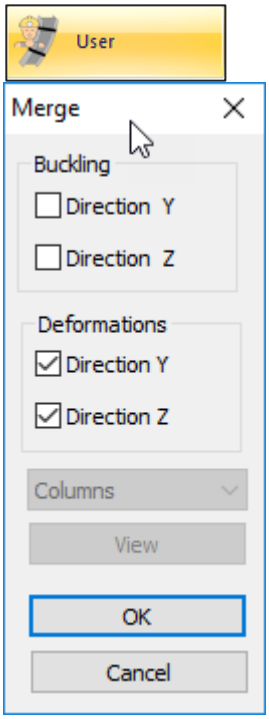
For Beam’s and Column’s **discontinuity levels,** *editing* can be achieved in two ways:

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.

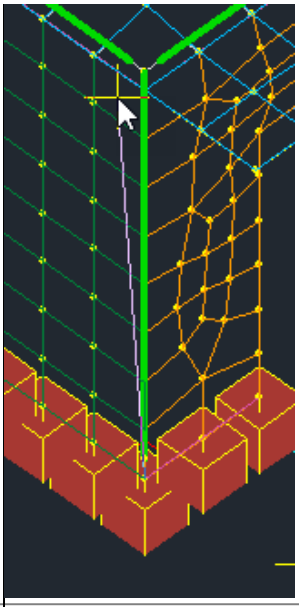
By selecting the second point (endpoint), the following dialog box appears:



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

1.4.2.1 Merge concrete columns

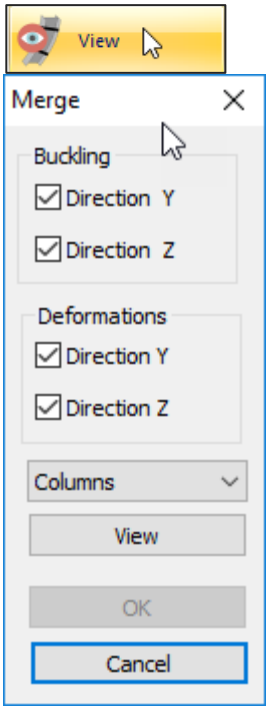
This command is mainly used in masonry buildings with vertical reinforced concrete elements which connect the nodes of the surface elements and which, to be designed, must be merged.



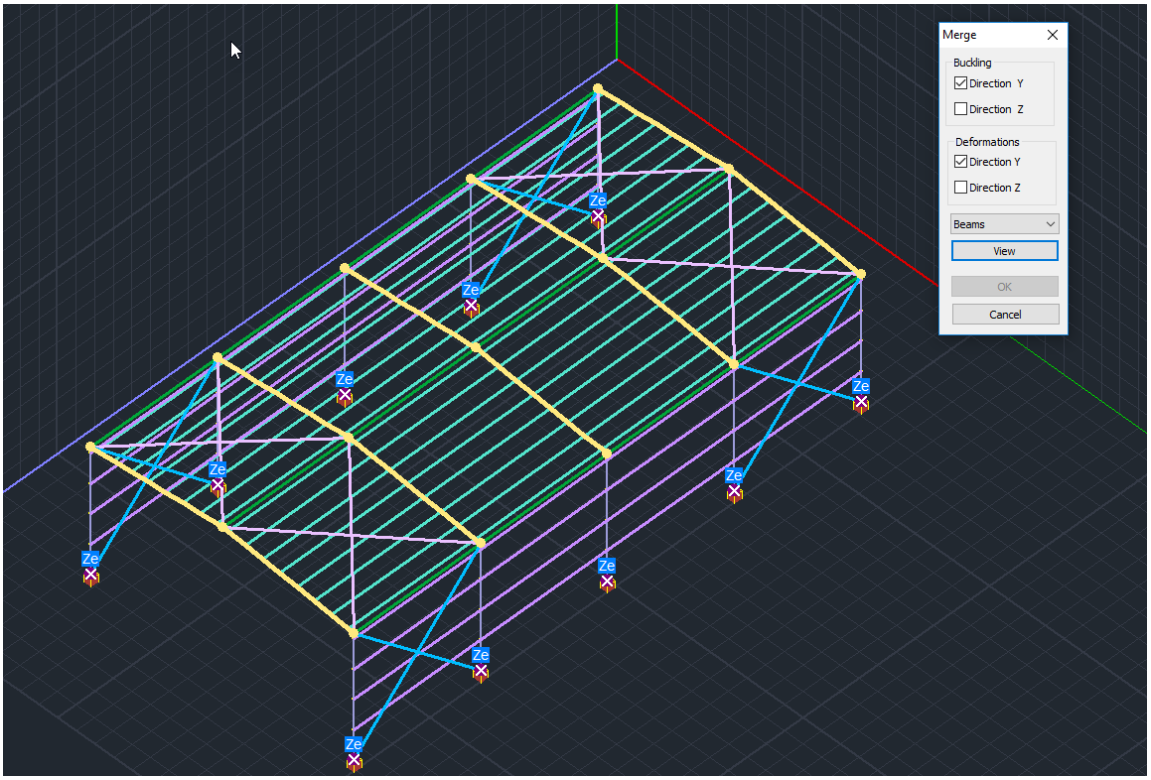
You select the command, and then you point the starting point and the end point of the members you want to merge.

1.4.3 View

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



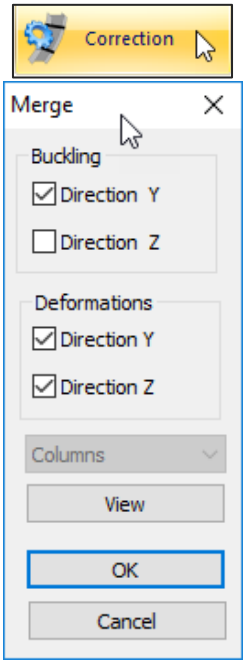
- Select element’s type from the list, then
- check in Buckling / Deformations the direction of merging to see the corresponding merged elements.



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:



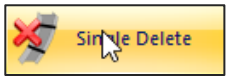
in which checks shows the merging direction.

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

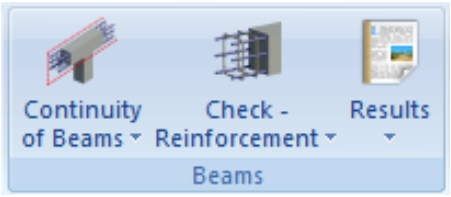


Select this command to delete a single merging.
Select the command and click a merged element. Right click to delete merging.

1.4.6 Overall Delete

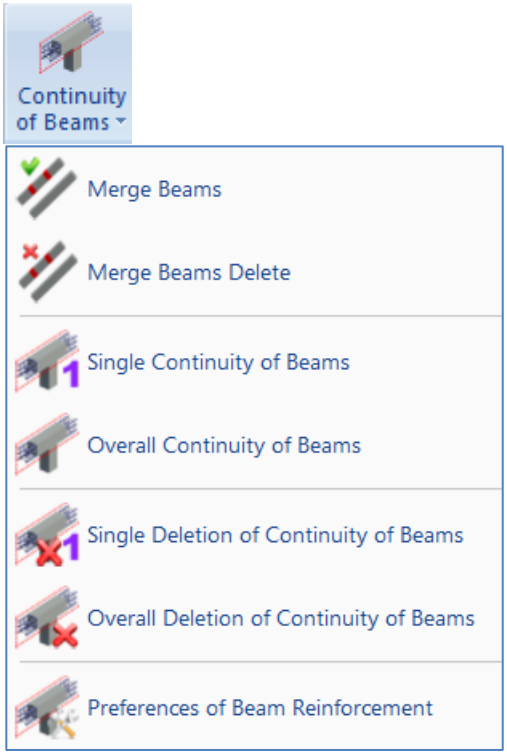
Select this command to delete all merging. Select to delete all merging from all the merged elements.

2. Beams



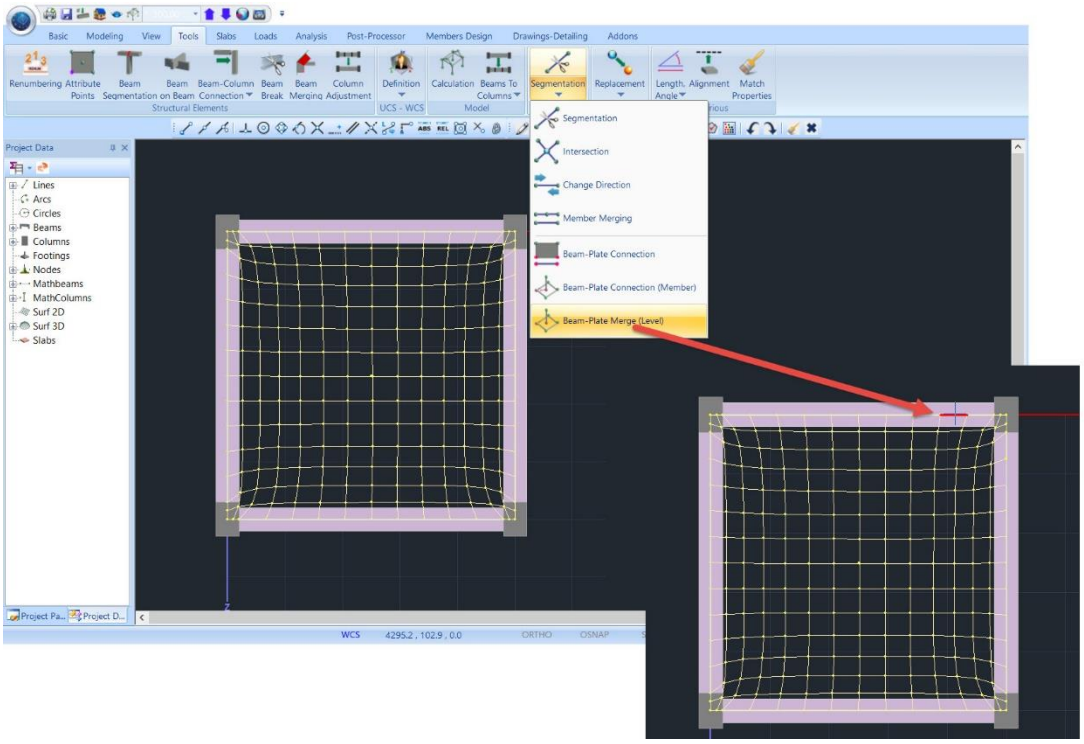
The “Beams” command group contains the following commands
- **“Continuity of Beams”**
- **“Check – Reinforcement”**
- **“Results”** related to the design checks and the checks of the steel reinforcement

2.1 Continuity of Beams



2.1.1 Merge Beams/ Merge Beams Delete

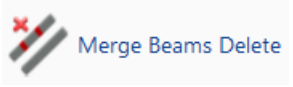
In case that beams include surface elements there is a need to break the members of the beams in order the necessary connections between the linear and surface elements to be ensured.



Consequently, by breaking the beam in small parts, the need for merging is born, for the beam to be designed as a single member. This is accomplished by using the command Merge Beams.

Select the command, and then point one by one in succession, the parts of the beam. Continue with “Continuity of Beams” and “Check Reinforcement”.

If for some reason you wish to delete a preliminary merging, select the command

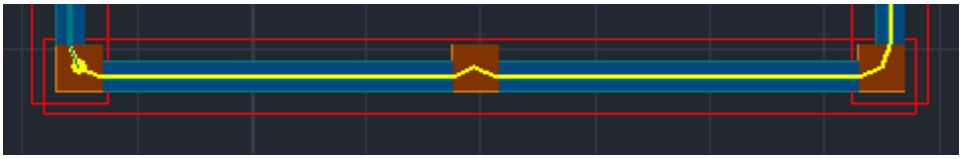


and then the first element of the merging beam. Right click to complete.

2.1.2 Single Continuity/ Overall Continuity of Beams

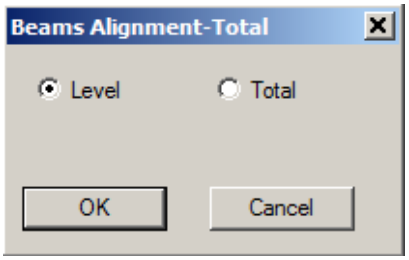
Single Continuity of Beams: This command is used to (selectively) define the beams that participate in a beam continuity.

- ✓ Left click to select the beams.
- ✓ Right click to end the selection.
- ✓ The red rectangles that appear on the screen indicate the beams’ continuity



Overall Continuity of Beams: This command is used to determine the beams’ continuities of the current level or the entire building automatically.

Select the command and in the dialog box check Level or Total:



Red rectangles that describe the beams’ continuity are displayed in the screen automatically.

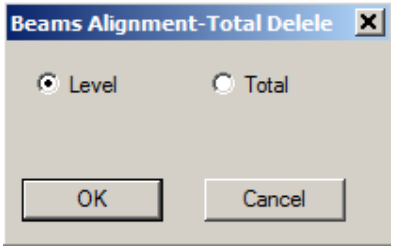
2.1.3 Single/Overall Deletion of Continuity

Single Deletion of Continuity: This command is used to (selectively) remove already created beams’ continuities.

Select the command
Left click to select the beams

Right click to end the selection

Overall Deletion of Continuity: This command is used for the removal of all existing beams’ continuities, of the current level or the entire building.



In the dialog box that appears, make the appropriate choice.

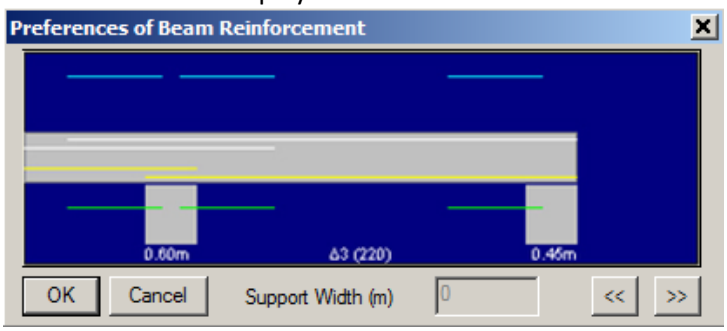
2.1.4 Preferences of Beams Reinforcement

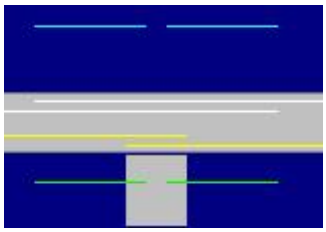

This command is used to:

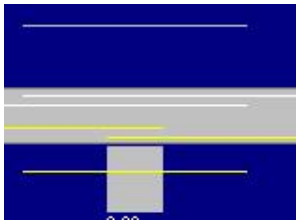
- insert one common bar or two different bars on the support of the continuous beams,
- take into account both of them,
- change the anchorage length.

EXAMPLES:

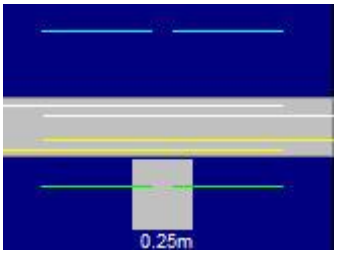
Select the command and then select a beam. The following dialog box “Preferences of Beam Reinforcement” is displayed:



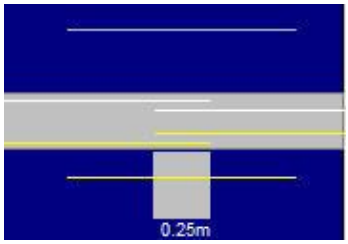




On the common support, the positioned bars consist of two different bars.
By clicking on the two bars, they are converted into one common bar.



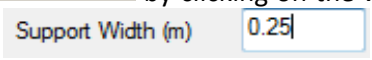
In the figure on the left, the positioned bars to the corresponding spans, pass through the spans on both sides, on the top and bottom of the beam.
This means that during the calculation, the program takes into account the bars from both spans (top and bottom).



If you want to take into account just the bars of one span, click on the yellow and white lines, to obtain the form on the left figure.

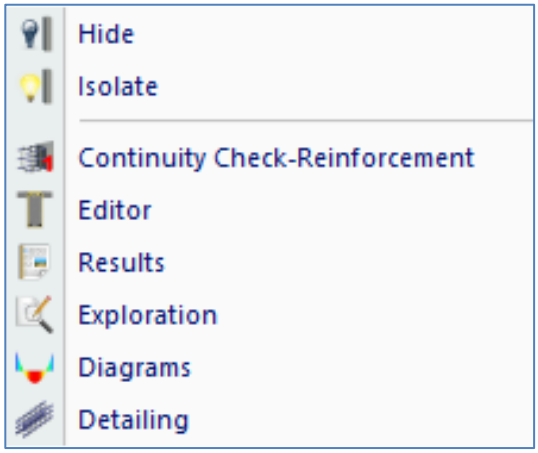
To change the bars on the left side (top and bottom) use the arrows  to view the left span.

You should change either the top or bottom rebars.

Activate the “Support Width” field  by clicking on the width value (e.g. 0.25m) to display the existing width of the beam  and type a new value, if you want to change the anchorage length.

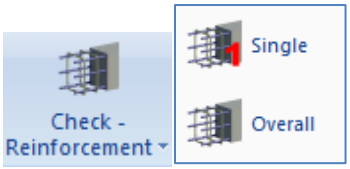
§ Mouse right button function

This is a function which is enabled by right clicking on a structural element. Move the mouse pointer over a structural element, such as a beam, and press the right button, then, the following menu options are displayed:



You can select a command from the menu on the left instead of using it from the Ribbons’ sections.

2.2 Check Reinforcement



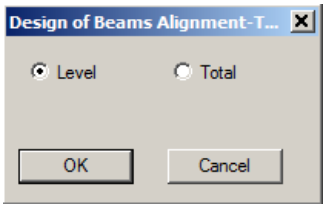
2.2.1 Single

Single: This command is used for the design of a beam or a continuity of beams. Select the command and use left click to select the beam continuity.
The program makes the design checks and displays the results using colors and symbols indicating, in this way, the type of the failure.
The colors indicating the beam’s failure are the following:
Red. Failure in Bending. It has exceeded the maximum steel reinforcement ratio ρ_{max}.
Pink. Failure in Shear / Torsion.
Cyan. Passed the design checks.
The symbol on the beam indicates:

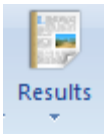
Failure in Bending	M
Failure in Shear	V
Failure in Torsion	T
Dense positioned Stirrups	Asw
It has exceeded the maximum steel reinforcement ratio	ρ
It has exceeded the maximum anchorage length	ldb
Capacity Design	αcd
It has exceeded the maximum crack width	Wk
Deflection failure	L/d

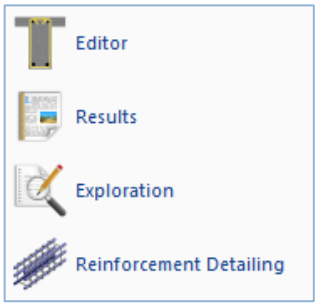
2.2.2 Overall

Overall: This command is used to apply the design checks of the beams according to their final design. Select the command and in the dialog box activate either “Level” or “Total”.



2.2.3 Results





This command includes a sub-group of commands related to the calculation process and the display of the designing as a result of beam continuity.

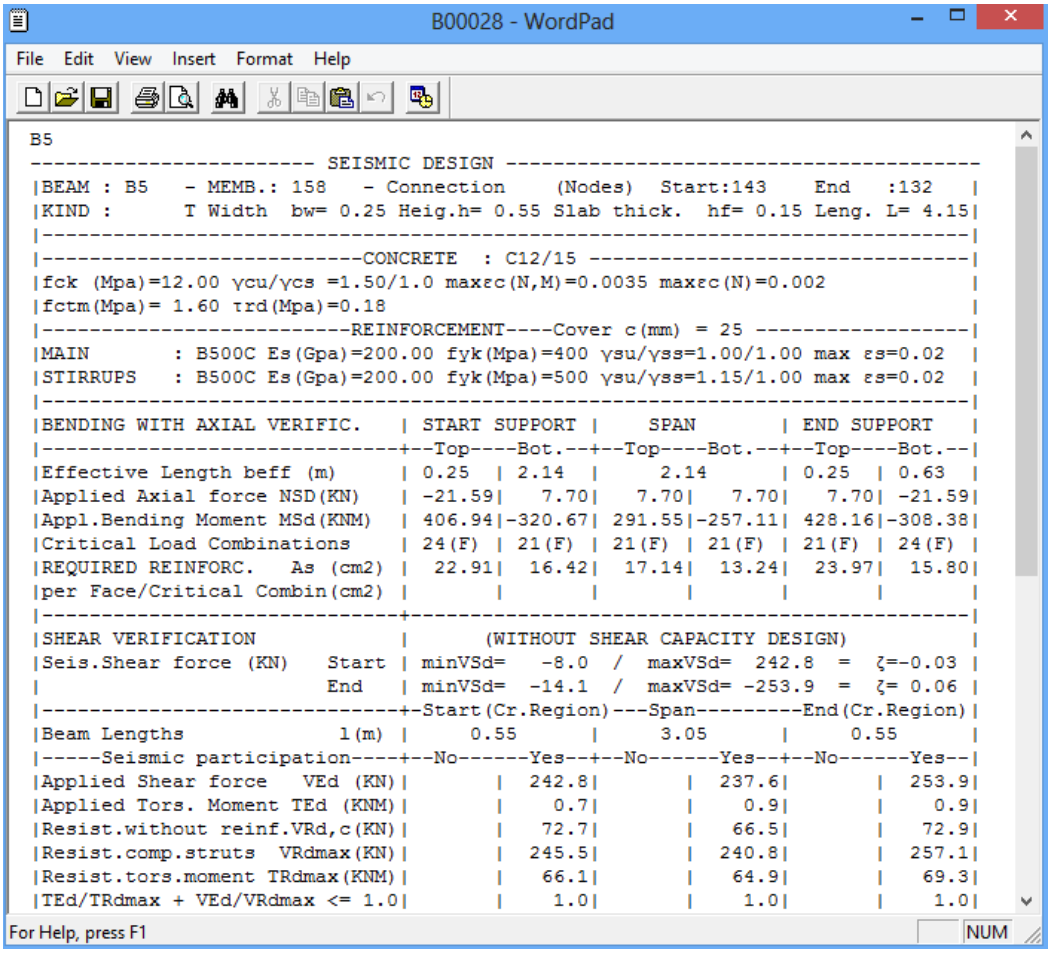
2.2.3.1 Editor

This is a command which, however, in the later versions of SCADA Pro is completely replaced by the command **Detailing**.
Use **Detailing** for editing a beam continuity in an integrated environment of calculation and design. You can also apply Retrofitting methods and calculate the new moment diagrams.

⚠ Detailed instructions on how to use this command are mentioned in the relative user’s manual (**chapter B. Beam’s Detailing**)

2.2.3.2 Results

This command is used to display a TXT file that contains the main design checks and calculation results of a beam continuity.
Select the command and then left click on the centroid axis of the beam to open the TXT file and read the results.
The TXT files that are displayed are the same files generated by the program for the printout.



B00028 - WordPad

File Edit View Insert Format Help

```
B5
------------------------ SEISMIC DESIGN ----------------------------------------
|BEAM : B5   - MEMB.: 158   - Connection    (Nodes)  Start:143    End   :132   |
|KIND :      T Width  bw= 0.25 Heig.h= 0.55 Slab thick.  hf= 0.15 Leng. L= 4.15|
|-------------------------------------------------------------------------------|
|---------------------------CONCRETE  : C12/15 ---------------------------------|
|fck (Mpa)=12.00 γcu/γcs =1.50/1.0 maxεc(N,M)=0.0035 maxεc(N)=0.002             |
|fctm(Mpa)= 1.60 τrd(Mpa)=0.18                                                  |
|-------------------------REINFORCEMENT----Cover c(mm) = 25 -------------------|
|MAIN       : B500C Es(Gpa)=200.00 fyk(Mpa)=400 γsu/γss=1.00/1.00 max εs=0.02  |
|STIRRUPS   : B500C Es(Gpa)=200.00 fyk(Mpa)=500 γsu/γss=1.15/1.00 max εs=0.02  |
|-------------------------------------------------------------------------------|
|BENDING WITH AXIAL VERIFIC.   | START SUPPORT |    SPAN       | END SUPPORT   |
|------------------------------+--Top----Bot.--+--Top----Bot.--+--Top----Bot.--|
|Effective Length beff (m)     | 0.25  | 2.14  |     2.14      | 0.25  | 0.63  |
|Applied Axial force NSD(KN)   | -21.59|  7.70|  7.70|  7.70|  7.70| -21.59|
|Appl.Bending Moment MSd(KNM)  | 406.94|-320.67| 291.55|-257.11| 428.16|-308.38|
|Critical Load Combinations    | 24(F) | 21(F) | 21(F) | 21(F) | 21(F) | 24(F) |
|REQUIRED REINFORC.   As (cm2) |  22.91|  16.42|  17.14|  13.24|  23.97|  15.80|
|per Face/Critical Combin(cm2) |       |       |       |       |       |       |
|------------------------------+-----------------------------------------------|
|SHEAR VERIFICATION            |      (WITHOUT SHEAR CAPACITY DESIGN)          |
|Seis.Shear force (KN)   Start | minVSd=   -8.0  /  maxVSd=  242.8  =  ζ=-0.03 |
|                        End   | minVSd=  -14.1  /  maxVSd= -253.9  =  ζ= 0.06 |
|------------------------------+-Start(Cr.Region)---Span---------End(Cr.Region)|
|Beam Lengths            l(m)  |     0.55      |     3.05      |     0.55      |
|-----Seismic participation----+--No------Yes--+--No------Yes--+--No------Yes--|
|Applied Shear force   VEd (KN)|       |  242.8|       |  237.6|       |  253.9|
|Applied Tors. Moment TEd (KNM)|       |   0.7|       |   0.9|       |   0.9|
|Resist.without reinf.VRd,c(KN)|       |  72.7|       |  66.5|       |  72.9|
|Resist.comp.struts  VRdmax(KN)|       |  245.5|       |  240.8|       |  257.1|
|Resist.tors.moment TRdmax(KNM)|       |  66.1|       |  64.9|       |  69.3|
|TEd/TRdmax + VEd/VRdmax <= 1.0|       |   1.0|       |   1.0|       |   1.0|
```

For Help, press F1 NUM

A NEW WARNING SYMBOL FOR INEFFICIENCY OF THE ANCHORING LENGTH IN THE BEAMS.

The way of calculating the lbd anchorage lengths for different regulations is summarized below: The total lbd is calculated and divided into l1 and l2. L1 is the linear anchorage length, and l2 is the one rotated into the node.

NOTES:

- EC2 does not provide a minimum linear anchorage length but provides a minimum TOTAL anchorage length (l1 + l2) which is also called lb, min.
- EC8 in paragraph 5.6.2, among others, provides ONLY for DCH the anchorage length to be linear (exaggerating).

Based on the above, resulting:

1. For EC2 w/o EC8 scenario as well as for all EC with DCL and DCM ductility classes, it does not obey a minimum linear anchorage length lb, min, but checks the total length lbd regarding lb, min according to 8.4.4 Of the EC2.

So, there will never be an error message because if the anchorage length is bigger than the width of the support, the bar will reach the alignment and then return to the node.

2. For EC with DCH high ductility class, the use of the minimum linear anchorage length is allowed according to 5.6.2 of EC8. The error message will be displayed.

⚠ Also, the affinity regions are now taken into account for calculating lb. The upper reinforcement is region II while the lower I.

2.2.3.3 Exploration

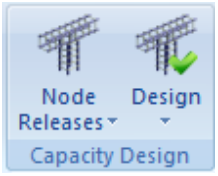
 This command is used to display the TXT file that contains all design checks and calculation results of a beam. Select the command and left click on the centroid axis of the beam to open the TXT file and read the results.

2.2.3.4 Reinforcement detailing

⚠ Use this command for editing a beam continuity in an integrated environment of claculation and design. You’re also able to apply Retrofitting methods and calculate the new moment diagrams.

⚠ Detailed instructions on how to use this command are mentioned to the relative user’s manual (**chapter A. Beam’s Detailing**)

3. Capacity design

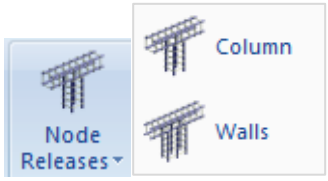


The command group “Capacity Design” contains commands for performing capacity design.

⚠ Capacity Design always precedes columns and walls design if necessary.

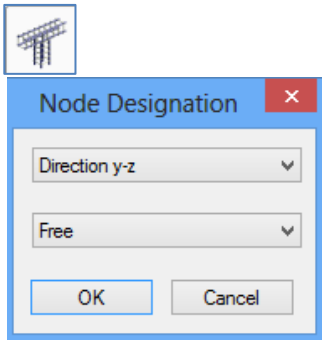
3.1 Node Releases

This command is used to determine the node releases of the columns and walls.

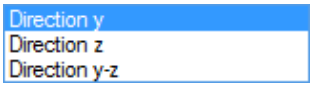


3.1.1 Column

Specify the node releases of the column for the capacity design application.
Select the command and the column’s nodes for changing their releases. Right click to end selection and the following dialog box opens:



The first list contains the directions for the capacity design application



The second list contains

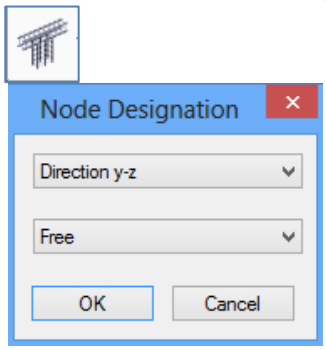
the node releases for the capacity design application.

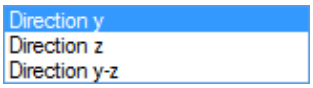
ATTENTION:

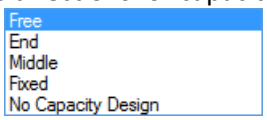
⚠ Note that this command is used only for **columns’** nodes.

3.1.2 Walls

Specify the node support releases of the wall for the capacity design application.
Select the command and the column’s nodes for changing the support conditions. Right click to end selection and the dialog box opens:



The first list contains the directions for capacity design application 

The second list contains 

the node condition for the designation.

ATTENTION:

- Note that this command is used only for **walls’** nodes.

NOTE:

- If the user skips the definition of the node releases, the program will consider them as free in both directions except for the fixed ones.

3.2 Design

This command is used to apply the capacity design and display the results.

NOTES:

- The beams design precedes the application of the capacity design
- The capacity design, always precedes the columns and the design of the wall, if it is necessary to be applied.
- A necessary precondition for both selective and total Design control is that beams have been designed and "Amplification" has been activated in Beams and Columns fields in the Structural Component Parameters window

3.2.1 Single

 This command is used to apply the capacity design on a single node. Select the command and left click on the node.

3.2.2 Overall

 This command is used to apply the capacity design on every node of the current level. Select the command and repeat the procedure for the rest of the levels.

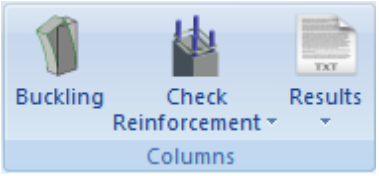
3.2.3 Results

 This command is used for the display of the TXT file that contains the results of the main design checks of the capacity design. Select the command and left click on the node to open the TXT file and read the results.

3.2.4 Exploration

 This command is used for the display of the TXT file that contains all the results of the design checks of the capacity design. Select the command and left click on the node to open the TXT file and read the results.

4. Columns



The command group “Columns” contains commands about buckling check, design checks, steel reinforcement checks and results for columns and shear walls.

NOTE:

⚠ Columns' and walls' design always follows the capacity design; if it is necessary to be applied. First, the capacity design is applied on every required level, and then the design checks for columns and walls are performed.

4.1 Buckling

This command will be fully available in the next version.

4.2 Check Reinforcement

This is a command list related to the design checks for columns and walls resulting in their final design.

4.2.1 Single:

Select the command and left click to select one or more columns or walls. The program performs the appropriate design checks and displays the results by colors and symbols indicating the type of failure.

A colored dot is displayed in the center of the element.
The color changes according to the type of failure as follows:

- **Red:** Failure caused by biaxial bending. The steel reinforcement exceeded the maximum ratio of 4%. Dense stirrups. No results are displayed.
- **Pink:** Failure by Shear / Torsion or exceedance of the ductility level. The results show the reason for failure.
- **Cyan:** All design checks are verified.

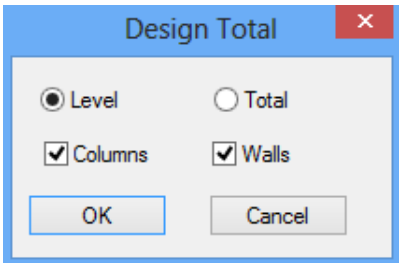
The initially indicated type of failure appears above the element as well:

Failure by biaxial bending	M-N
Failure by Shear	V
Confinement failure	ωwd
Buckling failure	λ
Failure by Torsion	T
Dense Stirrups	Asw
Exceedance of 4% steel reinforcement ratio	ρ
Exceedance of the ductility index	ν

4.2.2 Overall

This command is used for the calculation of the final columns and/or walls design in total.

Select the command and the following dialog box is displayed:



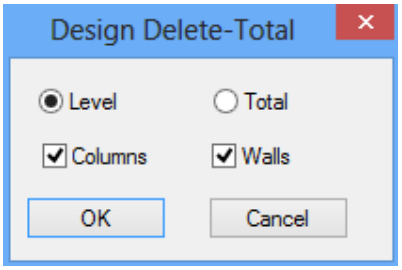
Select whether to design all columns/walls of the current level or the building in total.

4.2.3 Single Delete

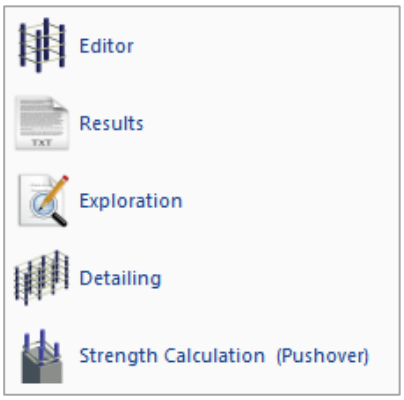

This command is used to delete the previous calculations from one or more columns or walls.
Press the command and left click to select the columns and the walls.

4.2.4 Overall Delete


This command is used to delete the previous calculations from columns or walls of the current level or the building in total according to the selection in the dialog box.



4.3 Results



This command contains a list of commands related to the elaboration and the display of the results derived from the design checks of beams continuity.

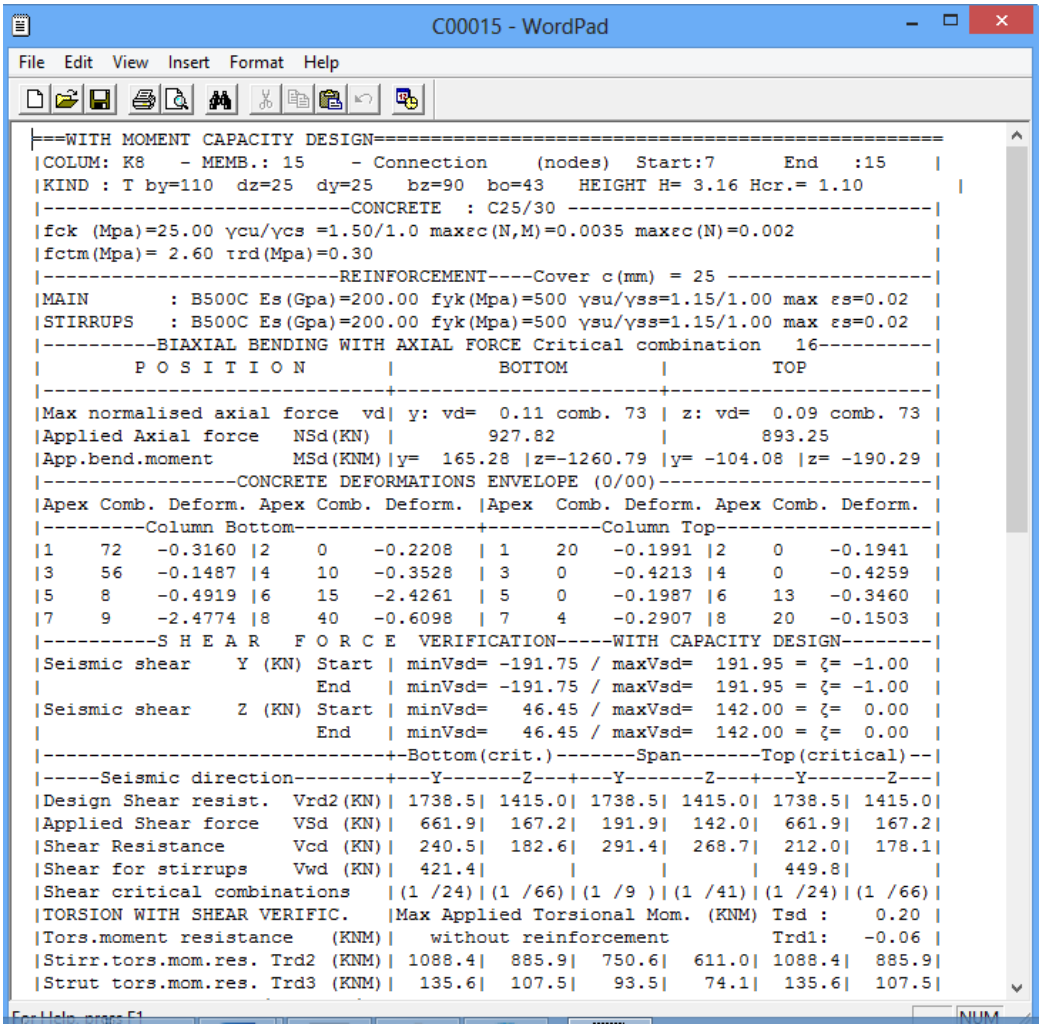
4.3.1 Editor


This is a command which, however, in the later versions of SCADA Pro is completely replaced by the command **Detailing**.
Use **Detailing** for editing a column in an integrated environment of verification and design. You can also apply Retrofitting methods and calculate the new moment diagrams.

⚠ Detailed instructions on how to use this command refer to the related user manual (**chapter B. Column’s Detailing**)

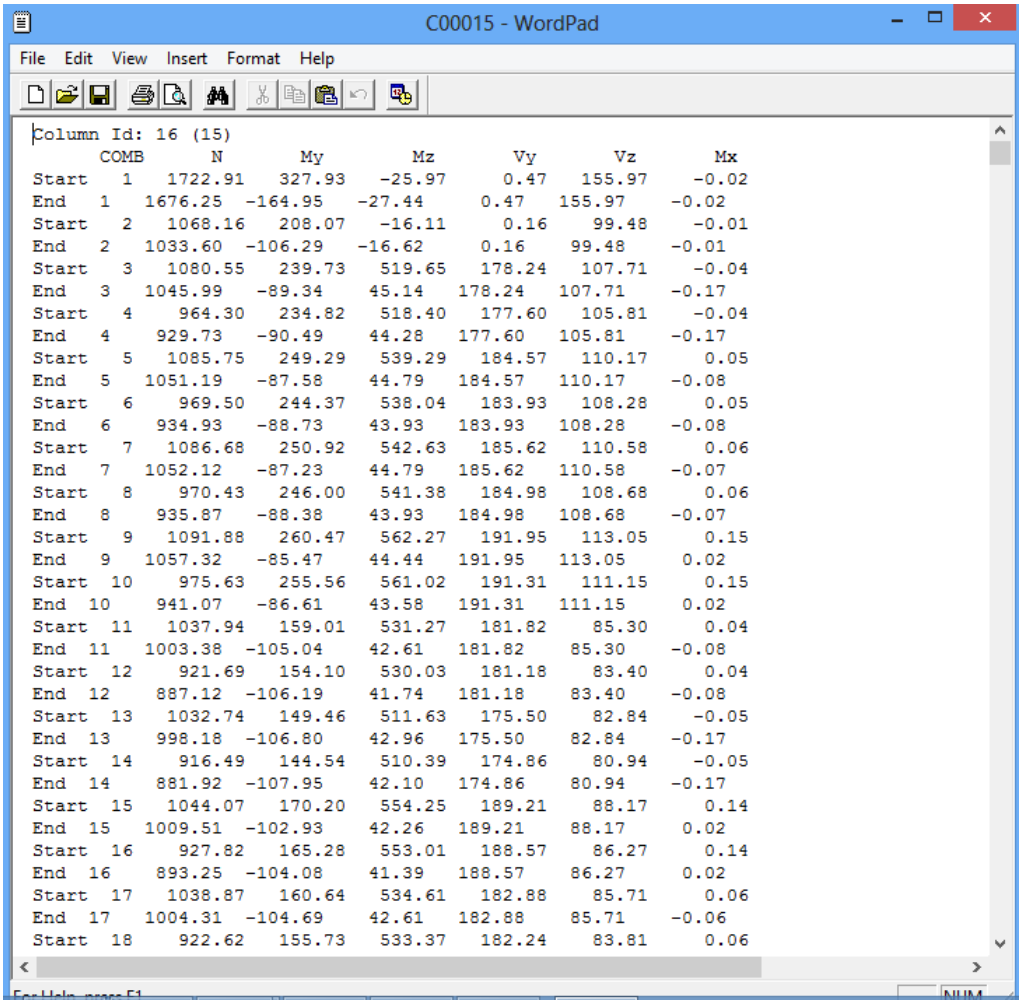
4.3.2 Results


Select to display the .txt file containing the main checks and design results of columns and walls. Select the command and left click on beam’s axis to open the .txt file and read the results. The *txt files displayed are those generated by the program for the printout.



4.3.3 Exploration

Select this command to display the TXT file containing the results of all design checks of columns and walls. Select the command and left click on the beam’s axis to open the TXT file and read the relative results.



C00015 - WordPad

File Edit View Insert Format Help

Column Id: 16 (15)

	COMB	N	My	Mz	Vy	Vz	Mx
Start	1	1722.91	327.93	-25.97	0.47	155.97	-0.02
End	1	1676.25	-164.95	-27.44	0.47	155.97	-0.02
Start	2	1068.16	208.07	-16.11	0.16	99.48	-0.01
End	2	1033.60	-106.29	-16.62	0.16	99.48	-0.01
Start	3	1080.55	239.73	519.65	178.24	107.71	-0.04
End	3	1045.99	-89.34	45.14	178.24	107.71	-0.17
Start	4	964.30	234.82	518.40	177.60	105.81	-0.04
End	4	929.73	-90.49	44.28	177.60	105.81	-0.17
Start	5	1085.75	249.29	539.29	184.57	110.17	0.05
End	5	1051.19	-87.58	44.79	184.57	110.17	-0.08
Start	6	969.50	244.37	538.04	183.93	108.28	0.05
End	6	934.93	-88.73	43.93	183.93	108.28	-0.08
Start	7	1086.68	250.92	542.63	185.62	110.58	0.06
End	7	1052.12	-87.23	44.79	185.62	110.58	-0.07
Start	8	970.43	246.00	541.38	184.98	108.68	0.06
End	8	935.87	-88.38	43.93	184.98	108.68	-0.07
Start	9	1091.88	260.47	562.27	191.95	113.05	0.15
End	9	1057.32	-85.47	44.44	191.95	113.05	0.02
Start	10	975.63	255.56	561.02	191.31	111.15	0.15
End	10	941.07	-86.61	43.58	191.31	111.15	0.02
Start	11	1037.94	159.01	531.27	181.82	85.30	0.04
End	11	1003.38	-105.04	42.61	181.82	85.30	-0.08
Start	12	921.69	154.10	530.03	181.18	83.40	0.04
End	12	887.12	-106.19	41.74	181.18	83.40	-0.08
Start	13	1032.74	149.46	511.63	175.50	82.84	-0.05
End	13	998.18	-106.80	42.96	175.50	82.84	-0.17
Start	14	916.49	144.54	510.39	174.86	80.94	-0.05
End	14	881.92	-107.95	42.10	174.86	80.94	-0.17
Start	15	1044.07	170.20	554.25	189.21	88.17	0.14
End	15	1009.51	-102.93	42.26	189.21	88.17	0.02
Start	16	927.82	165.28	553.01	188.57	86.27	0.14
End	16	893.25	-104.08	41.39	188.57	86.27	0.02
Start	17	1038.87	160.64	534.61	182.88	85.71	0.06
End	17	1004.31	-104.69	42.61	182.88	85.71	-0.06
Start	18	922.62	155.73	533.37	182.24	83.81	0.06

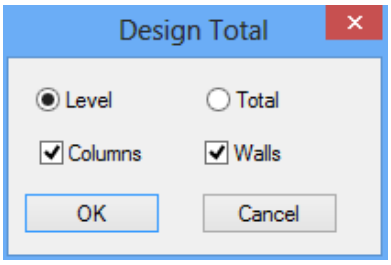
4.3.4 Detailing

- Use this command for editing a beam continuity in an integrated environment of verification and design. You can also apply Retrofitting methods and calculate the new moment diagrams.
- Detailed instructions on how to use this command refer to the related user’s manual (**chapter B. Column’s Detailing**)

4.3.5 Strength Calculation (Pushover)

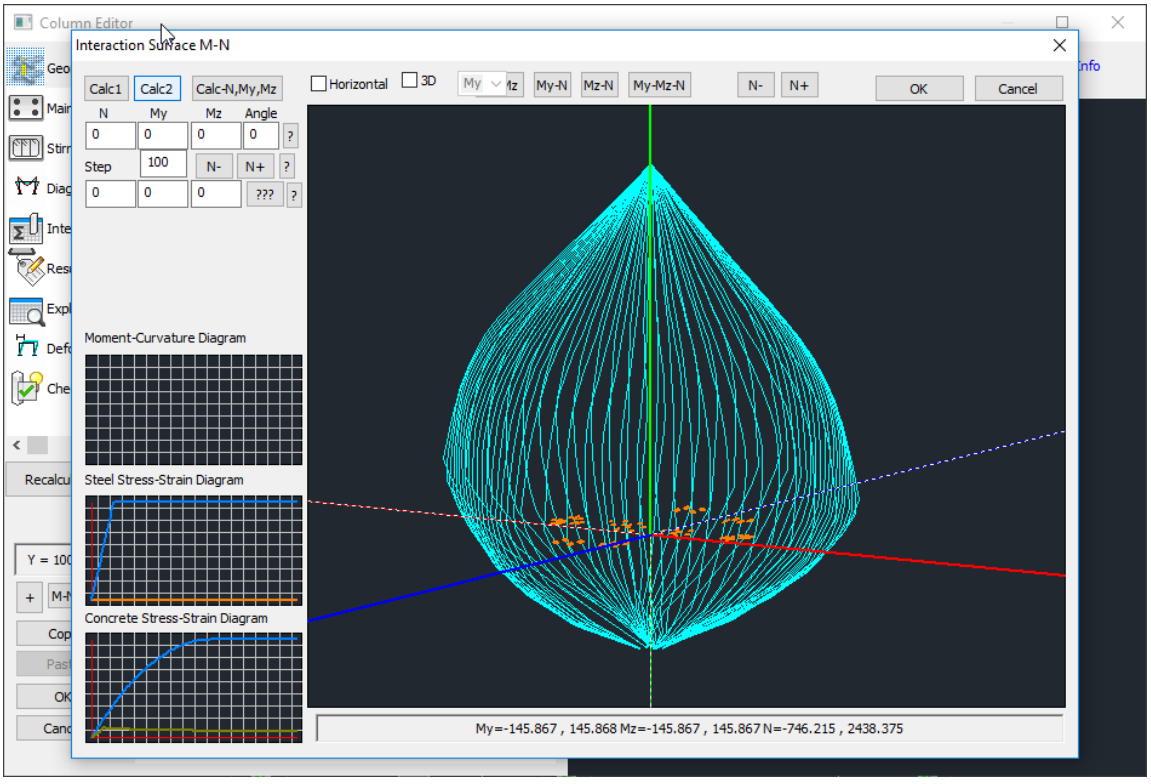
This command refers to the nonlinear static analyses (Pushover analyses). First, the existing structure is designed by the design provisions that were active the time of the construction of the structure. This means that the dimensions of the structural elements as well as the existing steel reinforcement are defined. Next step is the performance of the Pushover analysis. Nevertheless, before the Pushover analysis, the "**Strength Calculation**" procedure precedes, by selecting the corresponding command.

Through this command, the program calculates the interaction diagrams between bending moment (M) and axial force (N) for all columns of the structure in all levels.



Press the command and in the dialog box choose the columns and/or the walls of the current level or the total building, to calculate the new interaction diagrams M-N.

Wait until the program completes the calculation of all selected items.

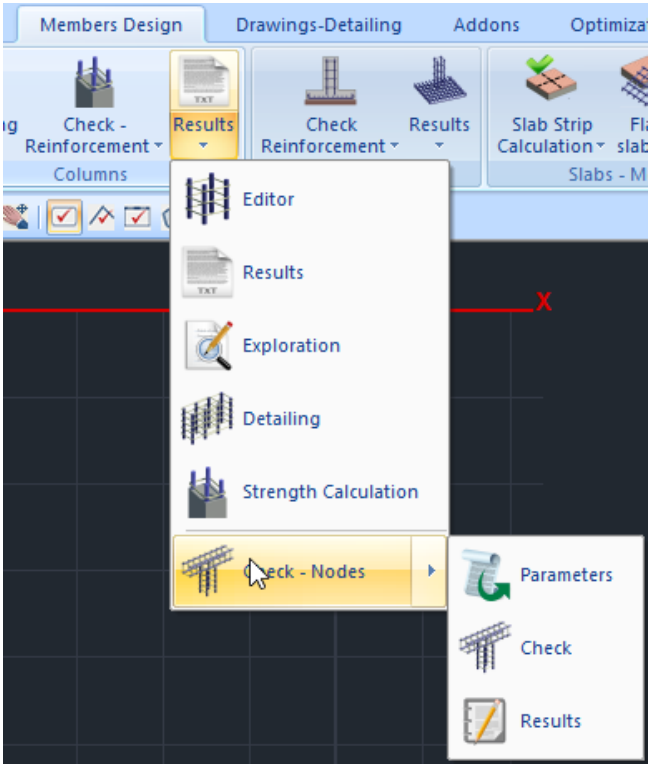


⚠ The calculation of the interaction diagram M-N can also be performed for a single column or wall, using the command "Detailing" (see the corresponding **chapter B. Column's Detailing**).

4.3.6 Check-Nodes

In the new version of SCADA Pro, the shear strength check of the node was added according to par. 7.2.5. Of KAN.EPE.
The two checks to execute are:

- Diagonal tension cracking
- Failure in diagonal ccompression

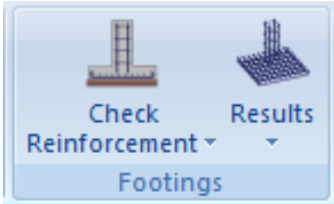


NOTE:

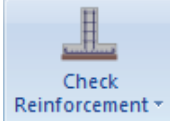
- This check concerns only the Greek KAN.EPE norm and is no valid in other countries.
- It works only in the Greek version.

5. Footing

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



5.1 Check Reinforcement



This command contains a list of commands related to the calculation of the footing designing.

5.1.1 Single

Select this command and then left click to select one or more footings. The program performs the design checks and the corresponding results are displayed by colors and symbols that indicate the type of failure.
The color of the node indicates that the design checks of the footing:

 were satisfied or  failed.

The type of failure is mentioned with a symbol as well:

Failure in Bending	M
Failure in Shear	V
Punching failure	vp
Soil failure	σuls
Damage bearing elements	Δ/l
Eccentricity	E

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

5.1.2 Overall

Select the command to check all footings on the current level (foundation).

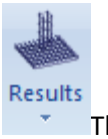
5.1.3 Single Deletion

This command is used to delete the results of the previous design from one or more footings. Select the command and then left click to select the footings.

5.1.4 Overall Deletion

This command is used to delete the results of the previous design from all footings in the foundation level.

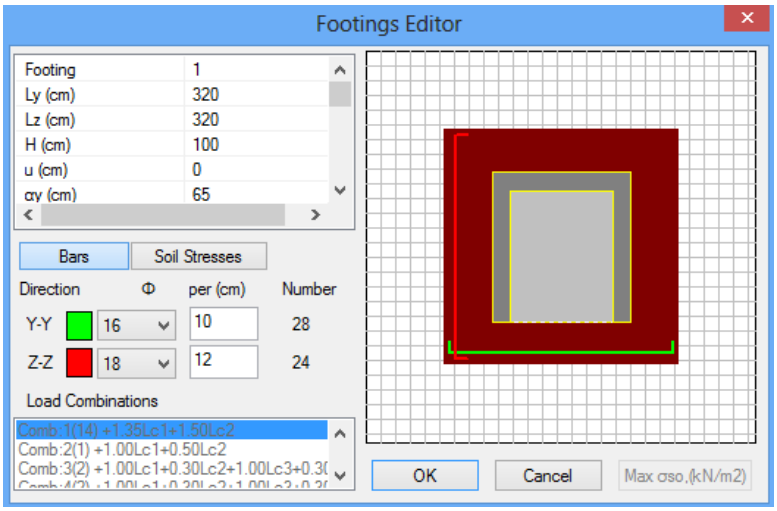
5.2 Results



This command contains a list of commands about editing the designed footings.

5.2.1 Editor

It’s an editing tool used to modify the results of the footing design process.
Select the command and left click on an already designed footing. Then, the following dialog box is displayed:



The two main choices are the following; Bars and Soil Stresses.

- If you choose the button “Bars”, the steel reinforcement section is activated:

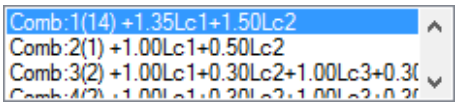
Direction	Φ	per (cm)	Number
Y-Y	16	10	28
Z-Z	18	12	24

Here you can view and modify the reinforcing bars of this footing.
Select a new diameter and type the spacing. Colors correspond to the respective bars shown in the figure.
The number of the bars presented in the “Number” column, changes automatically according to the new spacing value.

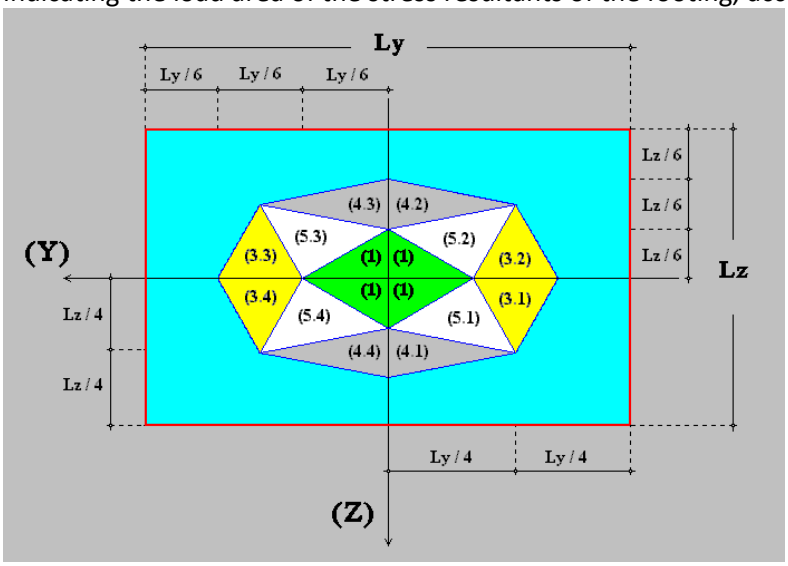
In the upper left section, a table is displayed with the title and the values of the geometric data of the footing, and other data related to soil.

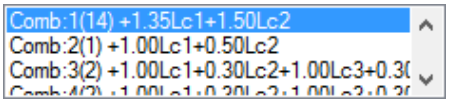
Footing	1	dz (cm)	205
Ly (cm)	320	hsoil (cm)	0
Lz (cm)	320	Footing Weight (KN)	256.00
H (cm)	100	Soil Weight(KN)	0.00
u (cm)	0	σal.(kN/m2) (102)	121.712
αy (cm)	65	σfr.(kN/m2) (36)	385.601

❖ If you click the “Soil Stresses” button the section of the load combinations is activated:



In the figure on the right, the soil stresses on the four vertices of the footing are displayed, indicating the load area of the stress resultants of the footing, according to the sketch below.

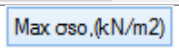


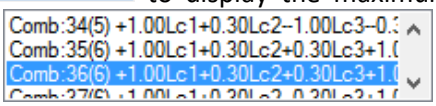


Choose a combination and scroll the list to read the respective stresses σ1, σ2, σ3, σ4 and the critical stresses σ_{al} and σ_{fr}:

σfr.(kN/m2) (36)	385.601
Stresses (kN/m2)	0
σ1	134.442
σ2	156.188
σ3	167.607
σ4	145.862

σal.(kN/m2) (102)	121.712
σfr.(kN/m2) (36)	385.601
Stresses (kN/m2)	0
σ1	134.442
σ2	156.188
σ3	167.607

Finally, press the button “Max σso (kN/m2)” 

to display the maximum stress developed, and the load combination of the origin. 

5.2.2 Results


Select this command to display the TXT file that contains the main design checks of footings and the corresponding results. Select the command and left click on a footing to open the TXT file and read the results.
The displayed TXT files are those generated by the program for the printout.

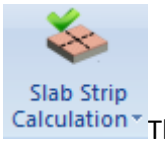
5.2.3 Exploration


Select this command to display the TXT file that contains all design checks for footings and the corresponding results. Select the command and left click on the beam’s centroid axis to open the TXT file and read the results.

6. Slabs-Mesh

The “Slabs-Mesh” command group includes commands related to the analysis of slabs with the strip method and the corresponding results and commands to insert, delete, edit and generate a mesh.

6.1 Slab Strip Calculation



This command list is related to the analysis of slabs with the strip method.

6.1.1 Single


Select the command and left click on the strip.
The slab strips are analyzed, the stress resultants are calculated and the designing of the slab is performed. The program calculates the tension (Fe) and compression (Fe') and the steel reinforcement in cm^2. Also the reinforcing bars in span, additional and secondary reinforcement and stirrups, for solid and Zoellner slabs, are calculated.

6.1.2 Overall


To calculate all slab strip of the current level.

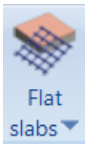
6.1.3 Single (Load Pattern)


Perform analysis of the selected strips by the load pattern as well.

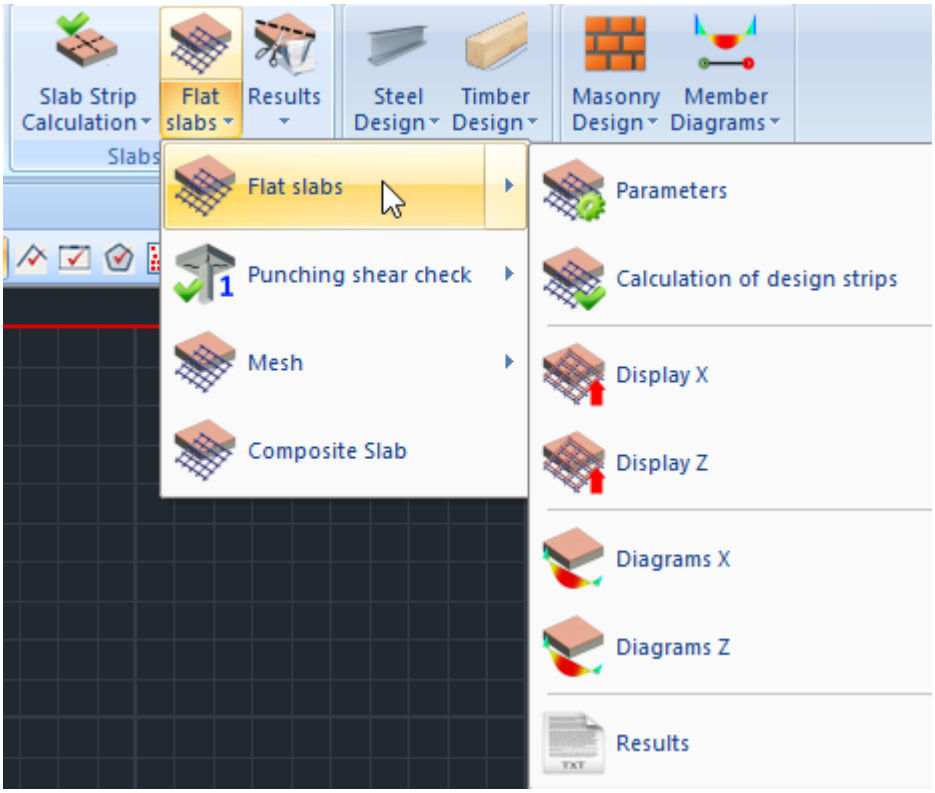
6.1.4 Overall (Load Pattern)


Perform analysis of all the current level strips by the load pattern as well.

6.2 Flat Slabs



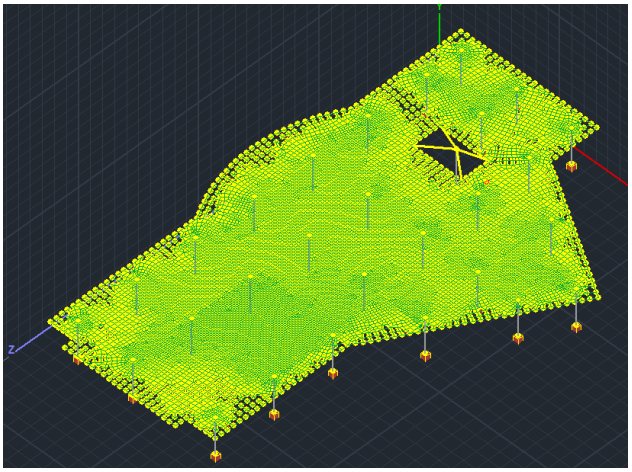
6.2.1 Flat Slabs



The new version of SCADA Pro offers the possibility of creating flat slabs (slabs without the presence of beams) with the finite element method.

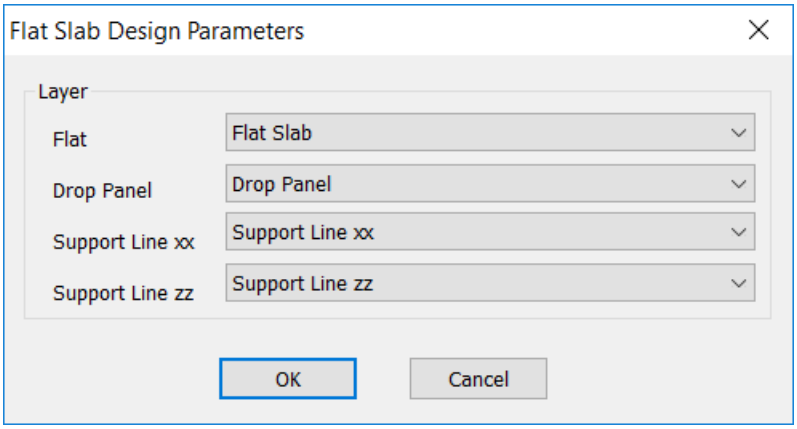
The procedure for the modeling of flat slabs requires:

- the 3D Mesh definition,
- the External Boundary creation,
- the Holes automatically creation in place of the Columns,
- the Mesh calculation and the mathematical model calculation.

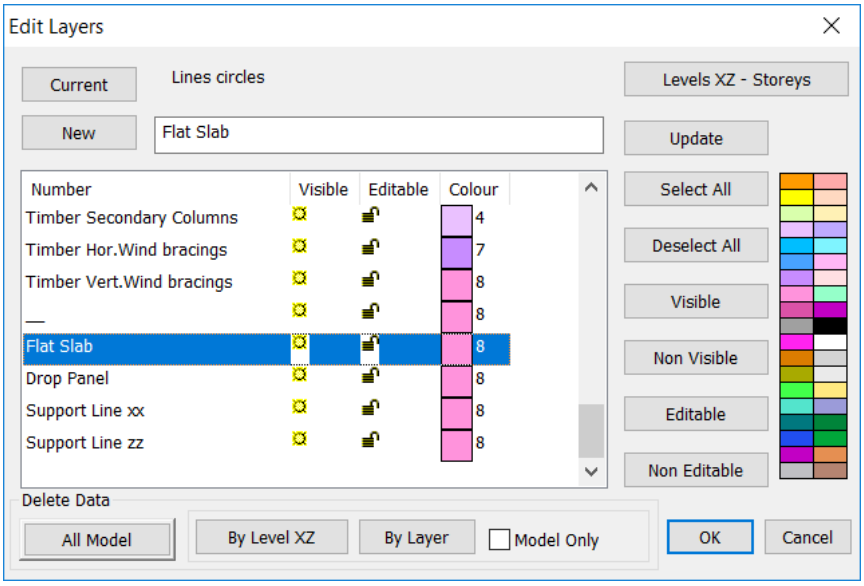


The command "Flat slabs" includes the commands:

6.2.1.1 Parameters

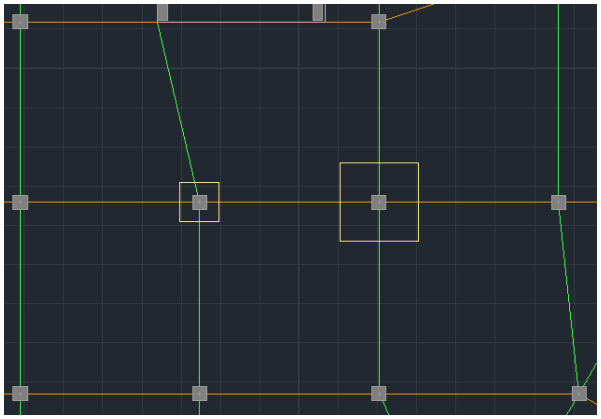


In the dialog box, you define the correlation between Layers.



The default Layer’s list of SCADA includes the layers related to the flat slabs.

- In “Flat Slab” layer transfer the outline of the slab and correspond it to the Layer “Flat”.
- In “Drop Panel” layer transfer the Lines that define the area around the columns, where you will increase the thickness of the slab locally. The "Drop Panels" are inserted optionally around the columns of the slab relieving the fatigue from drilling.
- In “Support Lines xx” and “Support Lines zz” transfer the Lines that define the Support Lines. These lines are inserted in both X and Z directions between successive points of the slab. They usually connect column’s nodes and end on the outline of the slab.



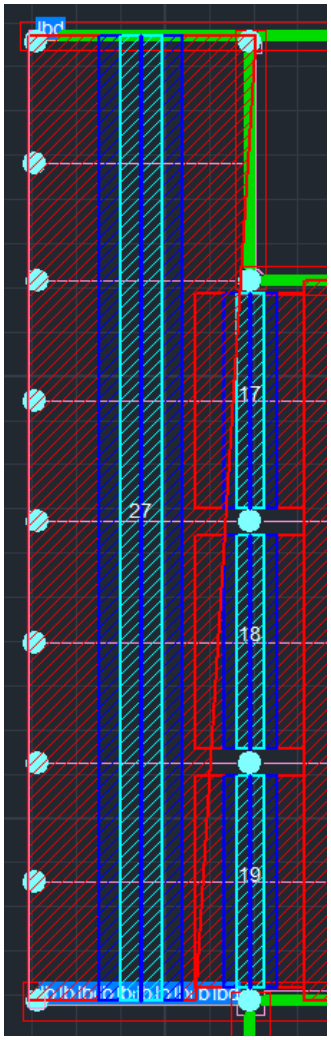
Drop Panels and Support Lines

⚠ Based on the designed Support Lines, the corresponding Design Strips will be generated

§ Instructions for inserting the support lines in flat slabs

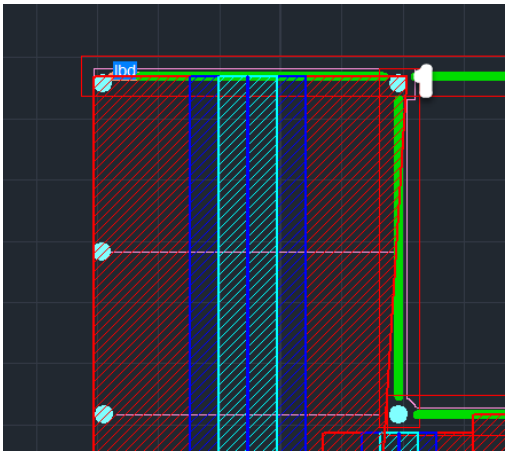
1. The support lines should start from a column and end to a column (or free end). In any case, they must contain at least one column.
2. The support line must reach the outline of the slab as long as it is a free end. Otherwise, it may stop on the outline or the node of the column.
3. If the boundary conditions (that is, right and left of the support line) change along the line, the line must break at these points

For example, in the following case

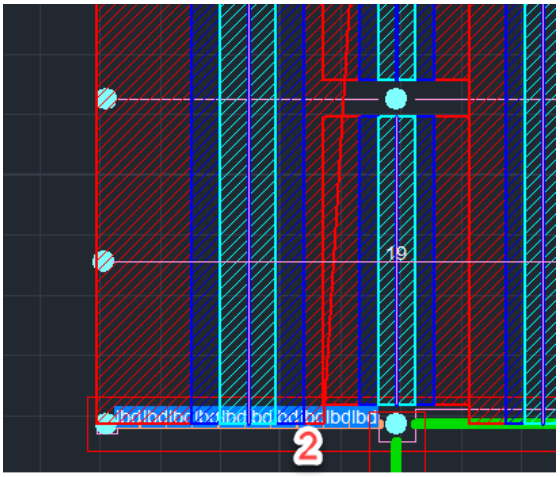


On the supported Line 27, the red right line has not formed correctly (it is the red inclined line). This happened because a support line was inserted which:

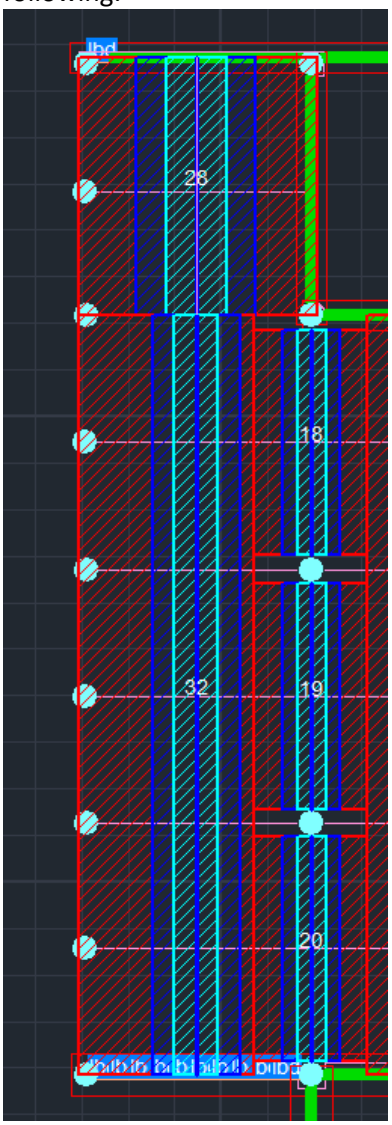
At the right top, it is bordered by beams, that is, by the slab boundary, and so the program has delineated the red out area at this slab boundary (point 1)



At the bottom, in which the right border is another support line (19) the program calculated another length of red area (point 2)



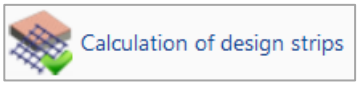
For that reason, the inclined boundary of the red area emerged.
But, in case we had inserted two support lines, one for the upper part(with the slab boundary) and one for the bottom part (boundary with the support lines 17,18,19) the correct result is the following:



In general, the above algorithm, based on the latest improvements, is suitable for slabs with normal rectangular shapes and a cannulated array of columns. In slabs with strange shapes, overlapping loading strips may occur as well as gaps between them.

Two support lines are now formed, that is, the 28 and 32.

6.2.1.2 Calculation of design strips

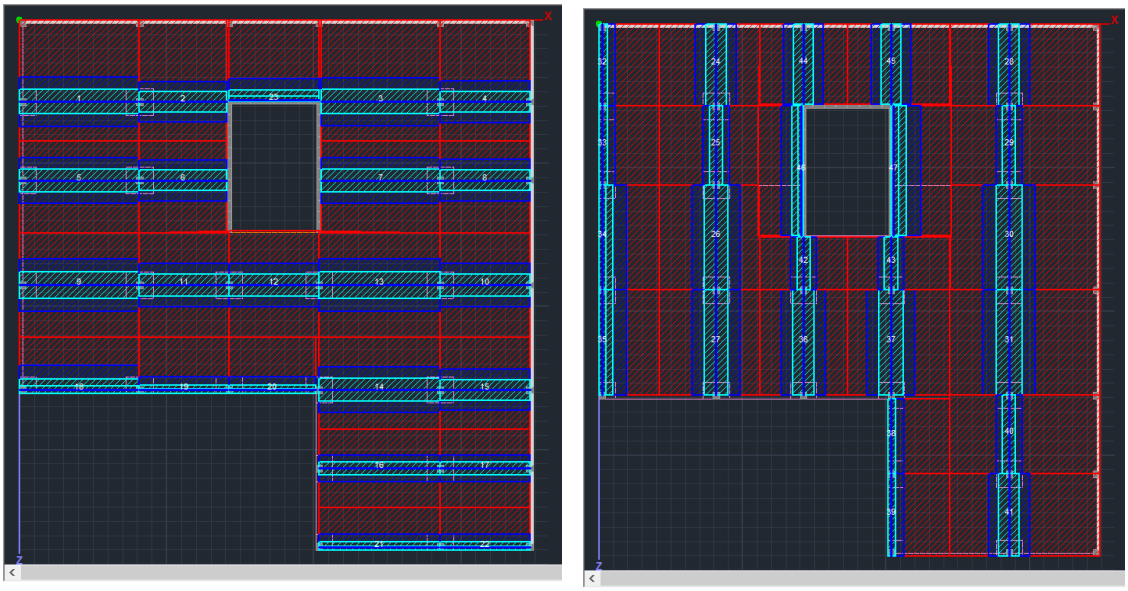


According to Annex I of EC2 flat slab is divided into design strips. These are the areas that are automatically created by the program on both sides of the Support Line, according to Figure I.1 of EC2.
Select the command Calculation of design strips and the program automatically creates them.

Each Design Strip is divided into sections along its length perpendicular to the Support Line. In each section, SCADA integrates the internal forces of finite surface elements that intersect. By completing that, the bending moment around the axis of the section occurs. This intensive value is used to calculate the armature in each section.

6.2.1.3 Display X, Z

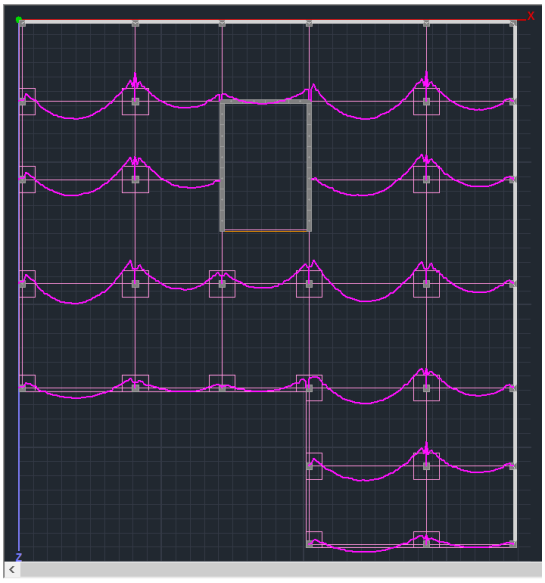
Select to display the Design Strips in both directions.

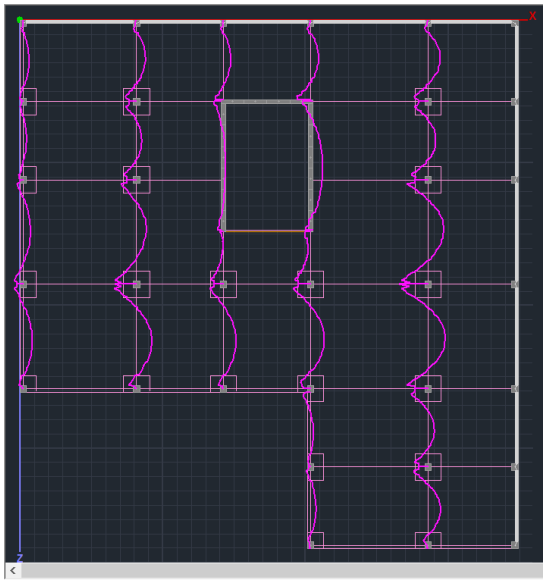


Design Strips along the X and Z axes

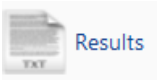
6.2.1.4 Diagrams X, Z

Select Diagrams in both directions to see the corresponding diagrams.





6.2.1.5 Results



This command opens the Results file through the Report.

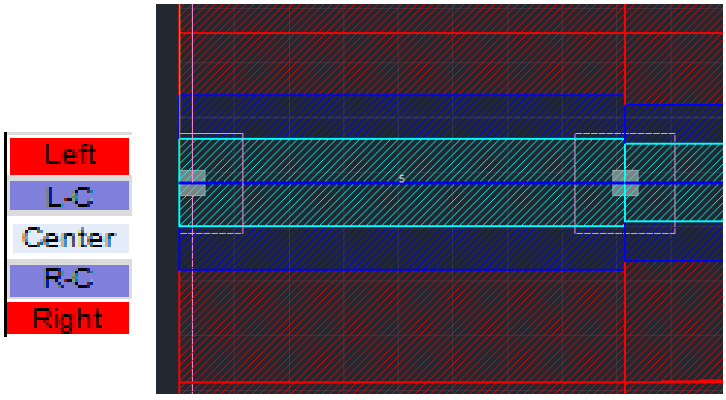
Each page concerns a Strip Line.

Initially, the characteristics of the Strip are described.

							Page : 1
Strip Calculations							
Description	**Value**	**Units**	**Code**	**Description**	**Value**	**Units**	**Code**
Floor	1			Starting point	corner column		9.4.1&2
# of strip	1			Drop panel	Yes		
Orientation	x-x			*Thickness*	182.88	(cm)	
Length	815.48	(cm)		*Width*		(cm)	
Concrete	C20/25			Finishing point	internal column		9.4.1&2
f_{ck}	20	(MPa)	Table 3.1	Drop panel	Yes		
f_{ctm}	2.20	(MPa)	Table 3.1	*Thickness*	182.88	(cm)	
Steel	S400s			*Width*		(cm)	
f_{yk}	400	(MPa)		Minimum reinforcement			
Cover	20	(mm)		*Tension reinf.*	0.00143	(cm^2/m)	9.2.1.1(1)
Slab thickness	0.25	(cm)		*Compression reinf. (% of span reinf.)*	25	%	9.3.1.2

Then the reinforcement results are displayed above and below in detail for each zone, dividing them into sub-zones.

- Left-Right -> red zone
- L-C R-C-> blue zone
- Center-> light blue zone



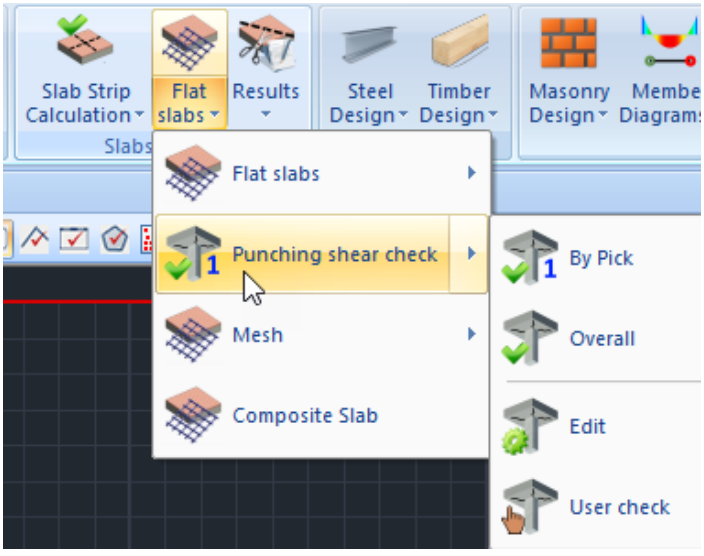
Analysis Results and Reinforcement										Top
	203.87 cm (L_{start})					407.74 cm (L_{centre})				
Zone	M (kNm)	Width (cm)	$A_{s,rqd}$ (cm^2/m)	$A_{s,prvd}$ (cm^2/m)	Φ/s	M (kNm)	Width (cm)	$A_{s,rqd}$ (cm^2/m)	$A_{s,prvd}$ (cm^2/m)	Φ/s
Left		400.0		3.246	8/15		401.1		0.812	8/20
L-C		400.0		3.246	8/15		85.3		1.763	8/20
Center	-80.283	46.0	27.271	27.271	14/5		170.5		6.818	8/7
R-C							85.3		2.043	8/20
Right							103.8		1.471	8/20

	203.87 cm (L_{end})				
Zone	M (kNm)	Width (cm)	$A_{s,rqd}$ (cm^2/m)	$A_{s,prvd}$ (cm^2/m)	Φ/s
Left	-88.070	401.1	2.873	3.246	8/15
L-C	-44.824	85.3	7.054	7.054	8/7
Center	-152.524	170.5	12.422	12.422	10/6
R-C	-51.588	85.3	8.172	8.172	8/6
Right	-45.848	103.8	5.886	5.886	8/8

Analysis Results and Reinforcement										Bottom
	203.87 cm (L_{start})					407.74 cm (L_{centre})				
Zone	M (kNm)	Width (cm)	$A_{s,rqd}$ (cm^2/m)	$A_{s,prvd}$ (cm^2/m)	Φ/s	M (kNm)	Width (cm)	$A_{s,rqd}$ (cm^2/m)	$A_{s,prvd}$ (cm^2/m)	Φ/s
Left	9.207	400.0	0.294	0.812	8/20	70.543	401.1	2.293	3.246	8/15
L-C	9.207	400.0	0.294	0.844	8/20	21.929	85.3	3.377	3.377	8/14
Center	80.591	46.0	27.408	27.408	14/5	43.857	170.5	3.377	3.377	8/14
R-C						21.929	85.3	3.377	3.377	8/14
Right						25.982	103.8	3.284	3.284	8/15

	203.87 cm (L_{end})				
Zone	M (kNm)	Width (cm)	$A_{s,rqd}$ (cm^2/m)	$A_{s,prvd}$ (cm^2/m)	Φ/s
Left	17.335	401.1	0.555	0.812	8/20
L-C	6.505	85.3	0.984	0.984	8/20
Center	23.135	170.5	1.762	1.762	8/20
R-C	3.583	85.3	0.539	0.844	8/20
Right	3.311	103.8	0.409	0.821	8/20

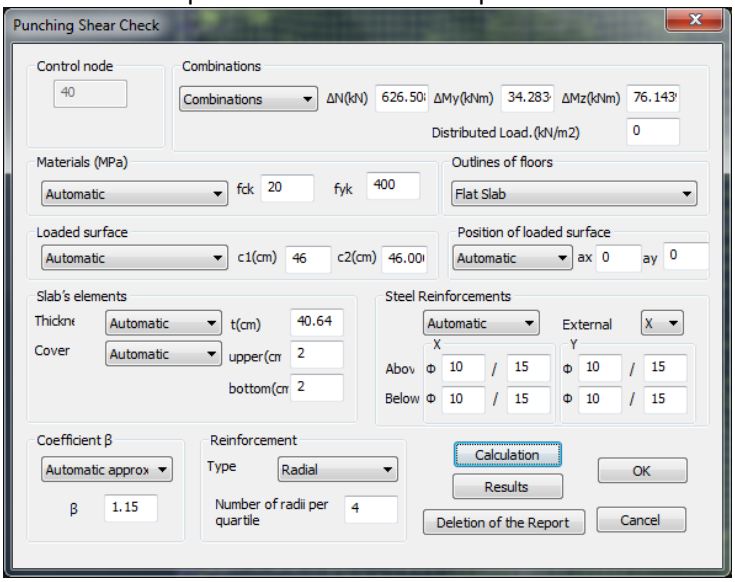
6.2.2 Punching shear checks

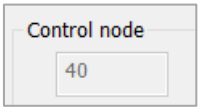


The punching checks are also added in the new version by EC2. The check is part of the check and design process of the flat slabs which runs automatically, but can also run as an individual check for any column. All data can be set automatically or even manually.

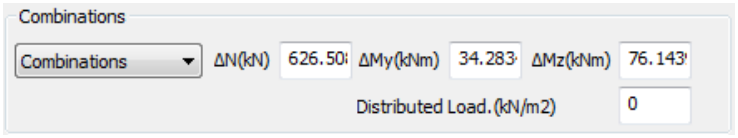
6.2.2.1 Pick

By Selecting “By Pick” command with the left mouse button point the node of a column and right-click to open the dialog box where you specify all the necessary parameters.
Here are the explanations for the other positions of the columns on the perimeter of the slab:





The number of the selected node is filled in automatically and is not editable.

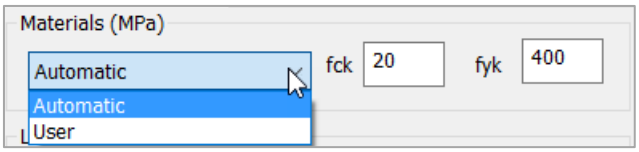


In Combinations field:

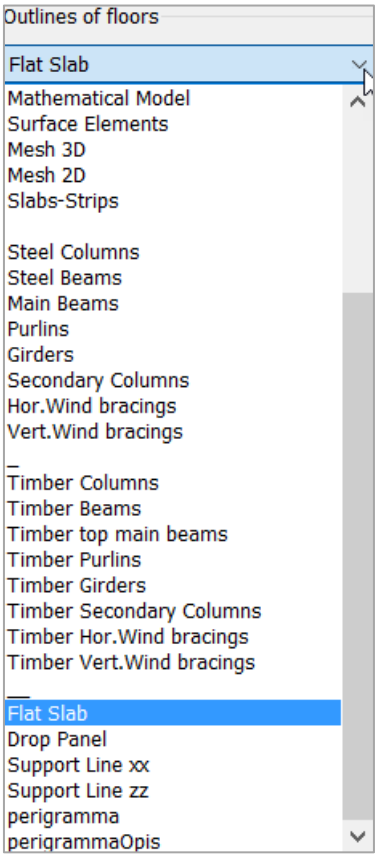
- By choosing Combinations, the program finds automatically the combination that gives the worst Axial ΔN resulting and displays its value along with the corresponding moments.
- Choosing User, allows user values to be set for axial force and moments, in their respective fields, as well as the definition of a Distributed Load

that works "relieving" the slab at that point, so the shear calculation be impaired relative to the original.

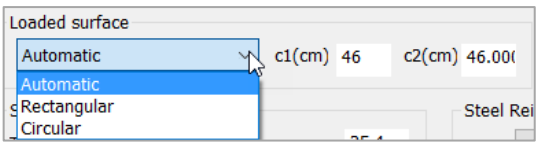
Shear force ($V_{Ed,init.}$)	626.5	(kN)
Distributed load (p)	50.0	(kN/m^2)
Reduced shear force ($V_{Ed,fin.}$)	478.8	(kN)



In the field Material, coefficients fck and fyk are filled in automatically with the Automatic option or defined by the user with the User option.



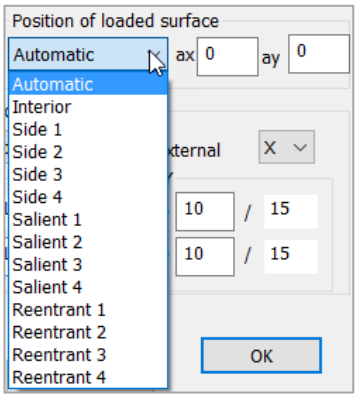
In Outlines of Floors, choose the Layer containing the contour of the slab including the columns located in the outline.
Then select the layer Flat Slab that includes exactly the lines that define the overall contour of the slab.



As the loaded surface is defined the equivalent surface of the selected column.

By choosing:

- Automatic, the program calculates the surface of any column-shaped by a reduction in the equivalent rectangular with the corresponding dimensions c1 and c2.
- Rectangular, the user defines his dimensions c1 and c2 for calculating the loaded rectangular surface
- Circular, the user defines his diameter c1 for calculating the loaded circular surface

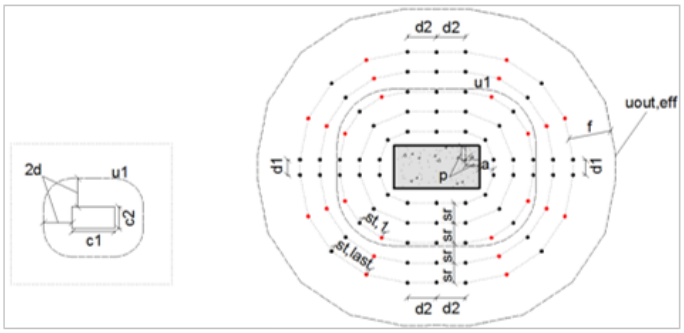


The Position of the loaded surface may be determined either automatically or selectively. It depends on the contour of the slab and the position of the selected column in it.

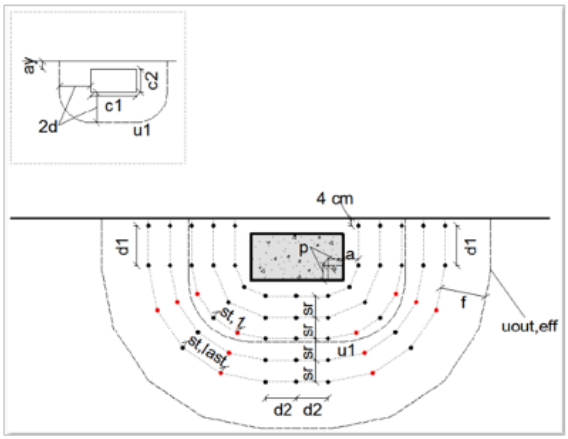
The proposed positions are:

- Interior
- Side: 4 directions
- Salient: 4 directions
- Reentrant: 4 directions

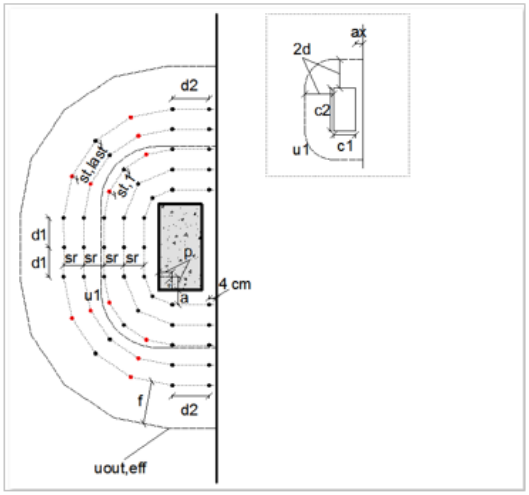
Select the position of the selected column and set the distances from the perimeter, ax, and ay, (except indoor) according to the following figures:



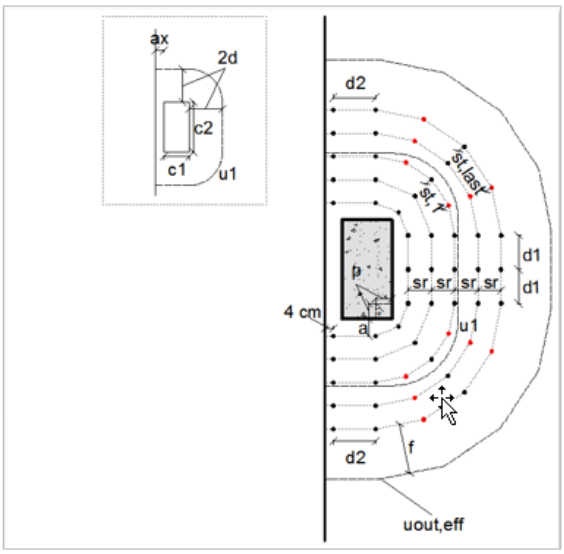
INTERIOR



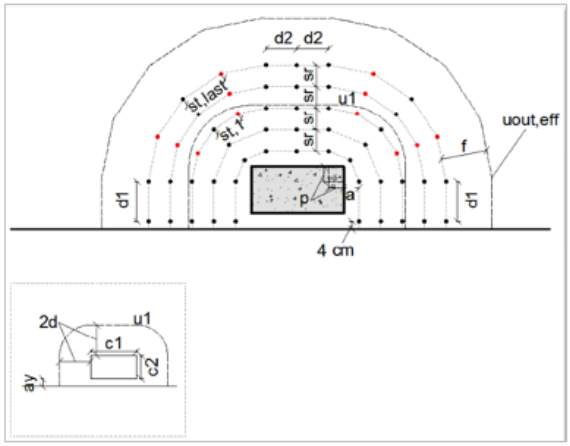
SIDE 1



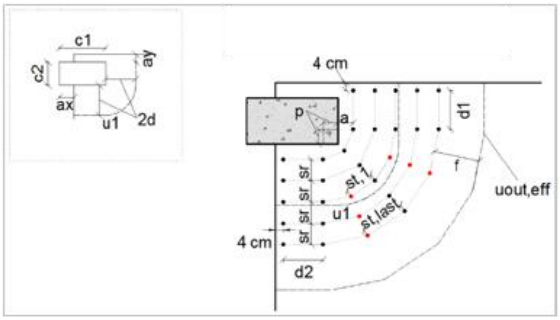
SIDE 2



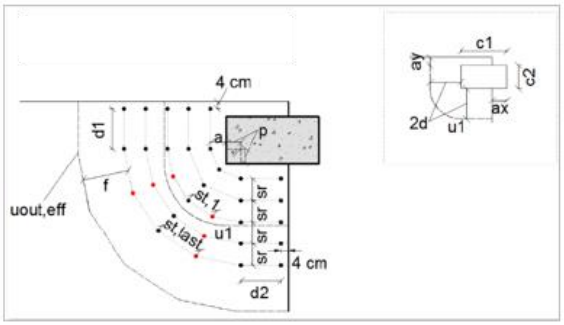
SIDE 4



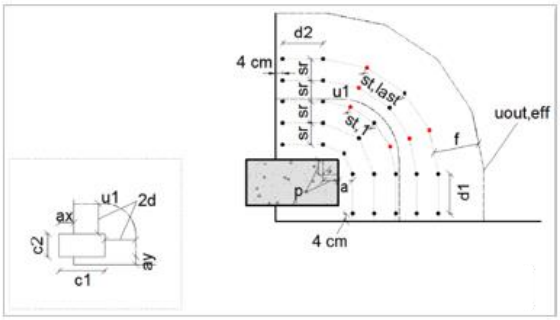
SIDE 3



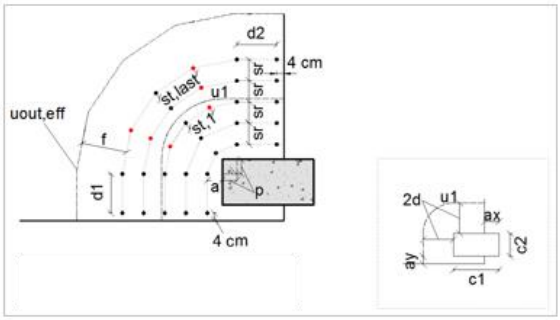
SALIENT 1



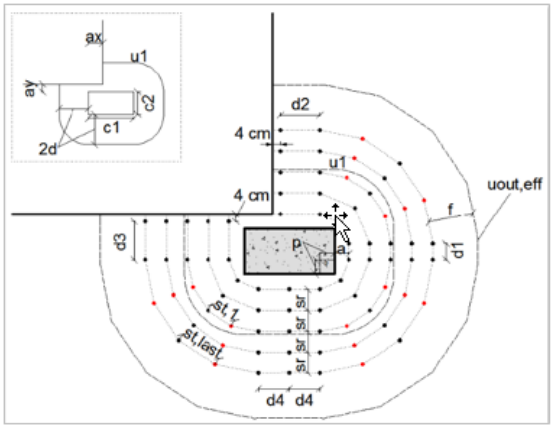
SALIENT 2



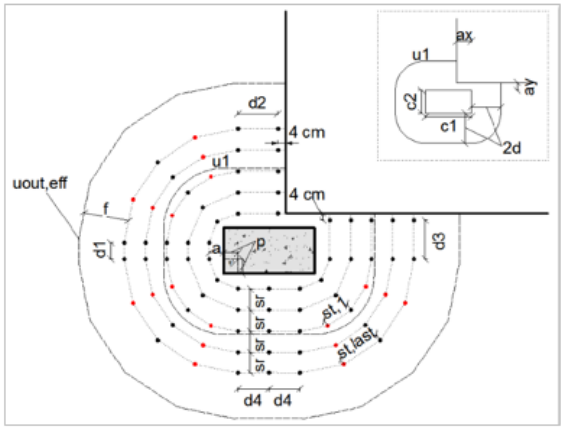
SALIENT 4



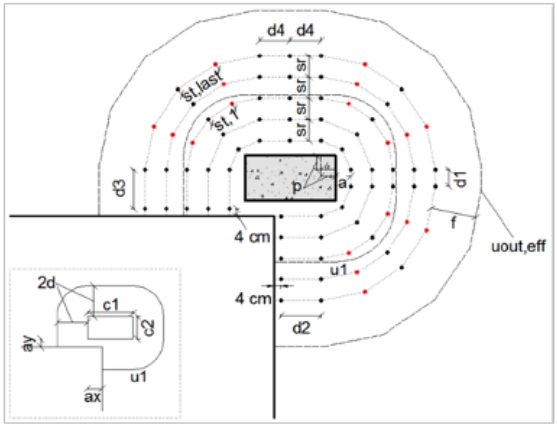
SALIENT 3



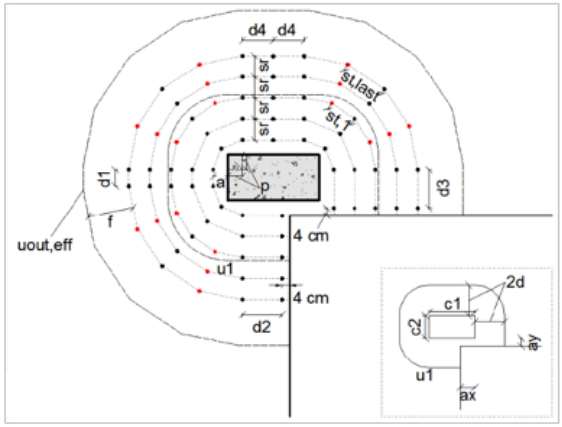
REENTRANT 1



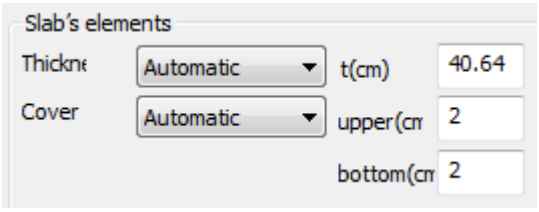
REENTRANT 2



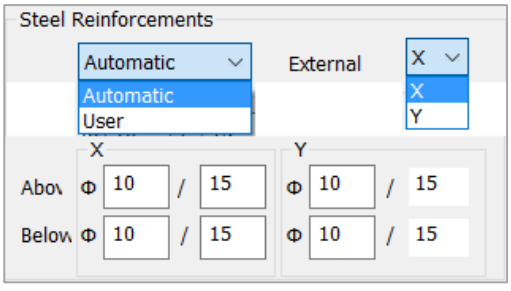
REENTRANT 4



REENTRANT 3



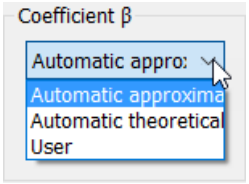
The Thickness and Cover of the Slab either considered Automatically or modified by the User, by typing the corresponding values for Thickness and Cover, upper and bottom, of the slab.



In Steel Reinforcements the longitudinal reinforcement resulting from the calculation of flat slabs in the selected column region is identified.
With the Automatic option taking into account the longitudinal reinforcement:

- Above for ΔN positive (+ΔN) (e.g. penthouse slab)
- Below for ΔN negative (-ΔN) (e.g. foundation slab)

By selecting External X or Y, the direction of the outer reinforcement of the slab longitudinal reinforcement mesh (either for the up or down mesh) is determined.



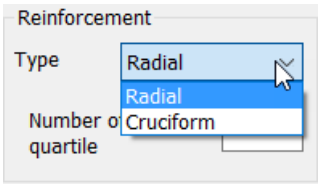
The Coefficient β for punching calculation can be calculated automatically in two ways:

- Automatic approximation or
- Automatic theoretical.

⚠ The Approximation way is a function of the position of the loaded area and the ax, ay.

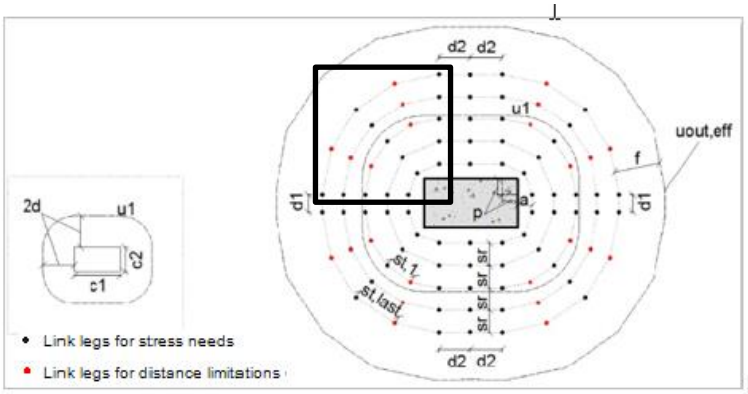
⚠ The Theoretical way is a function of the Moments My, Mz.

User selection allows the introduction of any value for the coefficient β.

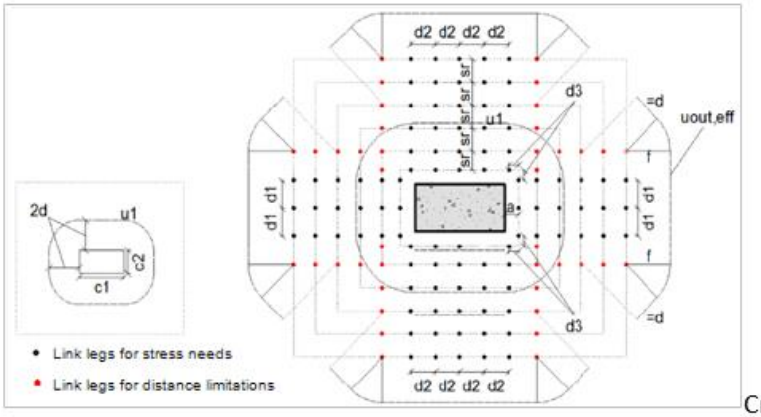


Finally, in Reinforcement set the layout type of punching reinforcement choosing between Radial and Cruciform.
For the Radial layout set the number of radii per quartile.

In the figure below the black box contains one quartile of the radial layout. The first perimeter has three radii of reinforcement, while there is an increase on the 3rd perimeter to 5 radii due to distance limitations set by the code. SCADA Pro automatically checks whether the distance limitations are satisfied on the first perimeter, and increases the number of radii when needed (even when the user's choice is not enough for the first perimeter).



Radial reinforcement Layout



Cruciform reinforcement Layout

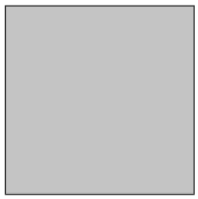
Calculation The Calculation command performs all the necessary checks for punching, taking into account all the above parameters.

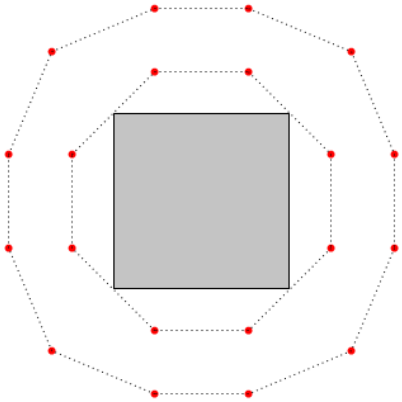
Results This command displays the results file:

Input Data					
Description	**Value**	**Units**	**Description**	**Value**	**Units**
Level - Storey	1		Eccntr. factor (β) (EC2-6.4.3)	1.150	
# of node	40		Slab depth	40.6	(cm)
Combination	1		Cover of reinforcement	2.0	(cm)
Shear force ($V_{Ed,int.}$)	626.5	(kN)	Bar size (outer layer)	10	(mm)
Distributed load (p)	0.0	(kN/m^2)			
Reduced shear force ($V_{Ed,fin.}$)	626.5	(kN)	Spacing of bars (outer layer)	15.0	(cm)
Bending moment (M_x)	34.3	(kNm)	Bar size (second layer)	10	(mm)
Bending moment (M_y)	76.1	(kNm)			
Shape of loaded area	Rectangular		Spacing of bars (second layer)	15.0	(cm)
c_x length (along x axis)	46.0	(cm)			
c_y length (along y axis)	46.0	(cm)	Concrete (f_{ck})	20.0	(MPa)
c diameter		(cm)	Steel (f_{yk})	400.0	(MPa)
Position of loaded area	Interior		Reinforcement pattern	Radial	
Dist. of slab perim. along x (a_x)		(cm)	# of radii of reinforcement in a quadrant (circular pattern)	2	
Dist. of slab perim. along y (a_y)		(cm)			

Input Data: list of all elements identified in the previous window and required to punching check.

Punching Reinforcement Layout: according to predetermined parameters and if, there is reinforcement requirement.





In **Check results** there are two different checks:

Page : 2

Check results

Description	Value	Units	EC2	Description	Value	Units	EC2
Effective depth of slab (d)	37.6	(cm)	(eq6.32)	Basic control perimeter (u_1)	657.0	(cm)	(fig6.15)
Perimeter of the loaded area (u_0)	184.0	(cm)	(eq6.53)	Design value of the shear stress at u_1 ($v_{Ed,1}$)	0.291	(MPa)	(eq6.38)
Design value of the shear stress at u_0 ($v_{Ed,0}$)	1.040	(MPa)	(eq6.38)	Punch. shear resistance without shear reinforcement ($v_{Rd,c}$)	0.356	(MPa)	(eq6.47)
Maximum punching shear resistance ($v_{Rd,max}$)	3.680	(MPa)	(eq6.53)	Constant (v_{min})	0.356	(MPa)	(eq6.3)
1st check: $v_{Ed,0} \le v_{Rd,max}$	Sufficiency			2nd check: $v_{Ed,1} \le v_{Rd,c}$	No reinforcement is required		

If the 1st check presents *Sufficiency*, then the 2nd check specifies the requirement or not of punching reinforcement.

Page : 2

Check results

Description	Value	Units	EC2	Description	Value	Units	EC2
Effective depth of slab (d)	37.6	(cm)	(eq6.32)	Basic control perimeter (u_1)	657.0	(cm)	(fig6.15)
Perimeter of the loaded area (u_0)	184.0	(cm)	(eq6.53)	Design value of the shear stress at u_1 ($v_{Ed,1}$)	0.744	(MPa)	(eq6.38)
Design value of the shear stress at u_0 ($v_{Ed,0}$)	2.657	(MPa)	(eq6.38)	Punch. shear resistance without shear reinforcement ($v_{Rd,c}$)	0.356	(MPa)	(eq6.47)
Maximum punching shear resistance ($v_{Rd,max}$)	3.680	(MPa)	(eq6.53)	Constant (v_{min})	0.356	(MPa)	(eq6.3)
1st check: $v_{Ed,0} \le v_{Rd,max}$	Sufficiency			2nd check: $v_{Ed,1} \le v_{Rd,c}$	Reinforcement necessary: - add punching shear reinforcement - increase slab longitudinal reinforcement		

Page : 2

Check results

Description	Value	Units	EC2	Description	Value	Units	EC2
Effective depth of slab (d)	37.6	(cm)	(eq6.32)	Basic control perimeter (u_1)	657.0	(cm)	(fig6.15)
Perimeter of the loaded area (u_0)	184.0	(cm)	(eq6.53)	Design value of the shear stress at u_1 ($v_{Ed,1}$)	1.860	(MPa)	(eq6.38)
Design value of the shear stress at u_0 ($v_{Ed,0}$)	6.642	(MPa)	(eq6.38)	Punch. shear resistance without shear reinforcement ($v_{Rd,c}$)	0.356	(MPa)	(eq6.47)
Maximum punching shear resistance ($v_{Rd,max}$)	3.680	(MPa)	(eq6.53)	Constant (v_{min})	0.356	(MPa)	(eq6.3)
1st check: $v_{Ed,0} \le v_{Rd,max}$	Insufficient. - increase size of the loaded area - increase slab depth - use of concrete of a higher quality			2nd check: $v_{Ed,1} \le v_{Rd,c}$			

If the 1st check presents *Insufficiency*, then the 2nd check is not performed and some interventions for the slab are proposed.

Detailing results

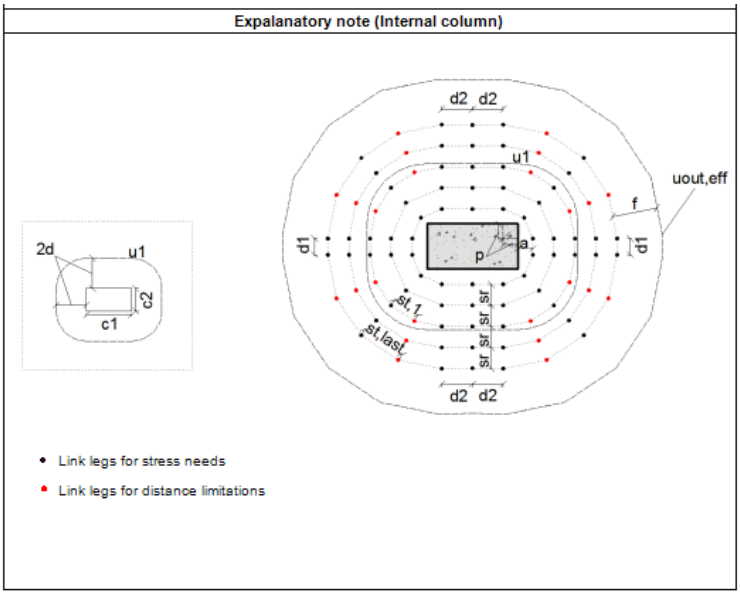
Description	Value	Units	EC2	Περιγραφή	Τιμή	Μονάδες	EC2
Perimeter $u_{out,ef}$	483.7	(cm)	(eq6.54)	Distance (d_3)		(cm)	
(a) - Distance of 1st perimeter of reinforcement from the loaded area	11.2	(cm)		Distance (d_4)		(cm)	
Limit: $0.3 \cdot d \le a \le 0.5 \cdot d$	6.7<=a<=11.2		(9.4.3)	Angle (φ)	90.0	°	
(f) - Distance of last perimeter of reinforcement from $u_{out,ef}$	29.7	(cm)		($s_{r,last}$) - Tangential distance between link legs on the last perimeter	54.4	(cm)	
Limit: $k \cdot d = 1.5 \cdot d$	33.6	(cm)	(6.4.5)	Limit: $2.0 \cdot d$	44.8	(cm)	
(s_r) - Radial distance of the perimeters of reinforcement	16.5	(cm)		($f_{ywd,ef}$) - Effective design strength of punching shear reinf.	306.0	(MPa)	(eq6.52)
Limit: $0.75 \cdot d$	16.8	(cm)	(9.4.3)	($A_{sw,1}$) - Necessary area of a link leg	0.535	(cm^2)	
($s_{r,1}$) - Tangential distance between link legs on the u_1 perimeter	54.4	(cm)		($A_{sw,min}$) - Minimum area of a link leg	0.535	(cm^2)	(eq9.11)
Limit: $1.5 \cdot d$	33.6	(cm)	(9.4.3)	Diameter of link leg chosen	10	(mm)	
Distance (p)	10.8	(cm)					
Distance (d_1)	24.5	(cm)		Area of link leg chosen	0.785	(cm^2)	
Distance (d_2)	24.5	(cm)					

In **Detailing results** the results from the two checks by their respective formulas and chapters of EC2 are listed in detail.

Grouping of punching shear reinforcement

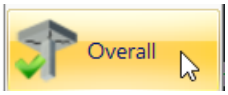
Group	Number of lines	Φ (mm)	Number of link legs on line	Height of link leg (cm)	Perimeter where the 1st link leg of the line stands	Distance of the 1st link leg from the loaded area
1	8	10	2	21.4	1	11.20
2	4	10	1	21.4	2	27.70

In **Grouping of punching shear reinforcement** the posted punching reinforcement resulting from the above controls, and the characteristics of the layout are listed.



The 3rd page shows the Legend containing characteristics by the position of the Loaded Surface.

6.2.2.2 Overall

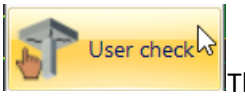


This is the command for making the punching checks in all columns included in the outline of the flat slab, automatically, using the parameters automatically from the program. The same parameters are used for all columns.

6.2.2.3 Edit

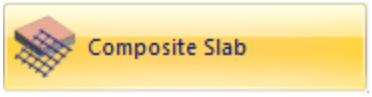
The Edit command allows you to change the parameters set by the “By Pick” check or “Overall” check.
Select the command and left-click on the node of the column to edit. Automatically the previous parameters window opens with the original settings for the selected column. You can make any changes you want and repeat the check using the command Calculate.

6.2.2.4 User check



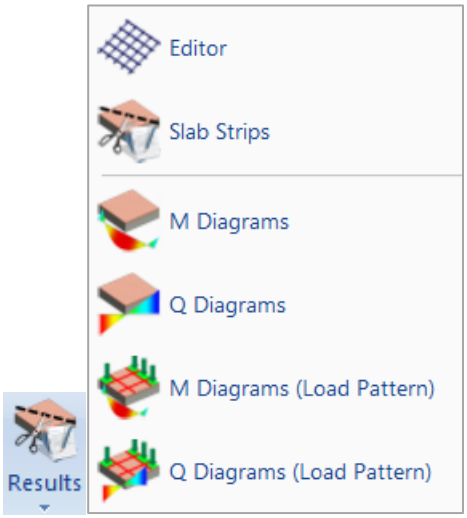
The user has the opportunity to test different data on different nodes for a supervisory result image. This is a "draft" that is not saved in the printout but allows the user to make tests to reach the desired solution.

6.2.3 Composite Slabs



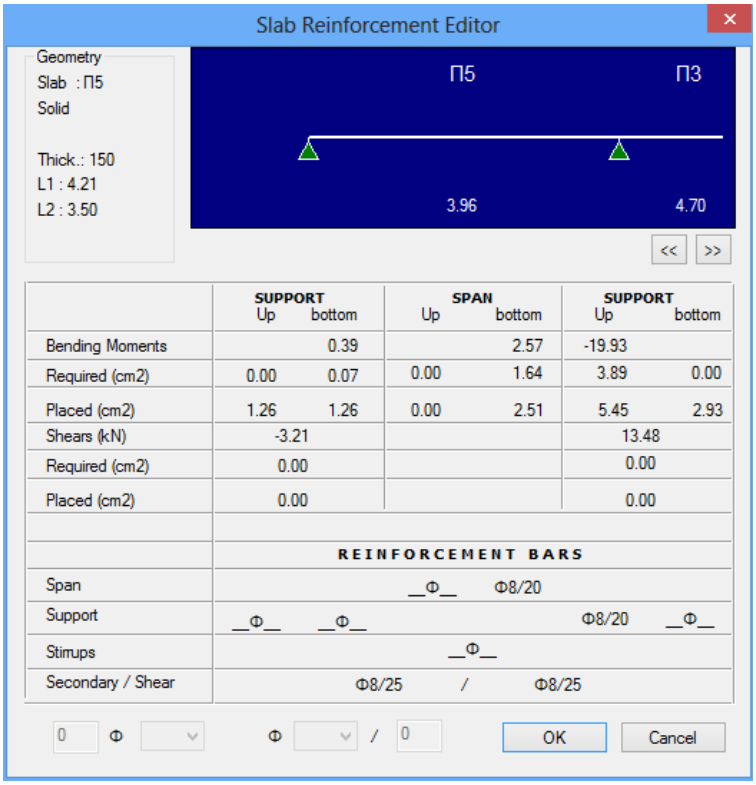
6.3 Results

This command contains a command list about editing and viewing the design checks’ results.



6.3.1 Editor


It’s an editing tool used to modify the results of the slabs’ design.
Select the command and left click on an already designed strip.
A dialog box is displayed:



At the top left, you see the slab data such as name, size, etc. The dimensions are measured from axis to beam axis. Then the bending moments calculated for the supports and the corresponding span and the square centimeters of reinforcement calculated at the corresponding points are listed.

	SUPPORT Up	bottom	SPAN Up	bottom	SUPPORT Up	bottom
Bending Moments		0.39		2.57	-19.93	
Required (cm2)	0.00	0.07	0.00	1.64	3.89	0.00
Placed (cm2)	1.26	1.26	0.00	2.51	5.45	2.93

The next section includes shear calculation and the corresponding required square centimeters.

Shears (kN)	-3.21		13.48
Required (cm2)	0.00		0.00
Placed (cm2)	0.00		0.00

The last section includes the corresponding reinforcing bars.

	REINFORCEMENT BARS
Span	__Φ__ Φ8/20
Support	__Φ__ __Φ__ Φ8/20 __Φ__
Stirups	__Φ__
Secondary / Shear	Φ8/25 / Φ8/25

When arming, the minimum reinforcement designated parameters are also taken into account.

Listed in order: span and supports reinforcements, stirrups for Zoellner slabs, as well as the secondary and shear reinforcement.
SCADA Pro always calculates the main reinforcement parallel to the strip direction and in the other direction calculates the secondary and shear reinforcement, regardless of how bending (in one or both directions).

To modify the calculated bars, click on the corresponding size, 

and automatically activate the field at the bottom of the dialog box where you can select the new diameter and enter the new distance 

.

6.3.2 Slab Strips


Select to display the TXT file that contains the design checks for slabs and the corresponding results.
Select the command and left click on a strip to open the TXT file and read the results.
The TXT files presented, are generated by the program for the printout.

```
+-------------------------------------------------------------------------------+
|B E N D I N G  VERIFICATION   |      -F5(EDGE)|   F5 (SPAN)    |      F5-(EDGE)|
|------------------------------+--Top----Bot.--+--Top----Bot.--+--Top----Bot.--|
|Bending Moment         MSd(KNM)|  1.19 |       | 17.20 |       |  1.19 |       |
|REQ. REINFORCEMENT   As (CM2) |  0.00 |  0.16 |  0.00 |  2.34 |  0.00 |  0.16 |
+-S H E A R   VERIFICATION-----+---------------+---------------+---------------+
|Applied Shear force  VEd (KN) |     11.78    1|               |     11.78    1|
|Resist.without reinf VRd,c(KN)|     82.94     |               |     82.94     |
|Compressed.trut res.VRdmax(KN)|    575.56     |               |    575.56     |
|REQ.ADDIT.SUPPORT BARS  (CM2) |      0.00     |               |      0.00     |
|FINAL REINFORCEMENT  As (CM2) | 1.26  | 1.26  | 0.00  | 2.51  | 1.26  | 1.26  |
+------------------------------+---Φ---+---Φ---+---Φ---+---Φ---+---Φ---+---Φ---+
|FINAL REINFORCEM.  BARS       |       |       |       | Φ8 /20|       |       |
+------------------------------+---------------+---------------+---------------*
```

NOTES:

Respect the older versions, changes have also been made in the presentation of the reinforcement results.
Titles above “Tension” – “Compression” has changed to "Up" - "Down" and determine the position of the reinforcements on the slab.
Now only one moment value is written, the sign of which determines whether the armature will be inserted up or down

- For a positive moment value, the tension strength is below and the reinforcement is entered in the bottom.
- For a negative moment value, the tension strength is up and the reinforcement is entered on the top.

It is possible to have a positive value, mainly on the support, and a requirement for a compressive reinforcement, in which case the required reinforcements goes both up and down.

⚠ *Especially for the Polish scenario, the slabs reinforcements are considered to be straight. This means that half of the reinforcement on the support is not considered, and where required, more support reinforcement is inserted.*

NEW FEATURES:

In the newest SCADA Pro version, the slabs Deflection Control is also included.
Deflection control based on 7.4.2 and 7.4.3 of EC2 is presented at the end of the results of each slab. The results of the two checks are displayed separately.

DEFLECTION CONTROL (EC2 7.4.2 & 7.4.3)

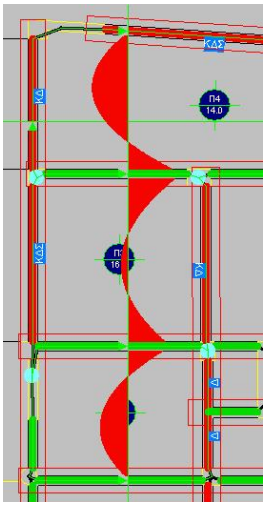
l/d	l/d perm.	Suf.	Suggested.min thick. hs(mm)	Max. M (kNm)	dul (mm)	a	l/a (perm.) (mm)	Suf.
34.59	80.10	YES	77	-7.64	0.42	250	18.40	YES

From the first check a minimum recommended thickness results, but that can not be proposed for the initial slab recognition, because its reinforcement is required to calculate it.
The calculation of the sizes of the first check does not involve intensive forces, while the second check considers the serviceability combination (s).

⚠ *Red sign marks on slab symbol when the thickness is less than permitted by regulation*

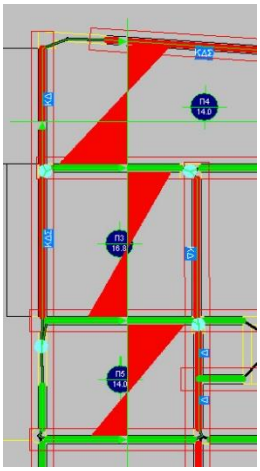
6.3.3 M Diagrams

Select the command to display a qualitative representation of the moment diagrams over the slab strip, selected with a left click. The moment diagrams result from the load combination 1.35G+1.50Q, after the multiplication with qx and qz, for strips parallel to x or z axes, respectively.

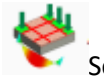


6.3.4 Q Diagrams


Select the command to display a qualitative representation of shear diagrams over the slab strip, selected with a left click. The shear diagrams result from the load combination 1.35G+1.50Q, after the multiplication by qx and qz, for strips parallel to x or z axes, respectively.



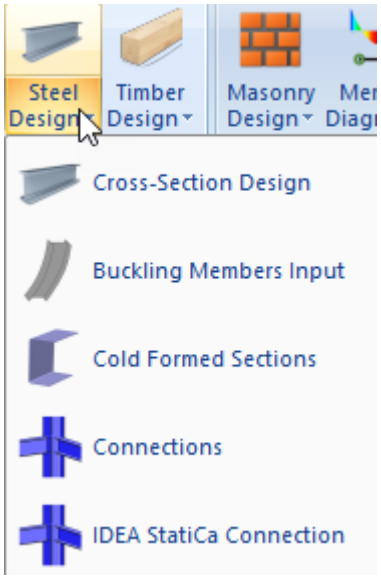
6.3.5 M Diagrams (Load Pattern)


Select the command to display a qualitative representation of the moment diagrams over the slab strip, selected with left click, resulting from a load pattern.

6.3.6 Q Diagrams (Load Pattern)


Select the command to display a qualitative representation of shear diagrams over the slab strip, selected by left click, resulting from a load pattern.

7. Steel



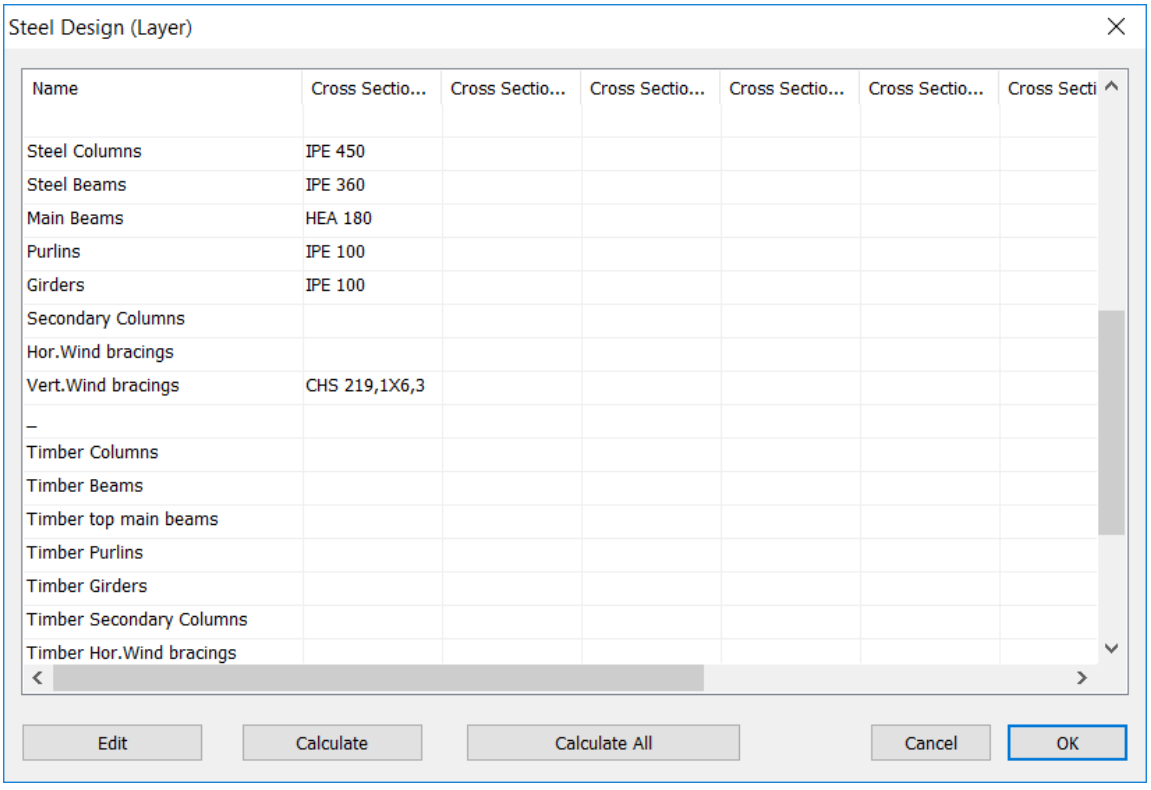
“Steel Design” command group contains commands for the cross-sections design, the buckling resistance, and the steel connections design.

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

7.1.1 Cross Section Design

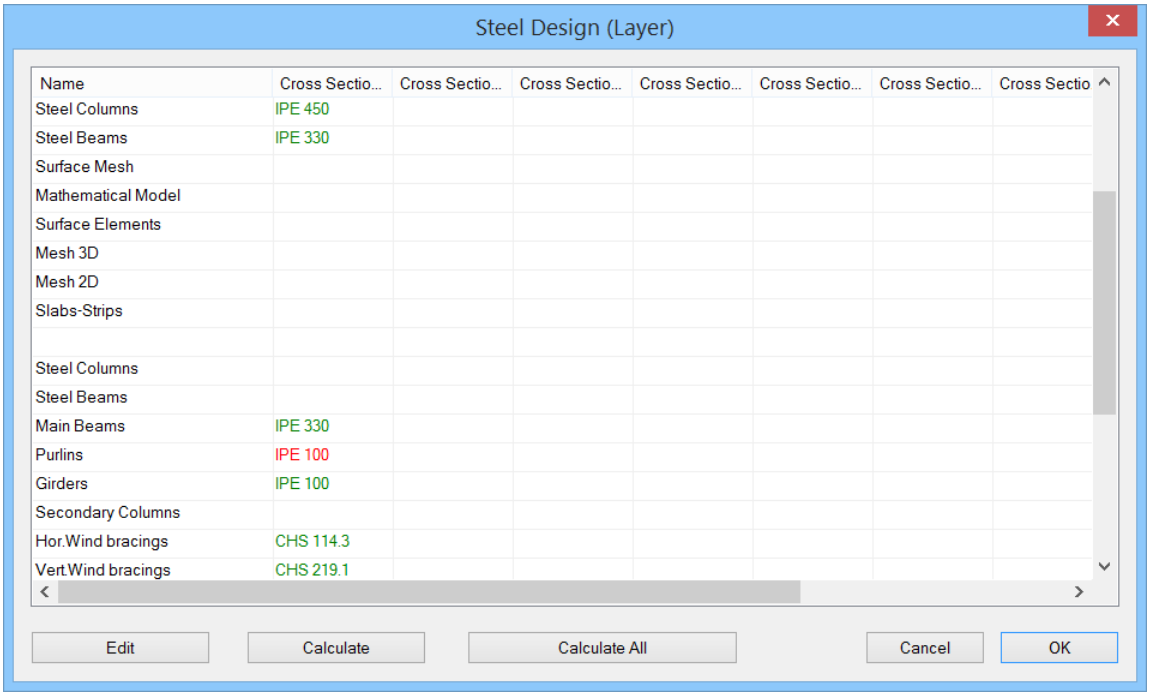
This command is used to check the adequacy of the steel cross-section.

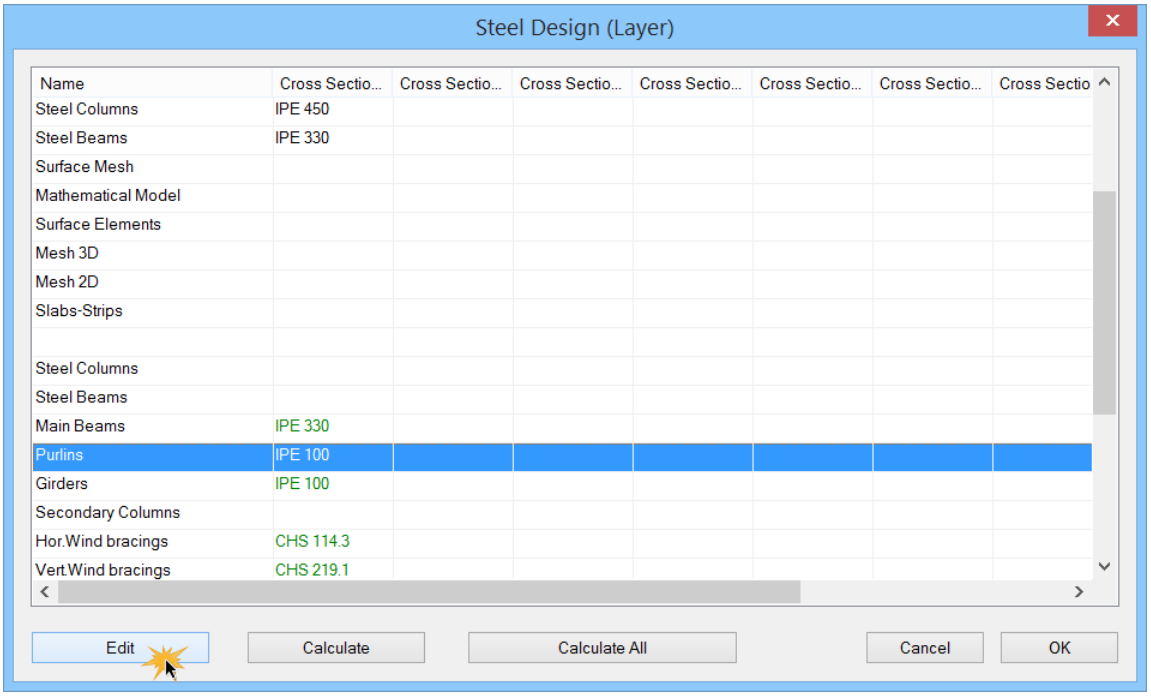
Select this command to open the following dialog box:



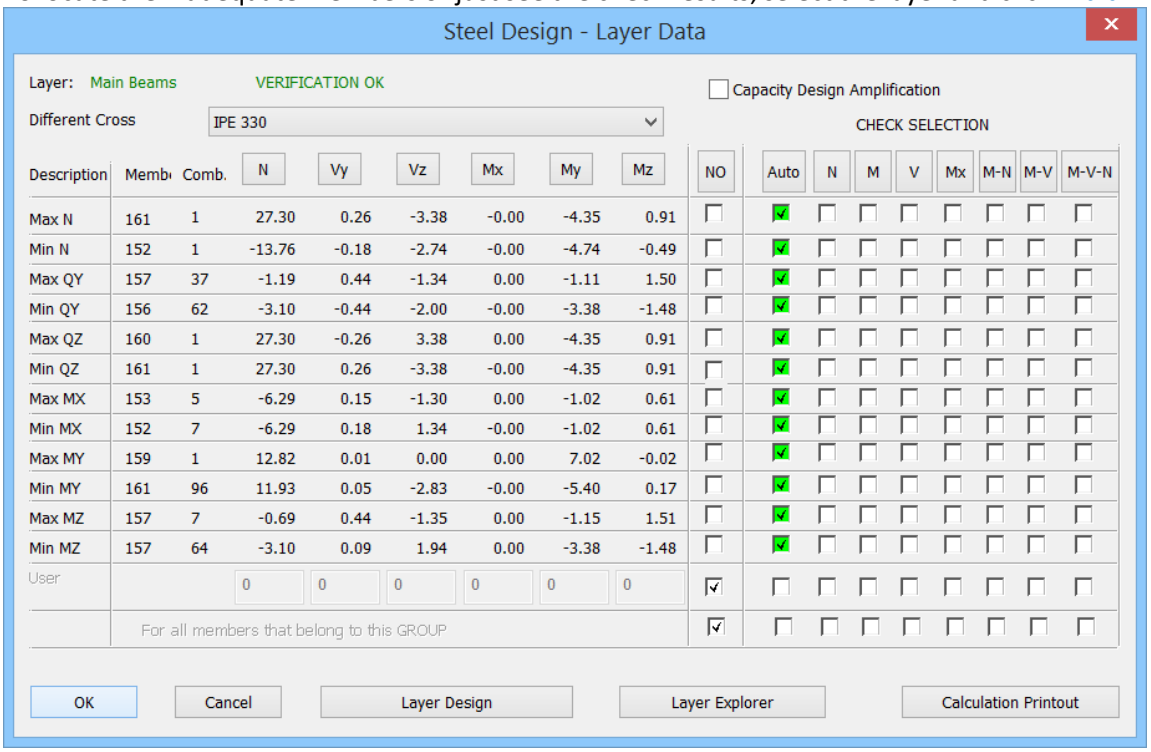
The first column contains the layers of the current project and the other columns the cross-sections that belong to each layer.

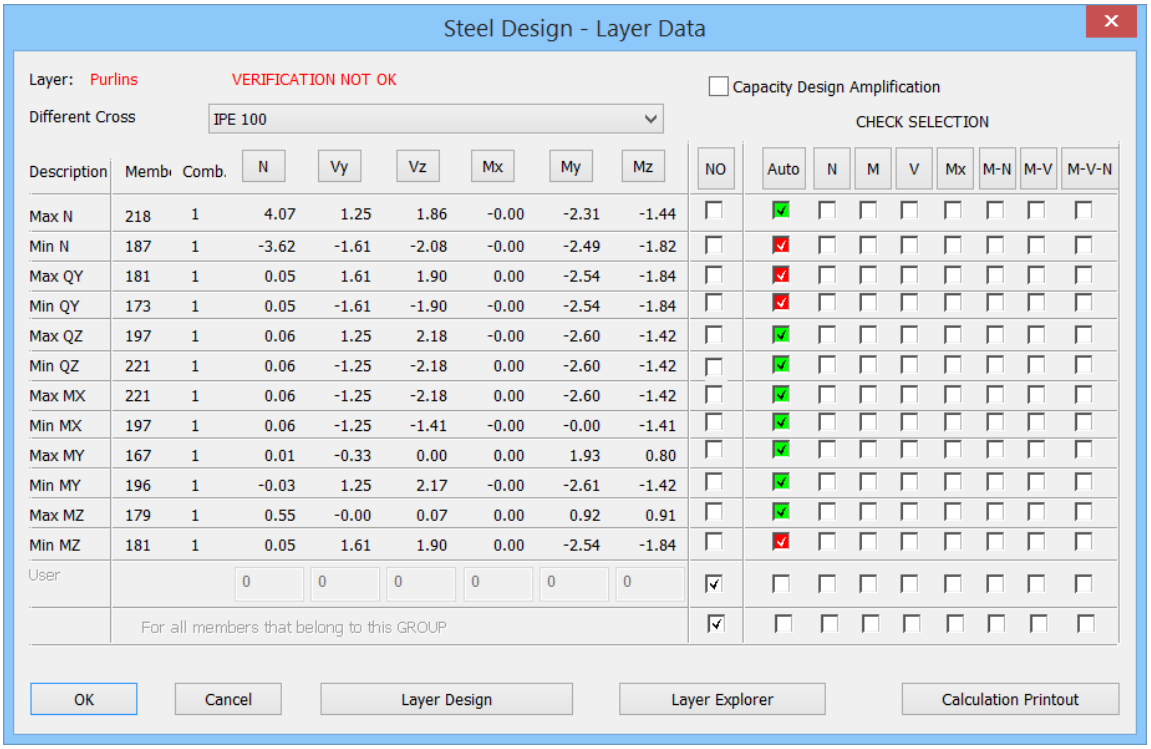
- Select the button “Calculate All” for the calculation of all sections.
- Alternatively, select the layers one by one and then click the button “Calculate”.





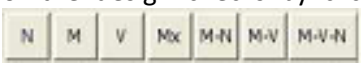
Green color indicates that all sections of this layer satisfy the design criteria (stress/resistance ≤1) red color that they don’t.
To locate the inadequate members or just see the check results, select the layer and click “Edit”.





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 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 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.

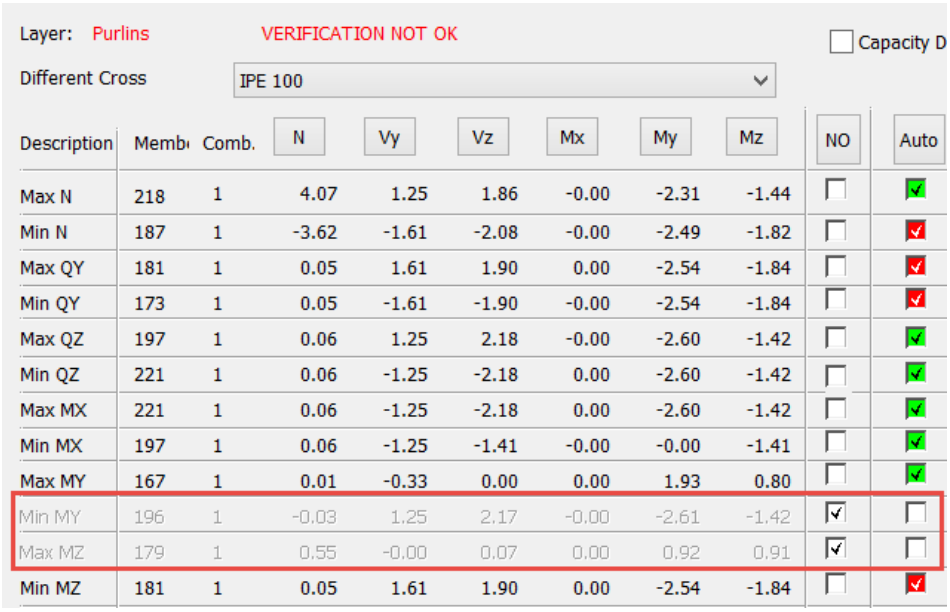
MORE DETAILS:

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 six 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.

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.



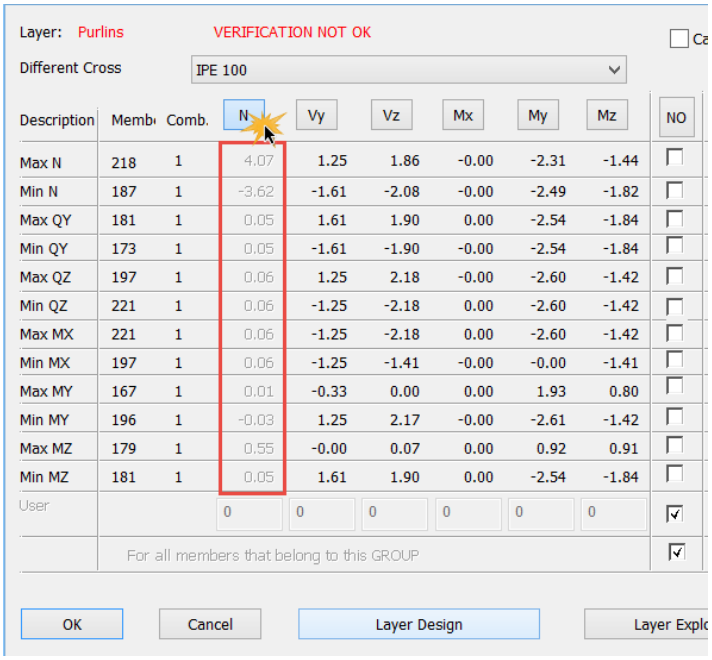
Layer: Purlins VERIFICATION NOT OK Capacity D

Different Cross IPE 100

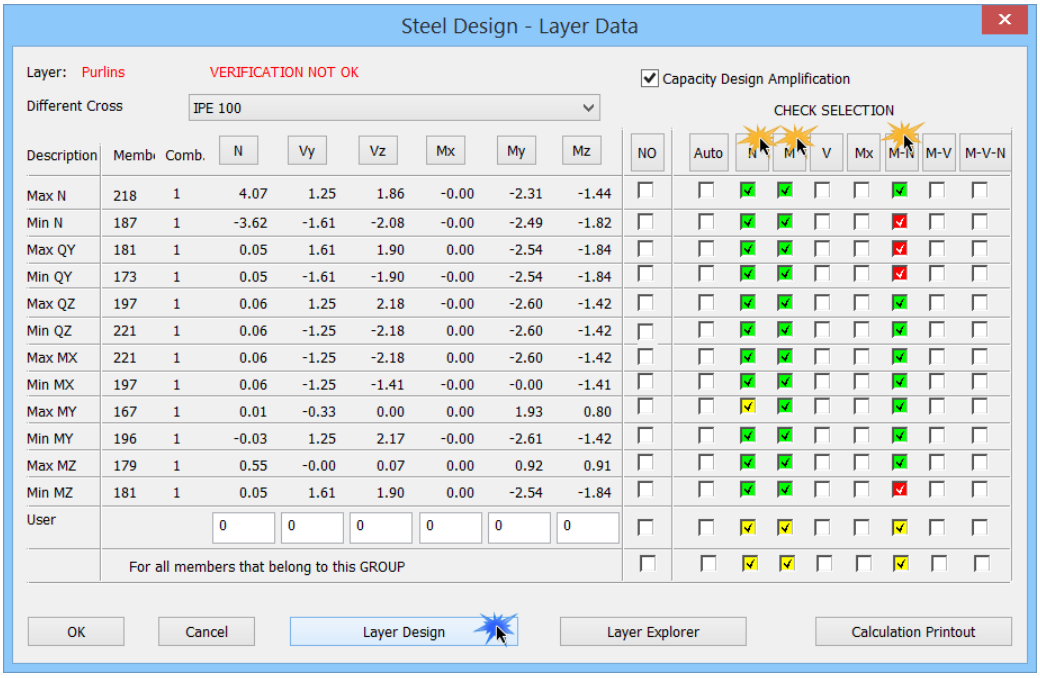
Description	Memb	Comb.	N	Vy	Vz	Mx	My	Mz	NO	Auto
Max N	218	1	4.07	1.25	1.86	-0.00	-2.31	-1.44	☐	☑
Min N	187	1	-3.62	-1.61	-2.08	-0.00	-2.49	-1.82	☐	☑
Max QY	181	1	0.05	1.61	1.90	0.00	-2.54	-1.84	☐	☑
Min QY	173	1	0.05	-1.61	-1.90	-0.00	-2.54	-1.84	☐	☑
Max QZ	197	1	0.06	1.25	2.18	-0.00	-2.60	-1.42	☐	☑
Min 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 

) and again “Layer Design” to receive the new results without considering the axial forces.



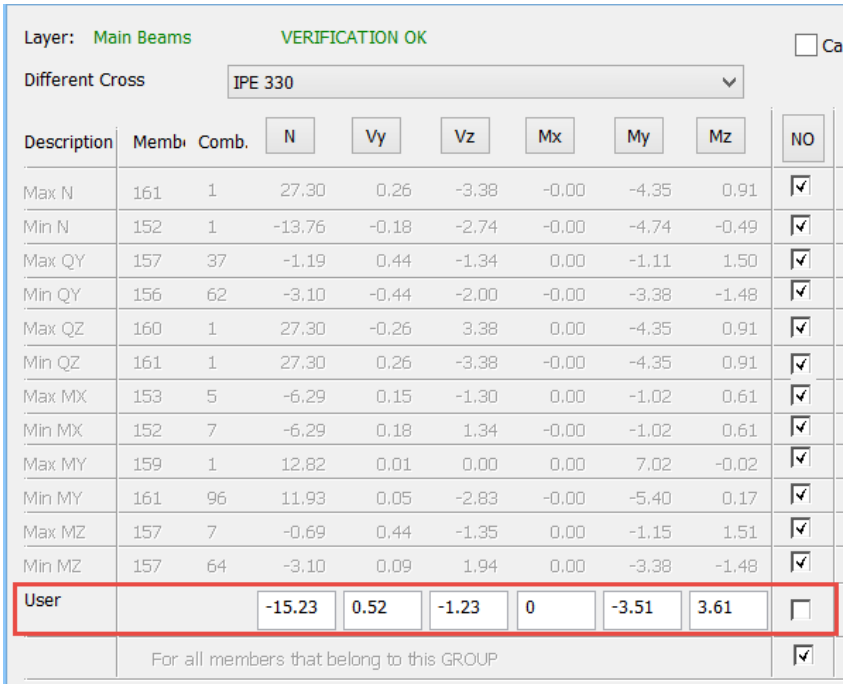
4. The "**AUTO**" **column** offers an automatic process through which the program calculates for each line of internal forces whose check should be done based on the values corresponding to each intensive force. This means that in case of considering N, My, Mz and Mx=Qy=Qz=0 the program performs Bending, 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.

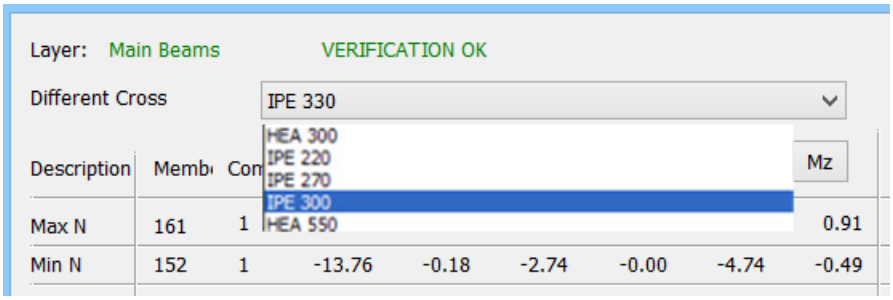
6. Activating "**User**" the user can type his values of the intensive forces to check the sections. In the next dialog box:



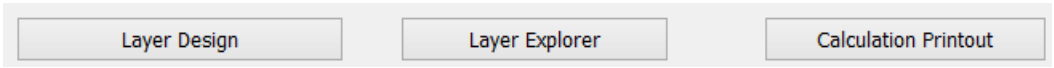
intensive forces are given by the user and those estimated by the program analysis are disabled.

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

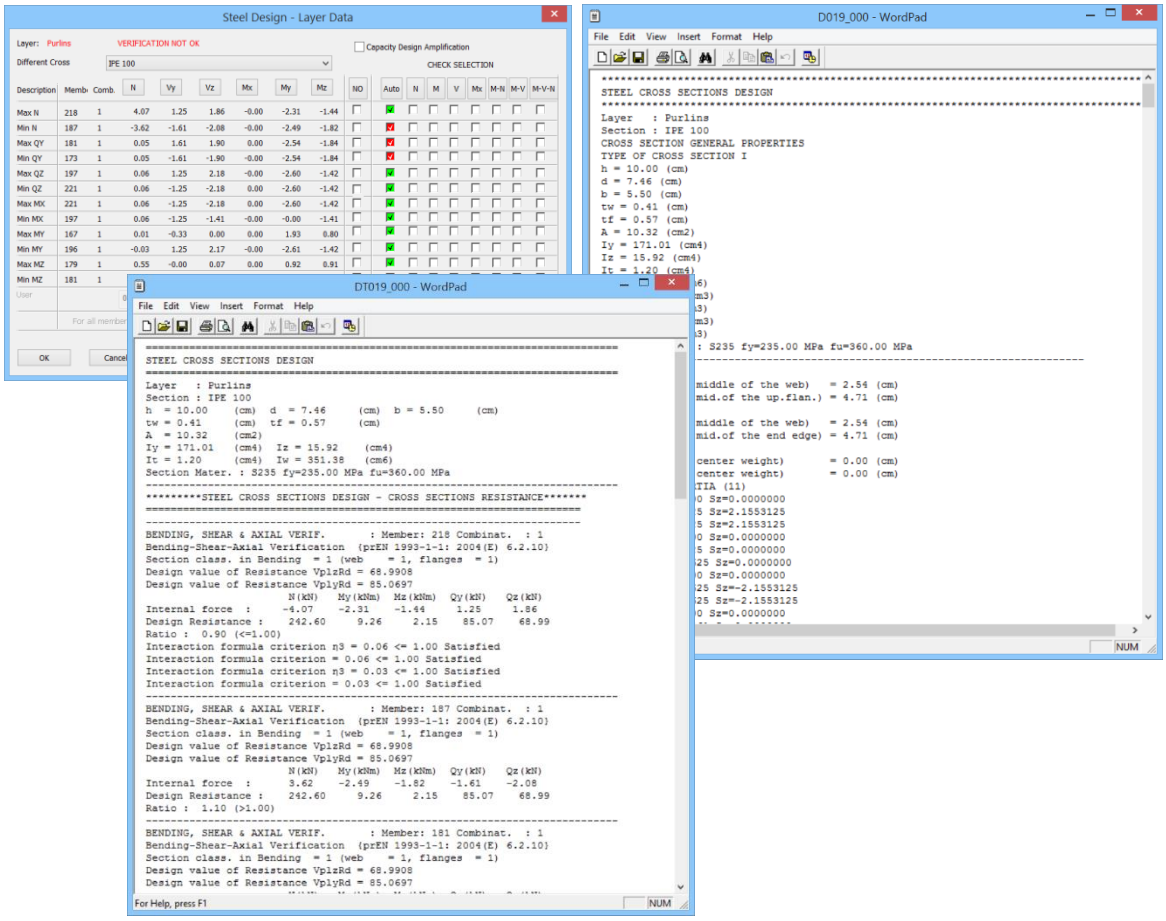
7. “**Different Cross Sections**” contains the different sections included in the "Steel Beams" layer.



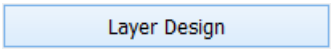
Follow the same procedures described above to design manually the other sections or to see the results:



- In table - Analytically - As Printout



8. Activate 

and press 

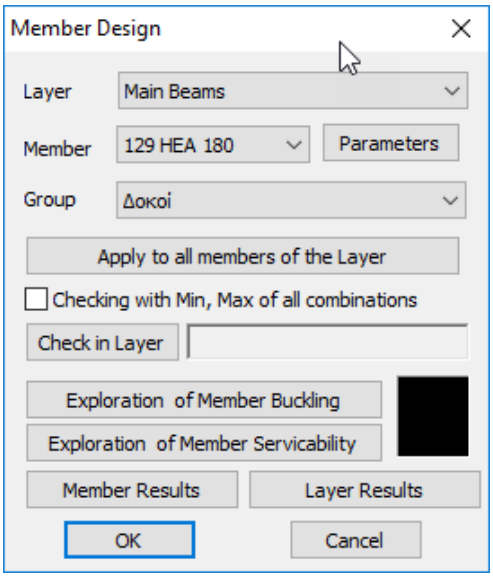
if you want to apply the capacity design in your checks.

7.1.2 Buckling Members Input

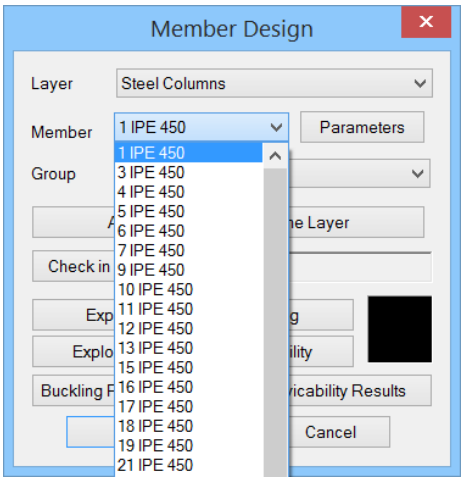
The buckling resistance check is one of the main design checks for steel structural members. Select the command “Buckling Members Input”, to apply on each member of each layer the following resistance checks:

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	

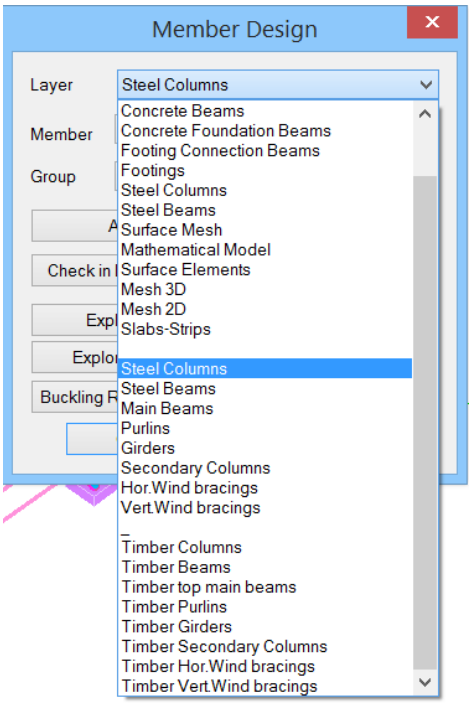
By selecting the command the following window opens:



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.

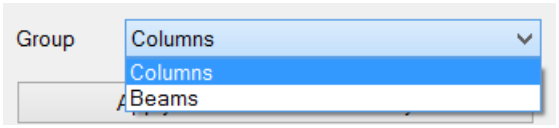


EXAMPLE:



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".

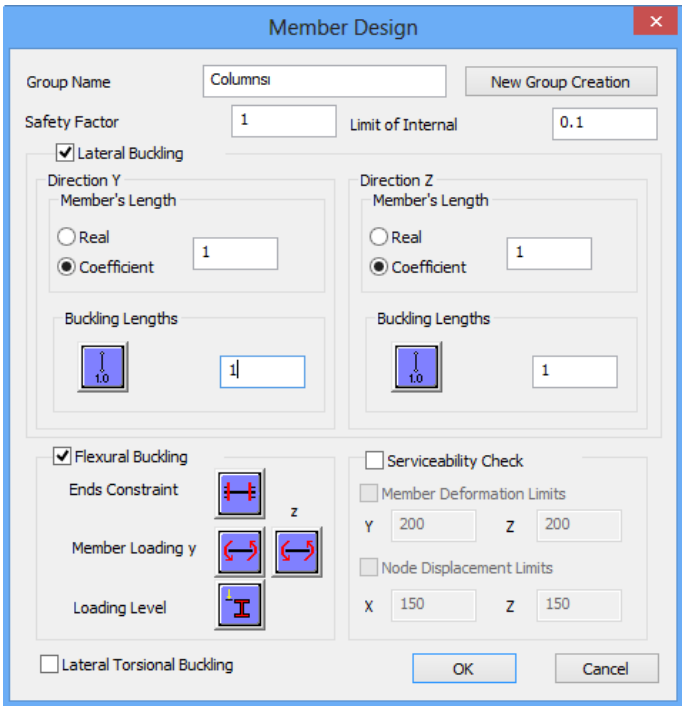


If you want to apply the same parameters to all members of the layer, then set the parameters once, keep the default name "Columns" and press the "Apply to all members of the layer".

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

Otherwise, 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.

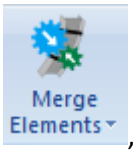
Select a “Layer” and click on the “Parameters”, and the following dialog box opens:



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.

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 one 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 four 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” 

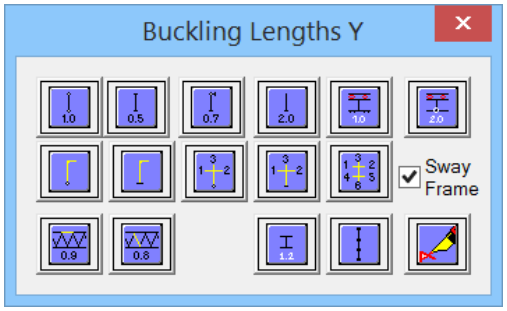
and type the real length in m, or activate the label “Coefficient” 

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

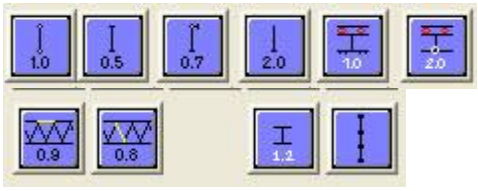
The parameter "Buckling Lengths" depends on the support conditions of the structural member.

Click on the following button 

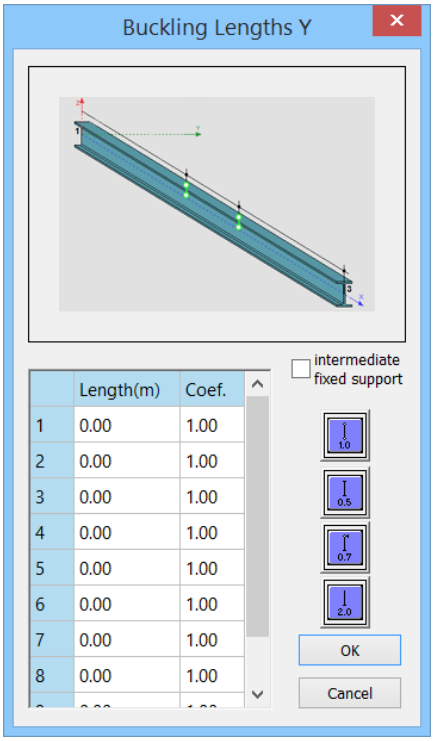
to open the following list and select the appropriate conditions so that the program automatically inserts the corresponding factor.



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 to calculate the corresponding reduced buckling lengths.

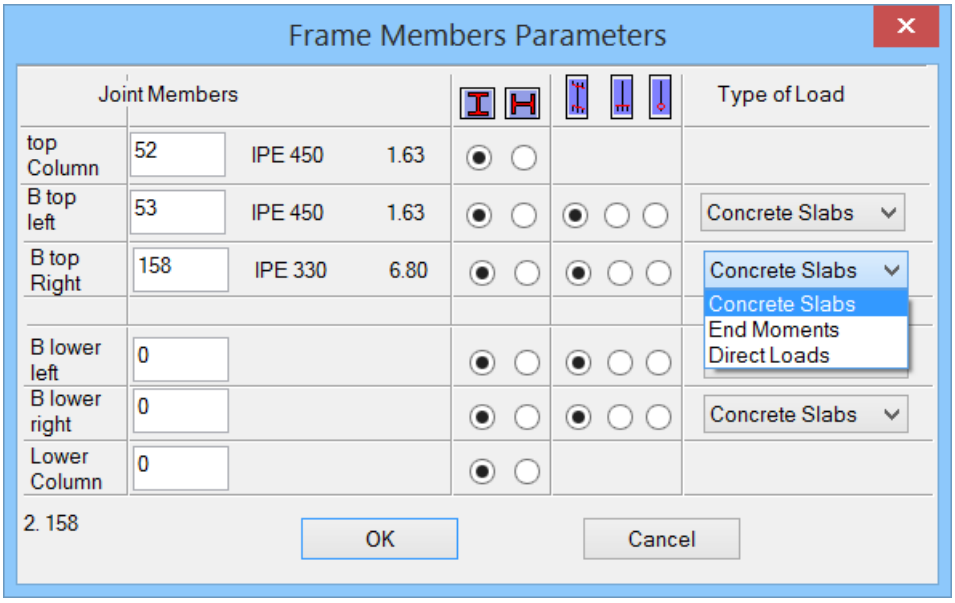


The second group

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 four horizontal) that offers succor to it (3 on the top and three 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).

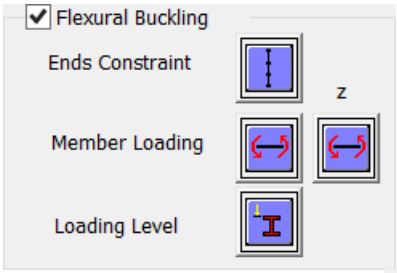
Then indicate their orientation an especially for beams the type of support on the other end, and the type of load imposed on them.

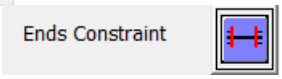
Pressing "OK", on buckling length the corresponding icon and the factor -1 appears, which generally means that the program based on the data you entered automatically calculates the buckling length for this member.

Finally, choosing/user can type his buckling length.

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

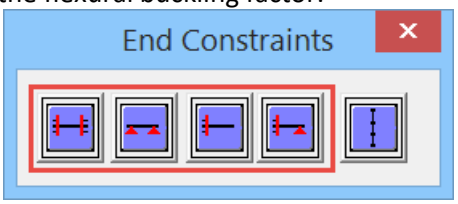
➢ Flexural Buckling resistance check:

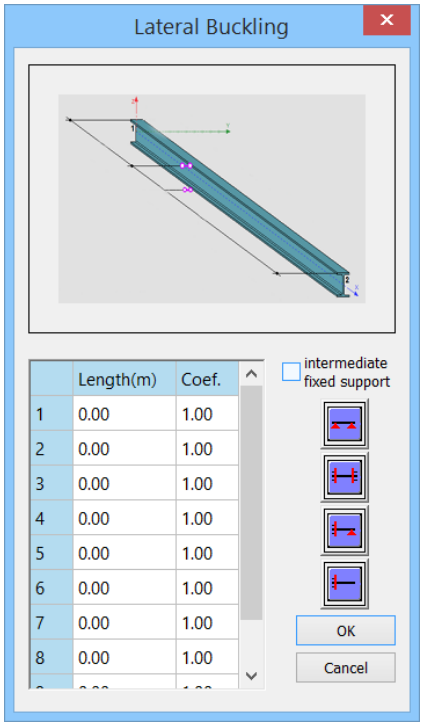


Activate the checkbox and press 

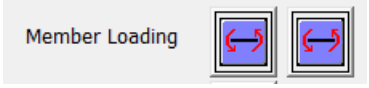
The "End Constraints" window, containing the various types of constraints opens.

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

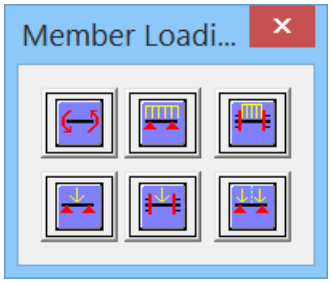




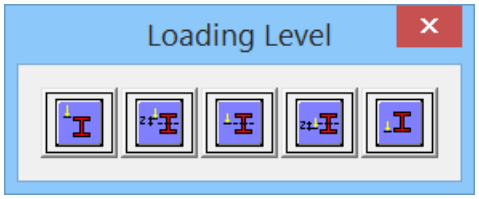
The last one  gives the user the opportunity to consider different constrains along the same member.

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.

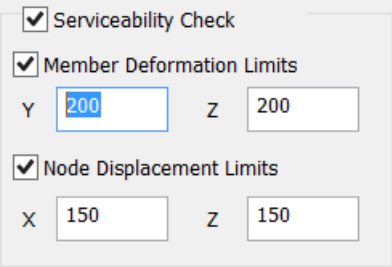
Loading levels for each icon:
1st icon: on the upper flange of the element
2nd icon: near and upward from the axis of symmetry of the element
3rd icon: on the axis of symmetry of the element

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.

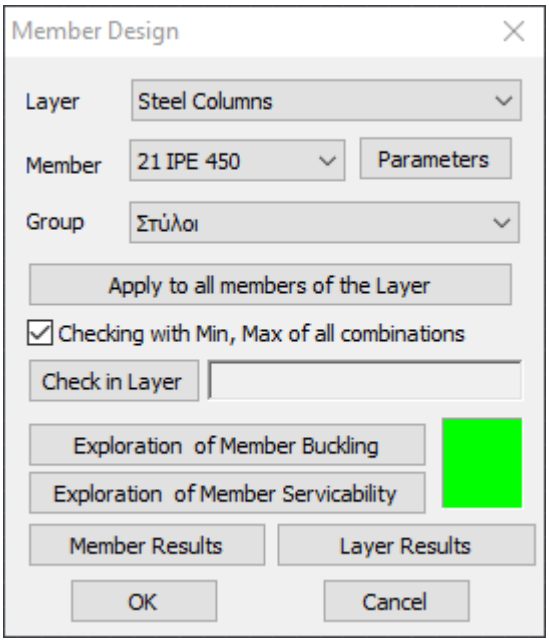
⚠ **NOTE**: For the lateral buckling and the lateral torsion buckling resistance check, the parameters are the same.

➢ 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.



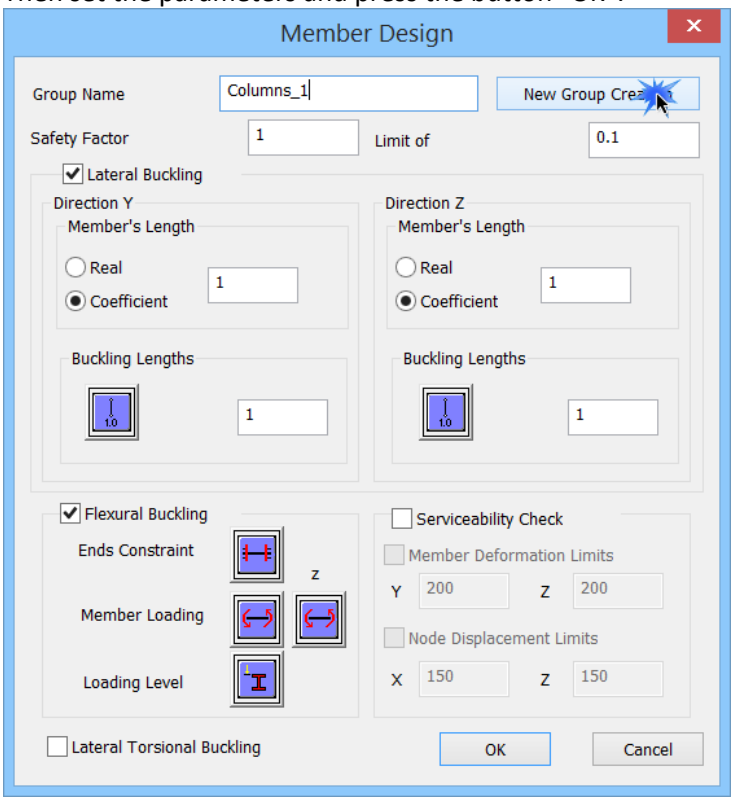
To apply the parameters you set to all members of a layer, select the command "Apply to all members of the Layer".
Click the button "Check in Layer" to check every member of the current layer, for every load combination. The results of the design checks are displayed in the black window that becomes 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 that the process will be completed at noticeably shorter times.

EXAMPLE:

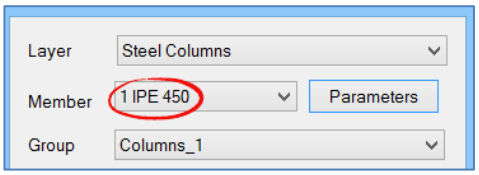
To define another set of parameters for some of the members of the layer follow these steps:

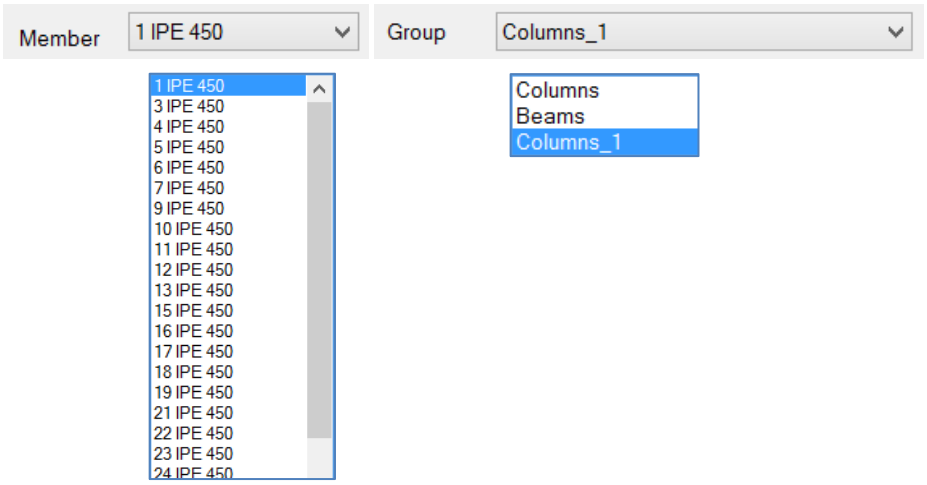
1st step: Press the button "Parameters" and open again the parameters dialog box.
Type the "Group Name" for the new set of parameters that will be created and press the button "New Group Creation".
Then set the parameters and press the button "OK".



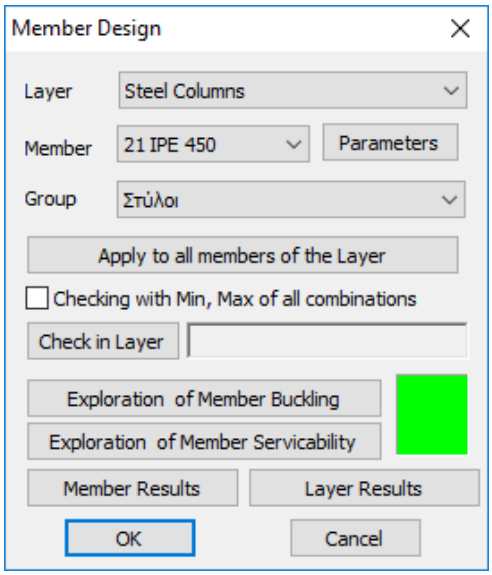
2nd step: define the members of the layer that will belong in “Columns_1” group.

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.



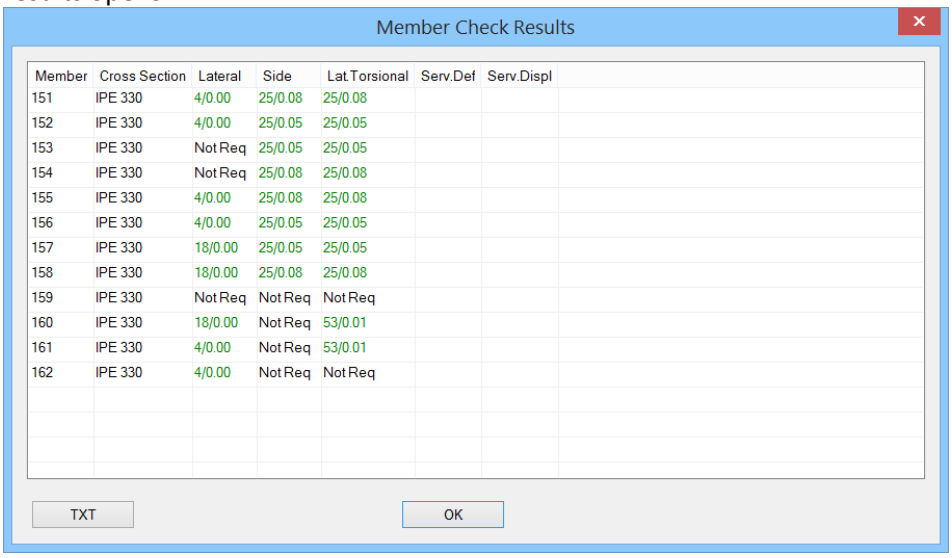


As soon as the layer check is completed, the color of the window on the right gives the information:



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:

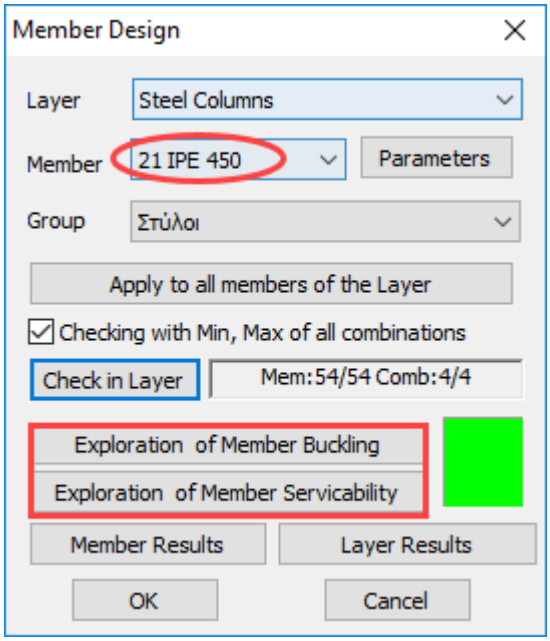


The first column indicates the number of the member, the second column indicates the cross section and in the next five columns, the least favorable ratio of strength and the combination number from which this ratio resulted is displayed.
Greens are the ratios below unity and red the ratios above it.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

NOTES:

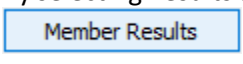
⚠ The check for the three types of buckling is performed for each member and all combinations. For each group of (N, My and Mz) the checks are made four times based on the following combinations:
N with min My and min Mz
N with min My and max Mz
N with max My and min Mz
N with max My and max Mz

⚠ That’s why in the output results and 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.

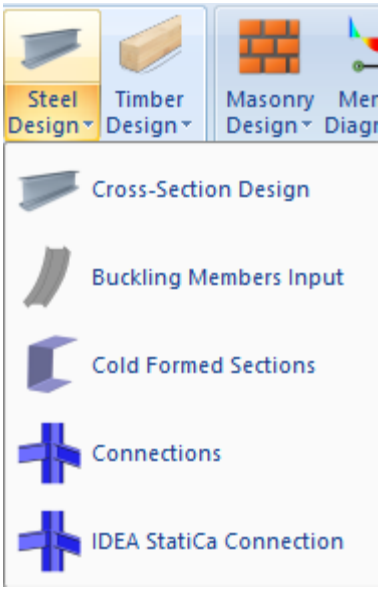
7.1.3 Cold Formed Sections

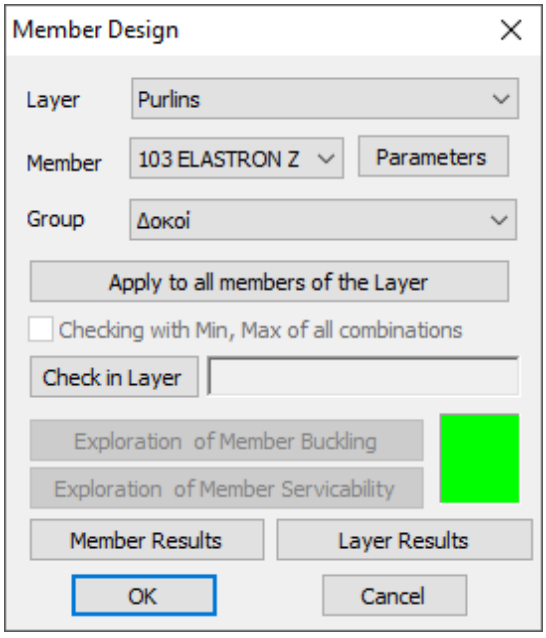
This command concerns the checks of cold formed sections.
The design of cold formed sections concerns the:

- **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

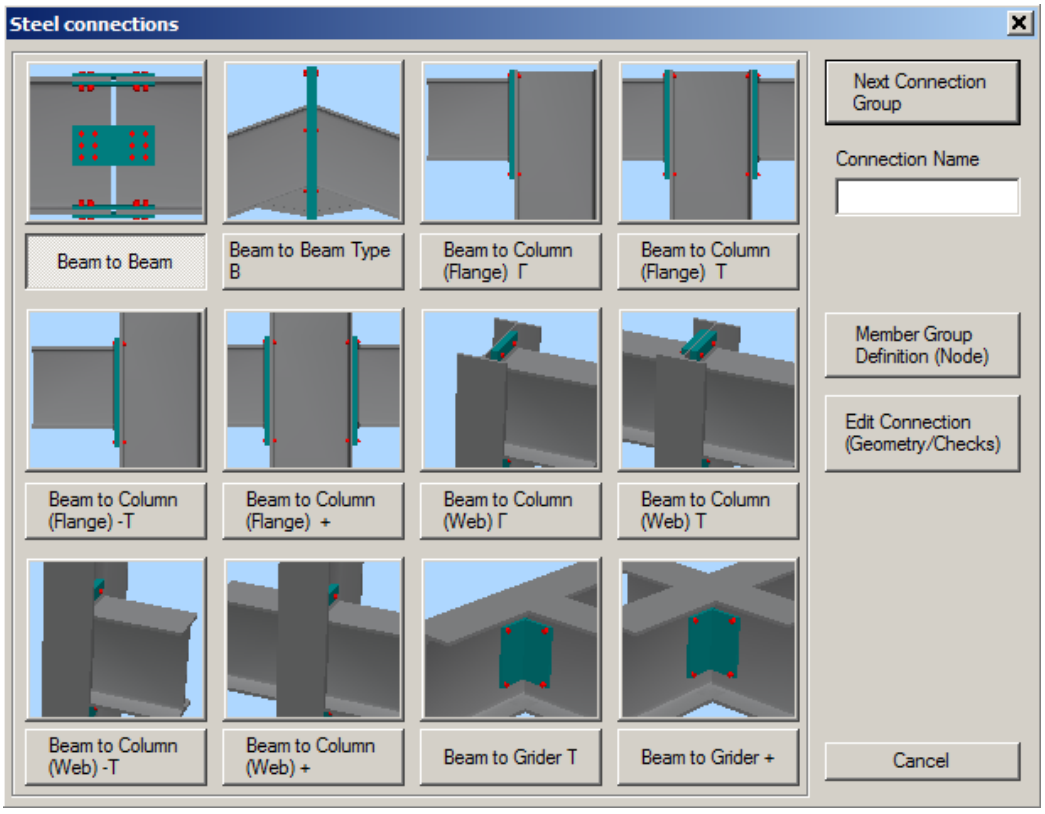
4. **Page 4: Cross-Section check for the 1st member with cross-section 1**
   Resistance checks based on §6.1
5. **Page 5: Member check for the 1st member with cross-section1**
   Resistance checks based on §6.2 & 6.3 and serviceability check based on §7
6. **Repeat steps 4 & 5 :**
   If multiple members 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)

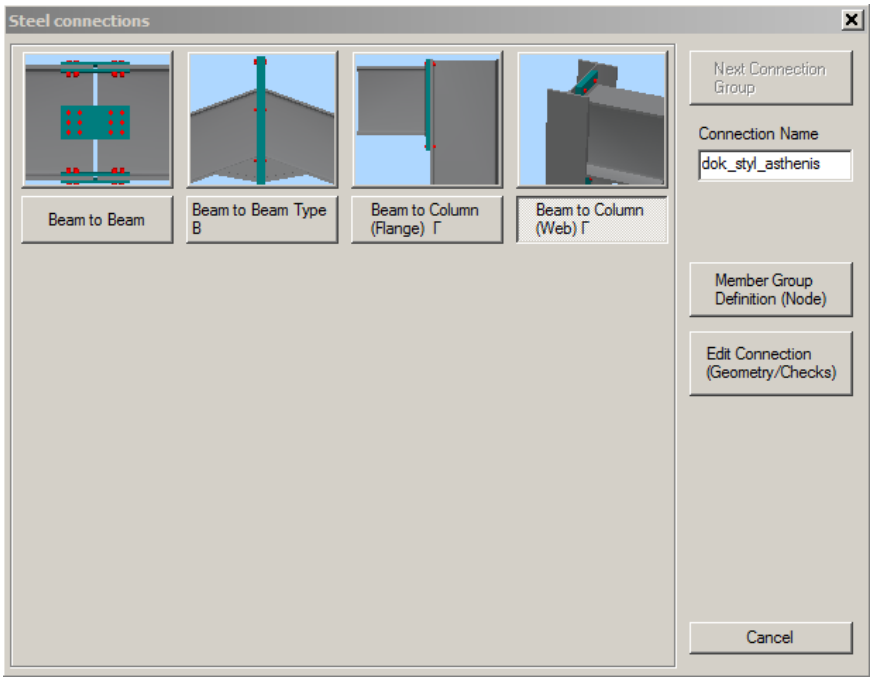
7.1.4 Connections

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.



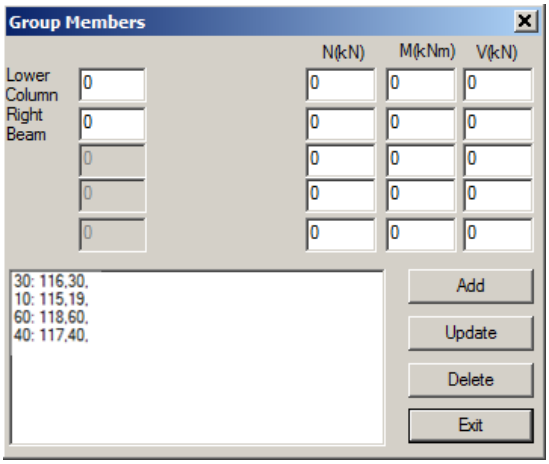
EXAMPLE:

Left click to select member 30 (column) and member 116 (beam) and right click to open the library with the four 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.

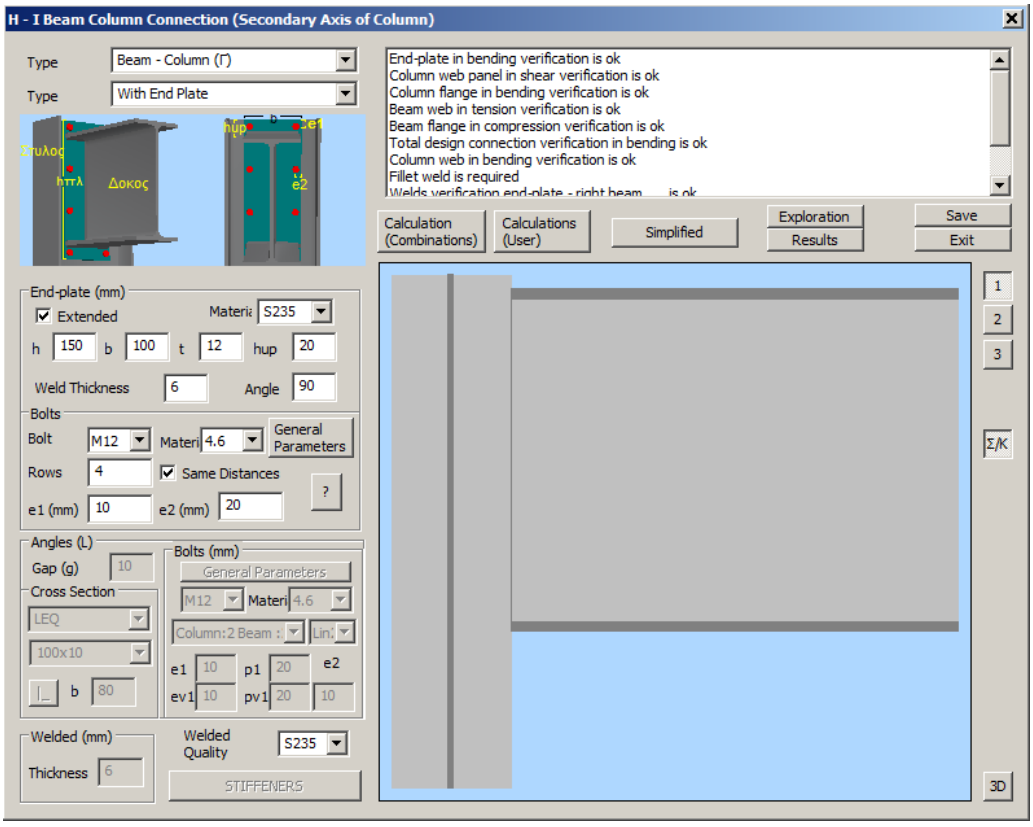
Then, select the “Member Group Definition (Node)” command and in the dialog box you can add more groups of members with the same connection features (i.e. column – beam) or type your values for the stress resultants N, M, V for the existing groups.
To add groups of members, click into the field “Lower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



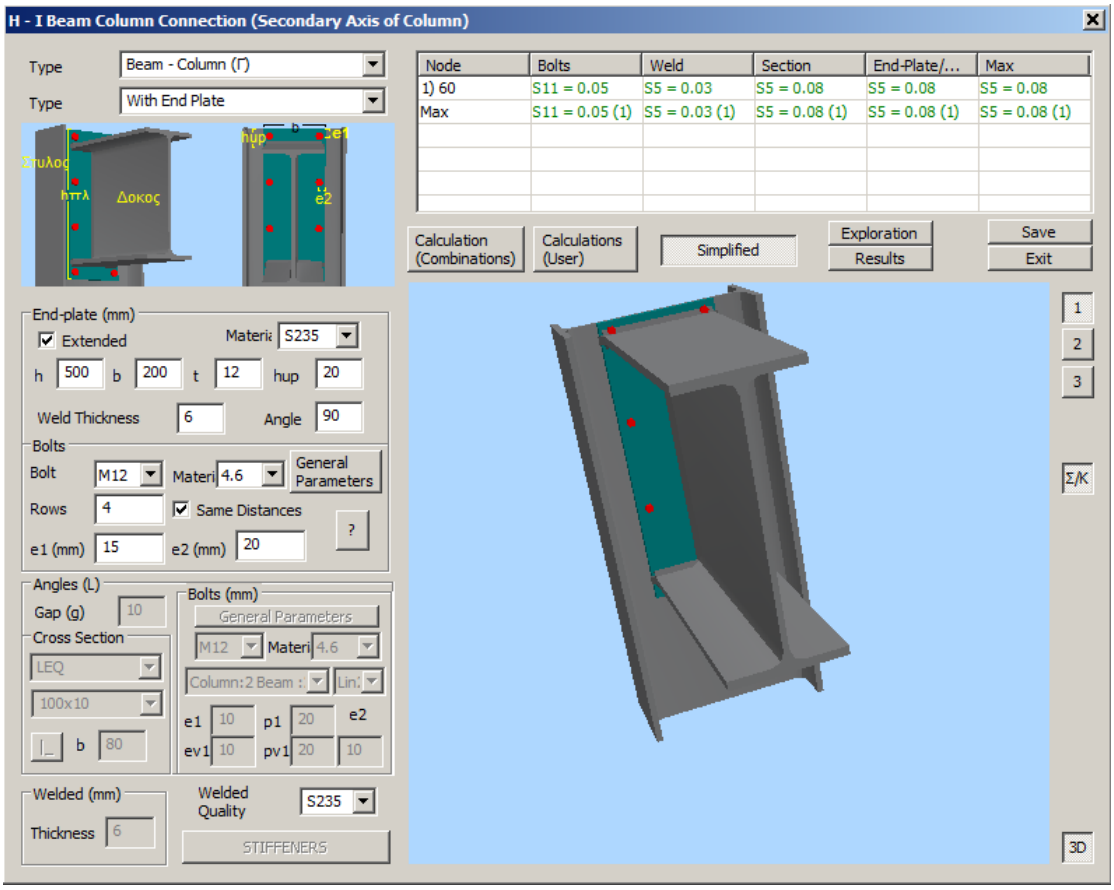
Use this dialog box for the design of steel connections with the same type and the same cross-sections in total (i.e. column IPE 450 - beam IPE 330).

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)".



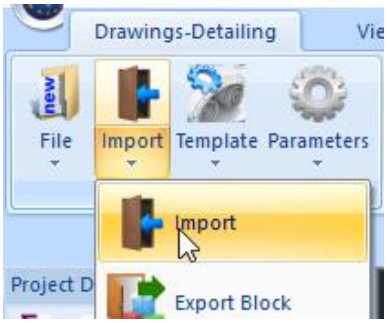
Click the button “3D” to see a three-dimensional representation of the connection that is updated as you change the parameters.
The buttons “1”, “2”, “3” are used for the display of the two side views (1 & 2) and the plan view (3). The button “Σ/K” is used for the display of the three-dimensional representation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

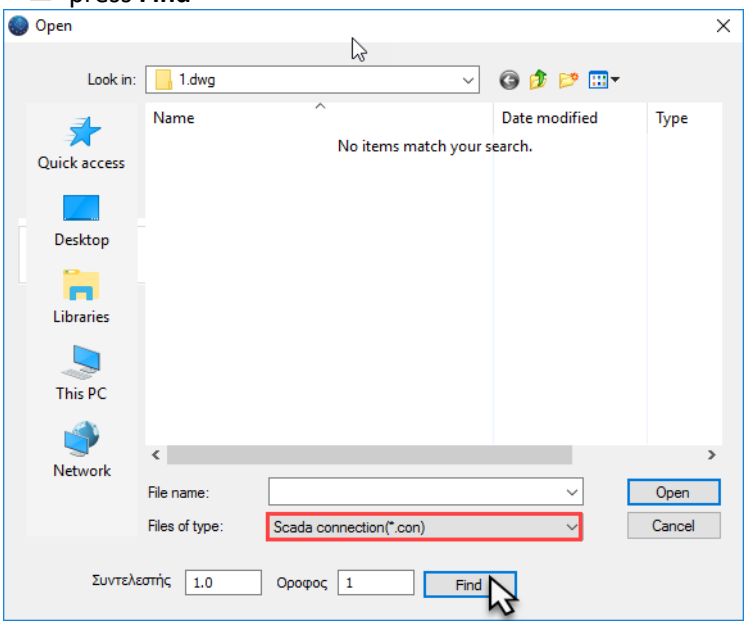
Connections' drawings are located in the folder of the project "sxedia":
C:\SCADApro\ "project name" \scades_Synd\sxedia

You can import them using:

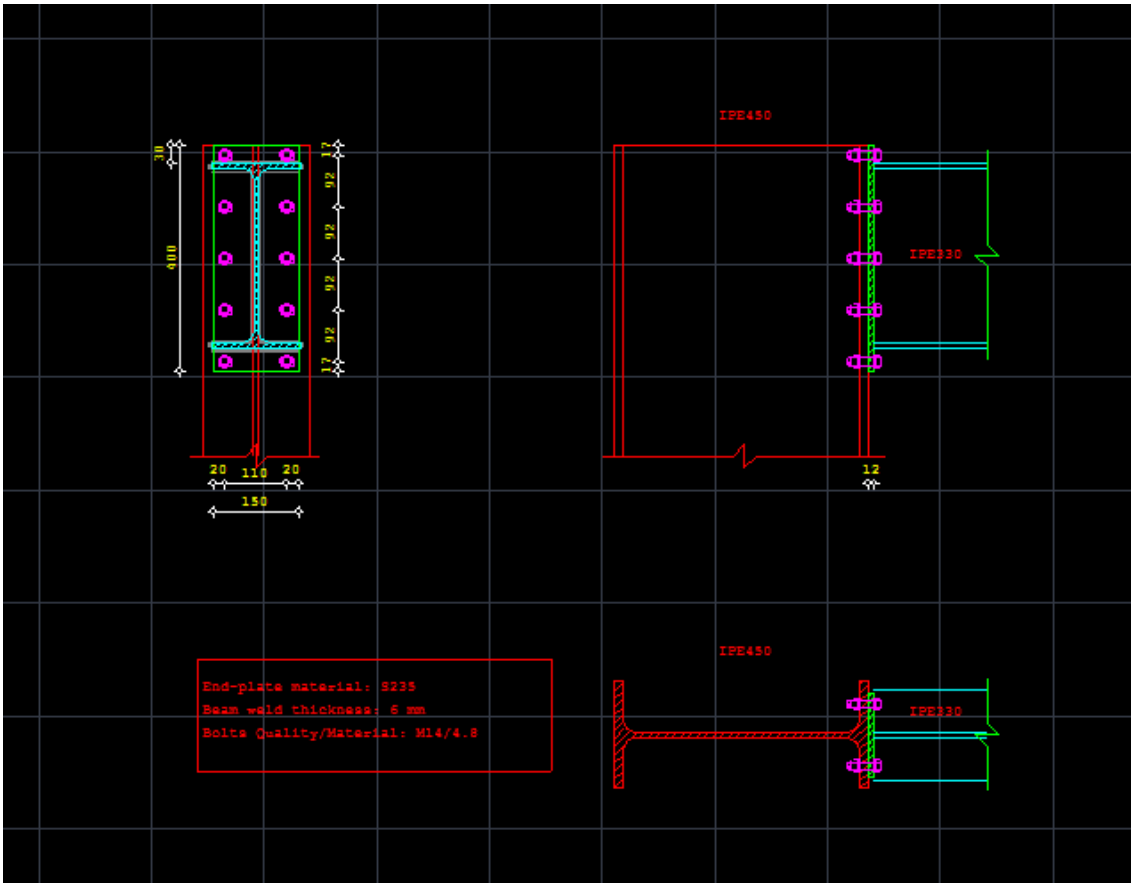


And in the dialog box:

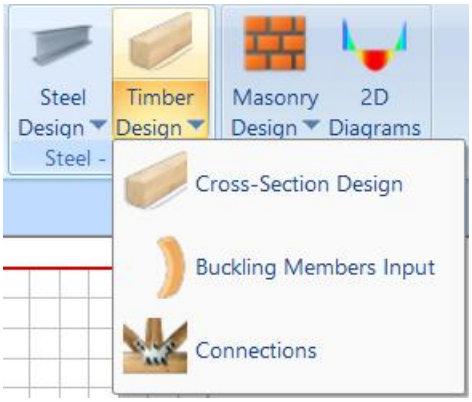
- in Files of Type select **SCADA Connection**
- press **Find**



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.



7.2 Timber Design



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

NOTES:

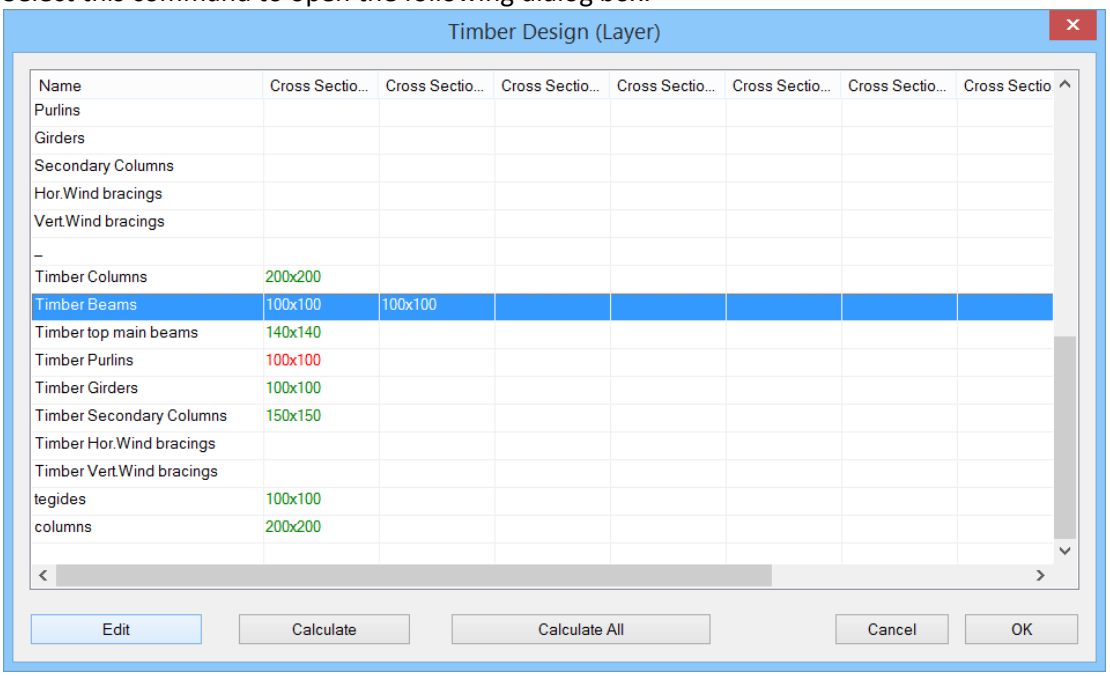
⚠ Always remember to calculate the corresponding load combinations in the parameters dialog box.

⚠ The design process of the timber sections is similar to that of steel sections.

7.2.1 Cross Section Design

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

Select this command to open the following dialog box:

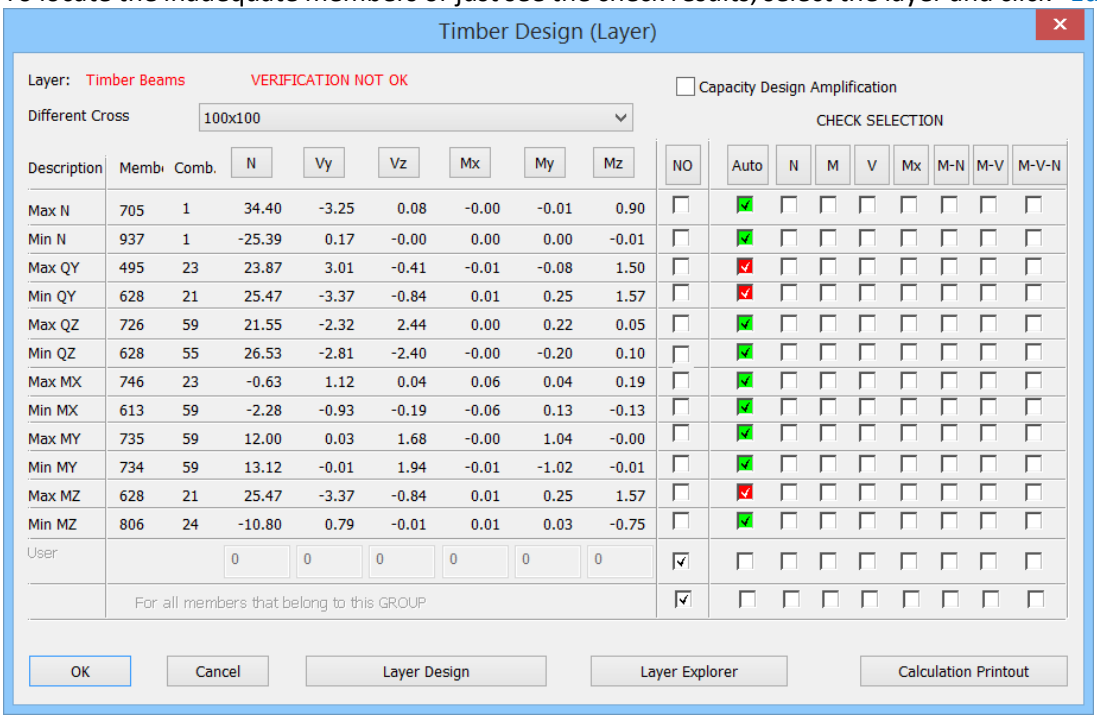


The first column contains the layers of the current project and the other columns the cross-sections that belong to each layer.

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

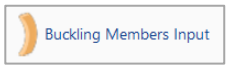
Green color indicates that all sections of this layer satisfy the design criteria (stress/resistance ≤1), while red color indicates that they don't.

To locate the inadequate members or just see the check results, select the layer and click "Edit".



⚠ *The whole procedure for the layer design is described in detail in the respective section of steel sections (See Steel> Steel Design> Cross Sections Design)*

7.2.2 Buckling Members Input

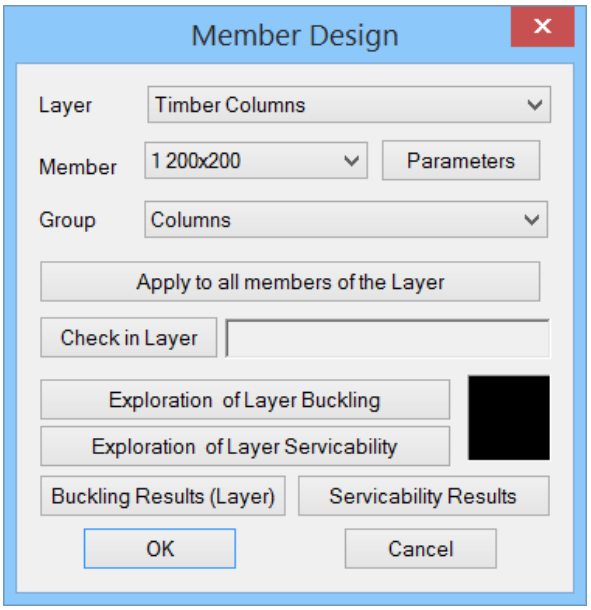


The buckling resistance check is one of the main design checks of the timber structural members. Select the command "Buckling Members Input", to apply on each member of each layer the following resistance checks:

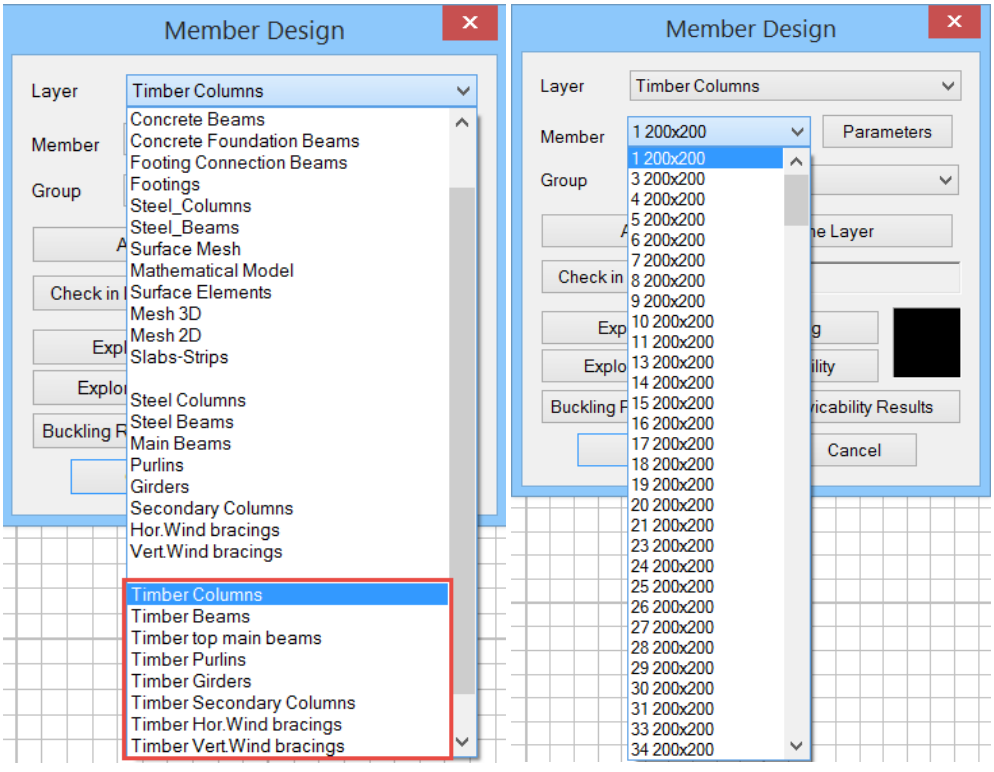
ULS (Ultimate limit state)	SLS (Serviceability limit state)
Flexural Buckling check	Member Deflection check
Torsional Flexural Buckling check	Node Displacement check
Lateral Buckling check	
Lateral Torsional Buckling check	

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

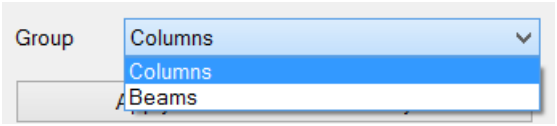
By aelecting the command the following window opens:



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

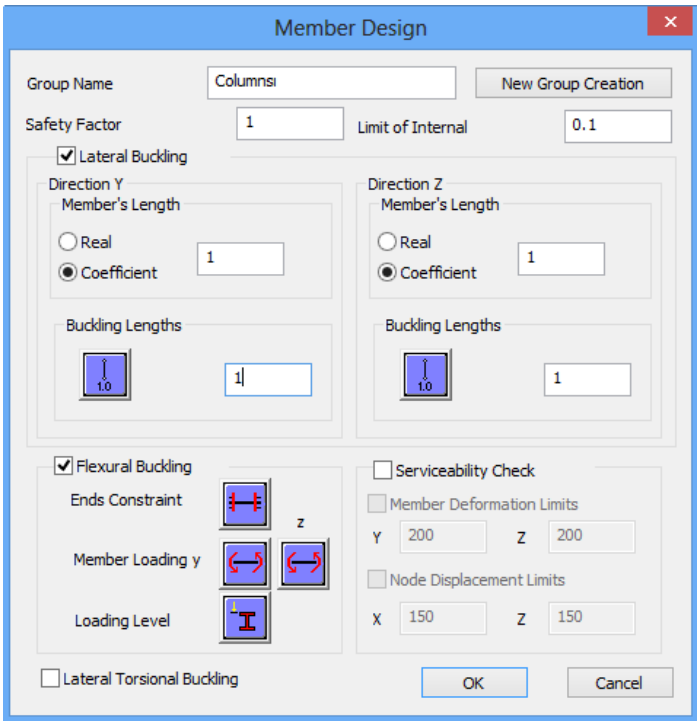


EXAMPLE: select from the drop-down list the “Timber Columns” layer. In the “Members” the list displays all the structural members that belong in the selected layer. If you want to define different parameters for some of them, you can create different “Groups” in the same layer. The program has two default Groups: "Beams" and "Columns".



In case 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 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:

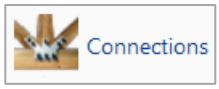


In the "Group Name," you see the name of the parameter group. If you want to create your group, give a new name and press the button "New Group Creation".
In the "Safety Factor," you can set the limit for the program for the design checks: the intensive forces to the respective strength of the member.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

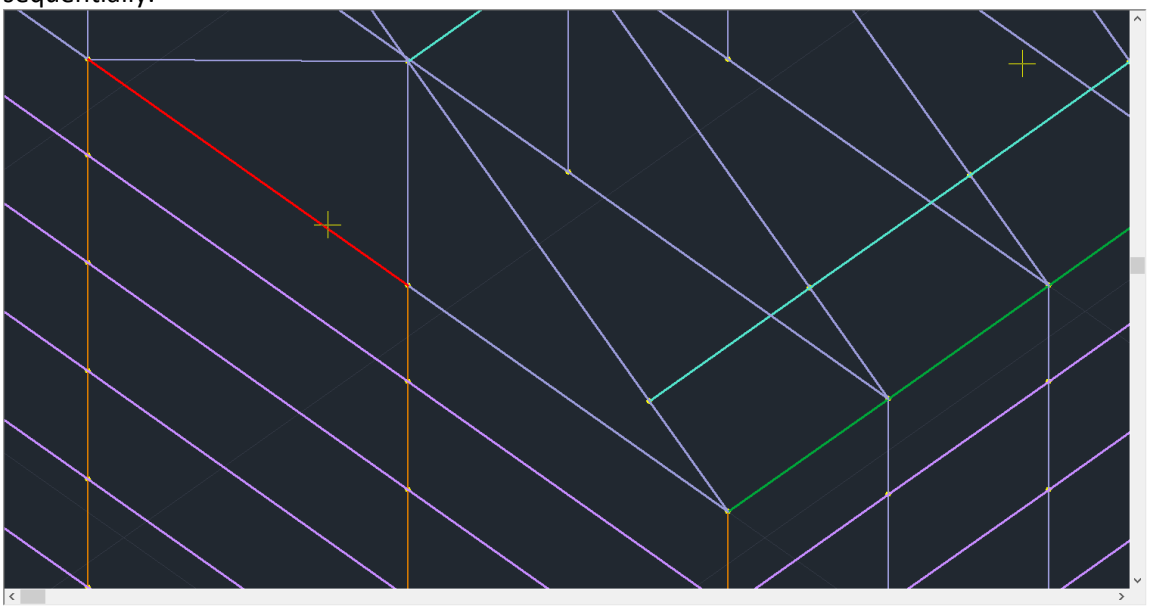
The rest of the form is divided into four parts, one for each check:
Lateral Buckling Lateral Torsion Buckling
Flexural Buckling Serviceability checks

⚠ *The whole procedure for the layer buckling resistance check is described in detail in the respective section for steel sections (See Steel> Steel Design> Buckling Members Input)*

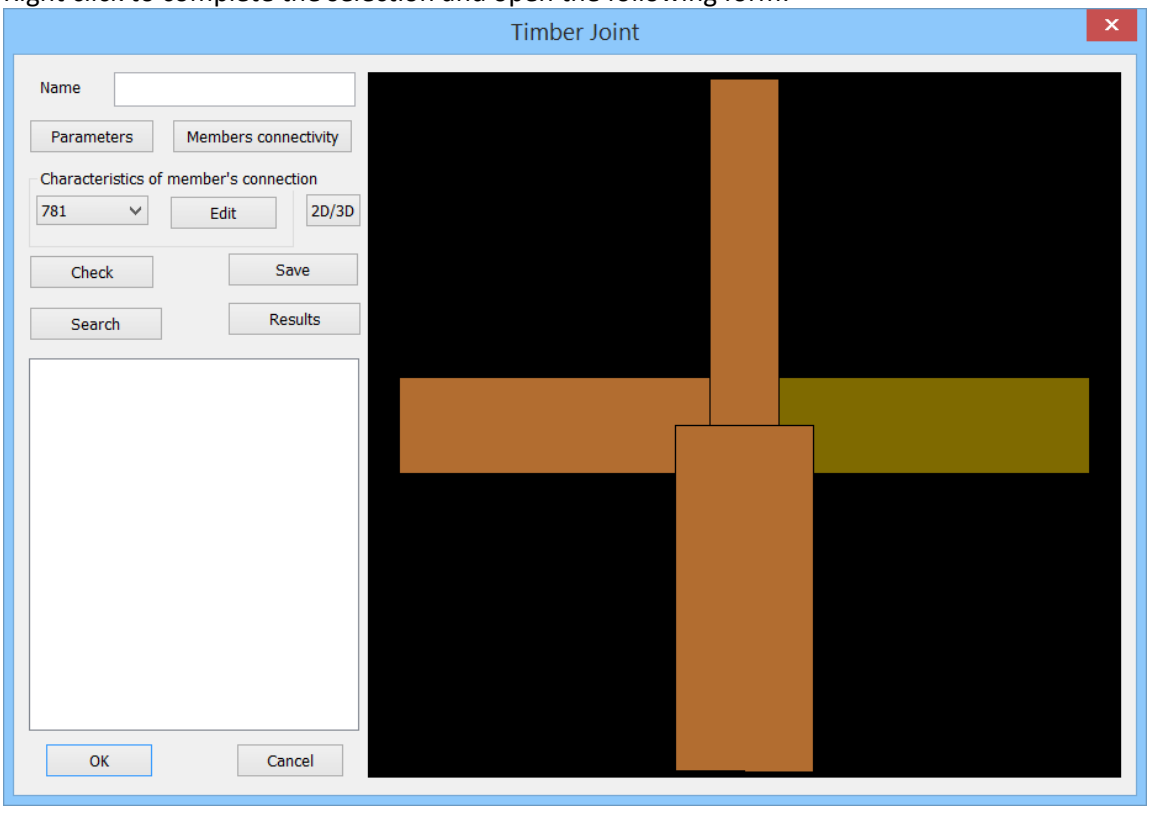
7.2.3 Connections



The last command of the “Timber Members Design” group command is the “Connections”, used for the timber connections’ design. Press the command and select the connected members sequentially.

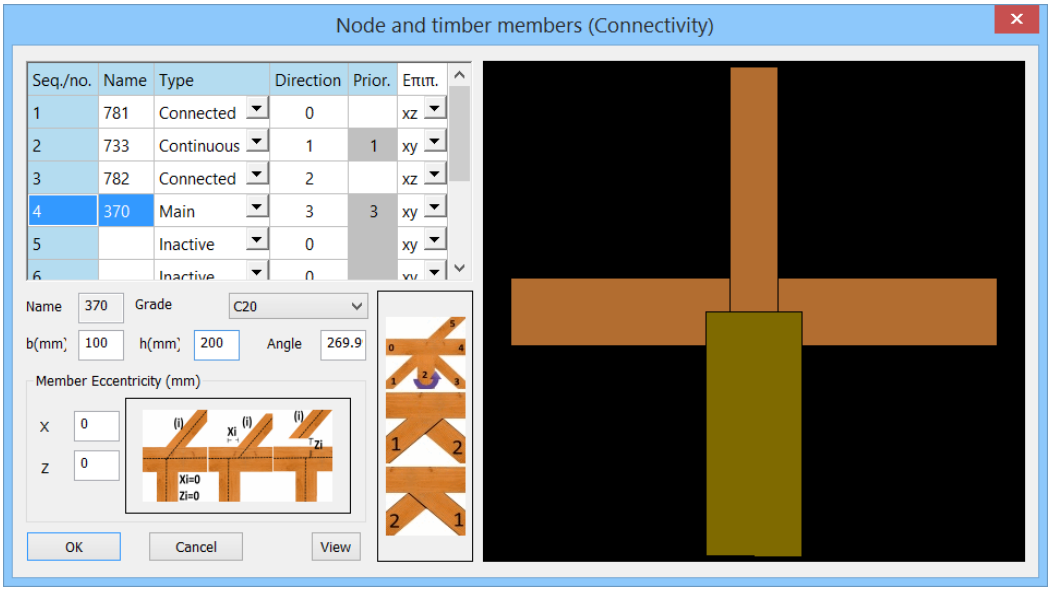


Right click to complete the selection and open the following form:

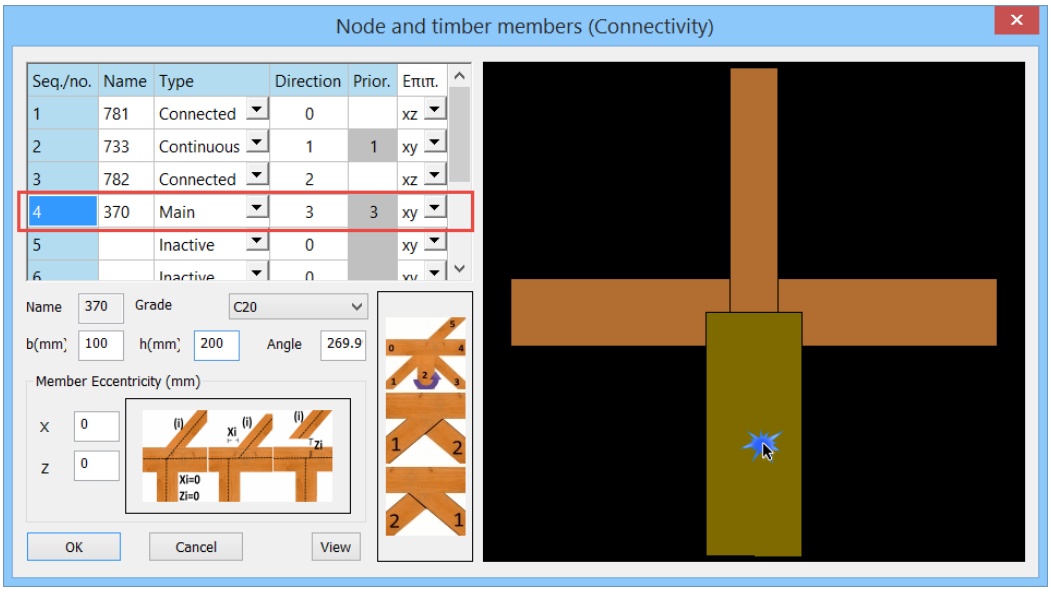


In the right part of the window, the connected members with b and h casually are displayed. Through Members connectivity command, the user defines the real dimensions of the members.

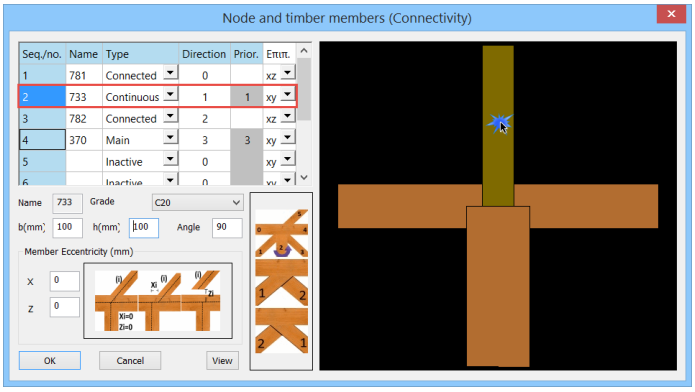
Name the connection and select the command **Members connectivity**.



In the first field, you specify the **Type** of the member.
Select graphically with a left click on the member to define the *Main Member* (each connection has only one Main member). In the left list, the selected member is automatically marked.



In this example, the Main member is 370.



If there is a collinear member (ex. 733) this can be defined either as **Connected** or **Continuous.**
All the other members of the connection are **Connected.**

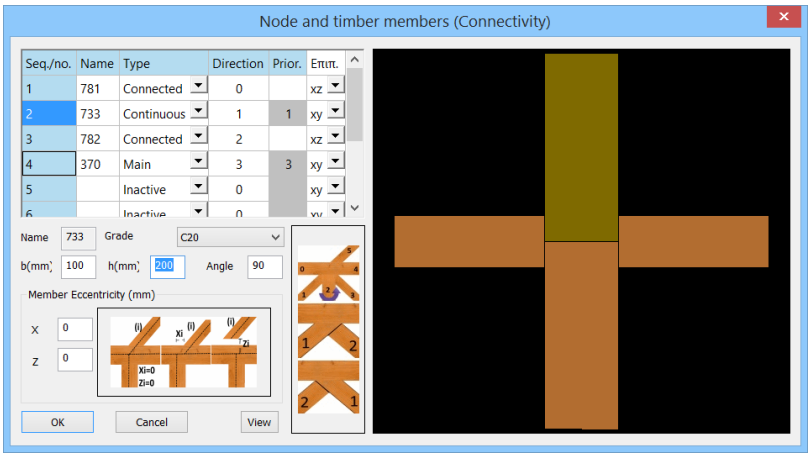
⚠ **Definitions:**

- **Main Member**: may be any member of the connection
- **Continuous**: is the member continued to the Main member without interruption. It is a single member and cannot have dimensions different than the main member.
- **Connected**: is the member connected with other members and can have different dimensions than the connected.

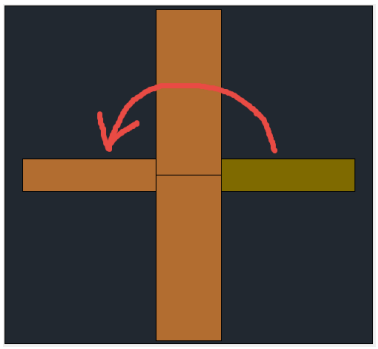
Therefore, set, in the same way, the Type of all members.

The next step is to define the **dimensions** of each member.

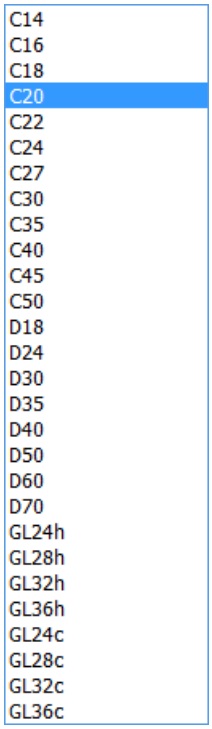
Choose from each list the values of b and h.
b= the thickness of the member (dimension perpendicular to the screen)
h= the height of the section (dimension in the screen level)



Angle: Is the angle of the member as to the connection. The angles are defined counterclockwise with 0 in + x.



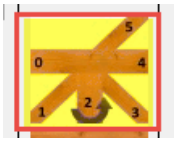
(right to the connection)



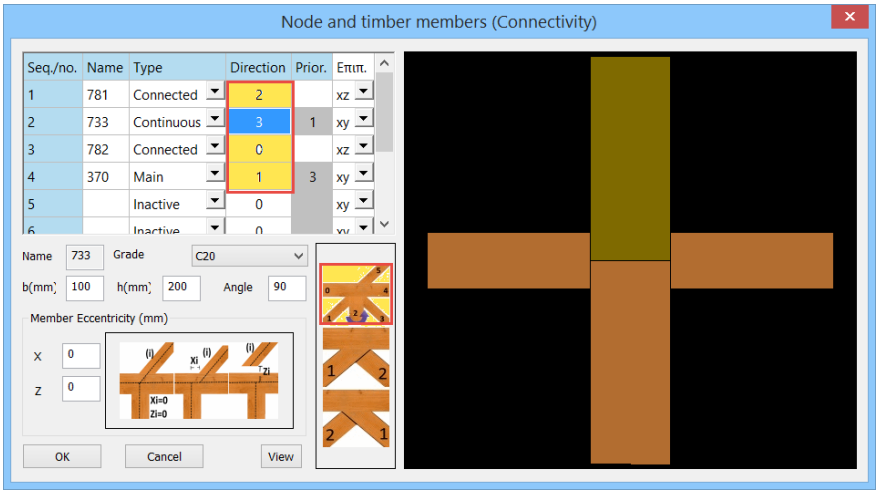
Grade: to set the quality of each member, select the member and its quality

⚠ **ATTENTION:**
The Main and the Continuous Members cannot have different dimensions. They are the same element!

View command displays the connection in total with the lengths of the members.



Direction: The direction of each member is defined according to the icon. Therefore, start by selecting the left-hand member to define the 0 directions and continue defining the direction of the remaining members of the connection.



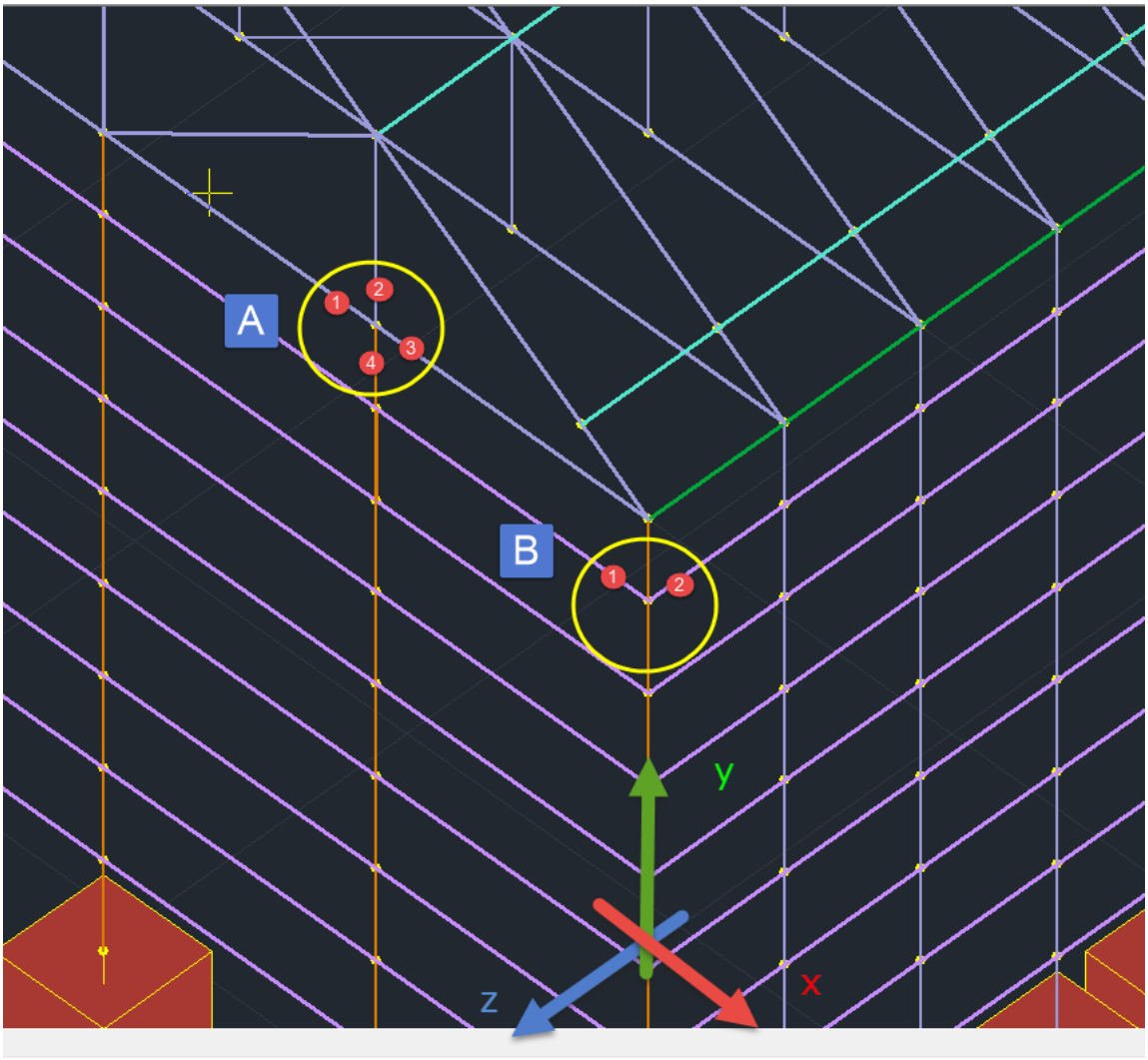
Priority: With the priority, you set the Connected member that "prevails" in the connection. This is related to the "cutting" of a Connected member that meets another member.

In the priority column set a number only for the Connected members.

Node and timbe

Seq./no.	Name	Type	Direction	Prior.	Lev.
1	781	Connected	2	1	xz
2	733	Continuous	3		xy
3	782	Connected	0	2	xz
4	370	Main	1		xy
5		Inactive	0		xy
6		Inactive	0		xy

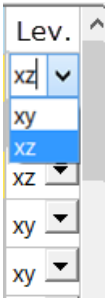
Levels:



The connection A is in XY level i.e. the steel connecting plate will be introduced at this level (vertically).
The connection B is in XZ level i.e. members 1 and two will be connected with a horizontal steel plate.
Only members that belong to the same level can be connected. Therefore column members can’t be connected in B connection.

NOTES:

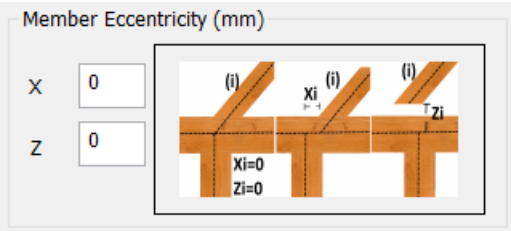
- The connection level defines the bending level of the members that will be considered according to their local axes, as well.
- Therefore from the six intensive forces (N, Mz, Vy, My, Vz, Mx) of each member, in the connection node, 3 of them will be taken into account, N, Mz, Vy in the xy level, and N, My, Vz in xz level.



Choosing the right level of each member based on local axes is defined in the **Level** column.

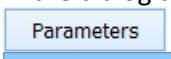
Member Eccentricity:

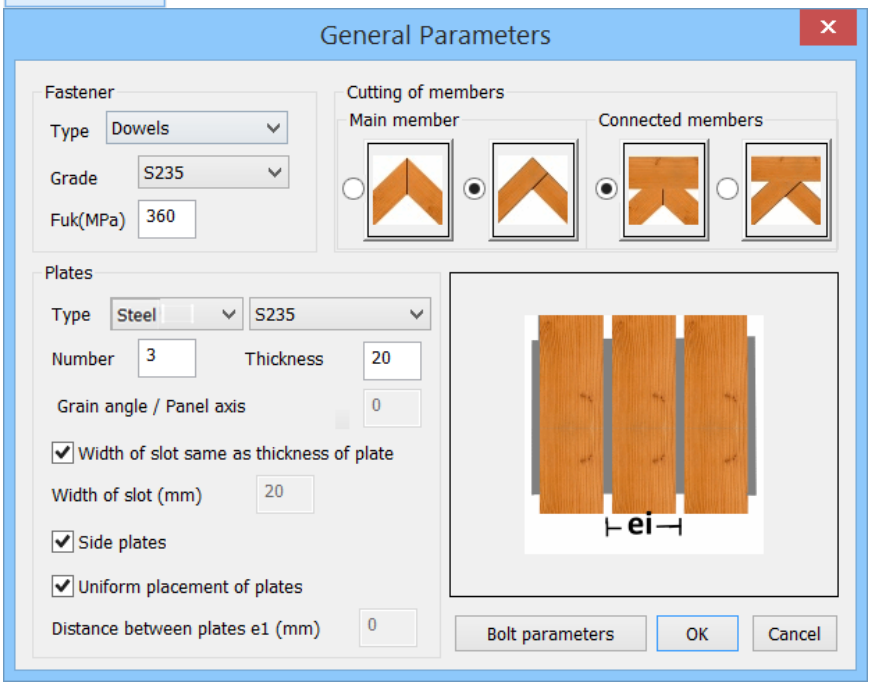
Through **Member Eccentricity**, the end of a connected member can be moved from the connection node according to the eccentricity.
In this way, structure eccentricities are covered.



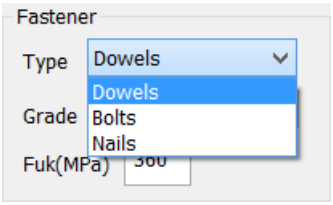
Select the member and according to the icon set the eccentricities in X and Z.

After Members Connectivity press **Parameters** to set the general parameters of the connection. In the dialog that opens define:

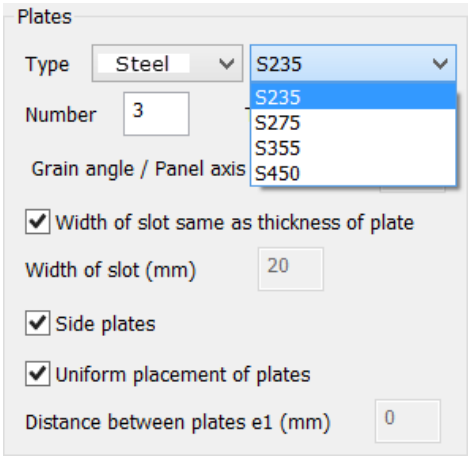


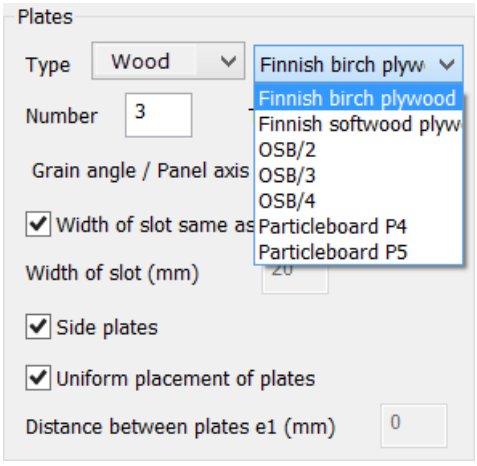


Edit

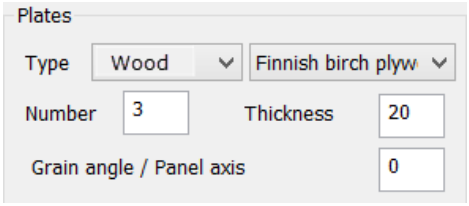


In the **Fastener** field define the connector **Type** (between Dowels, Bolts, and Nails) and the **Grade** and the Fuk tensile strength value is automatically updated.
Alternatively, the user can enter a value for the tensile strength to be taken into account during the checks.

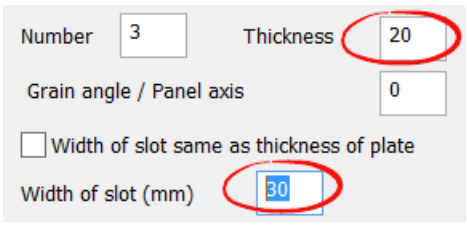




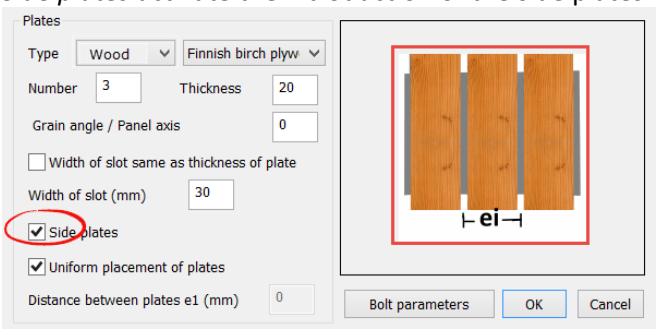
⚠ In case of **Wooden Flange**, the parameter Grain angle / Panel axis where the user defines the angle of the grain of the wooden flange as to the axis of the Main member, is activated.



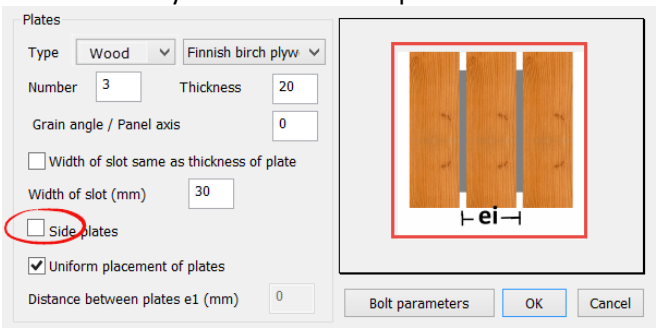
⚠ When the *Width of the slot* is bigger than the plates, deactivate the checkbox and define the width of the slot.



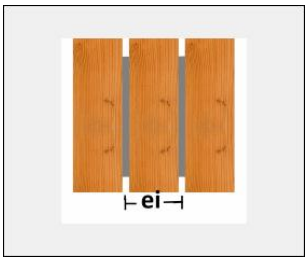
Side plates activate the introduction of the side plates.



Otherwise only the intermediate plates exist.

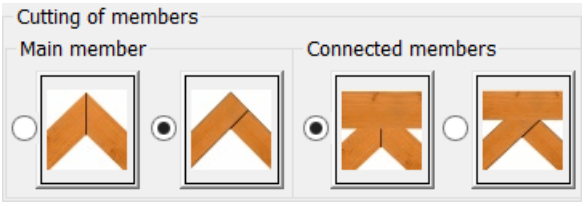


Plate's placement in the section of the timber member (along with its thickness) can be *Uniform*, i.e. divides the cross section into equal parts (active checkbox) or not.

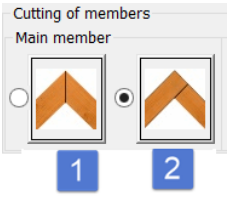


In the second case deactivate the check and set the distance **ei** as defined in Figure.

⚠ Connections with Side plates require uniform placement of plates.



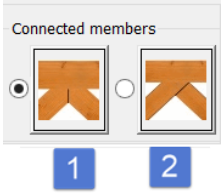
In **Cutting of members,** field chooses the sectioning modes of member’s profiles.



The first two options concern the cut of the Main member with the Connected member with Priority 1:

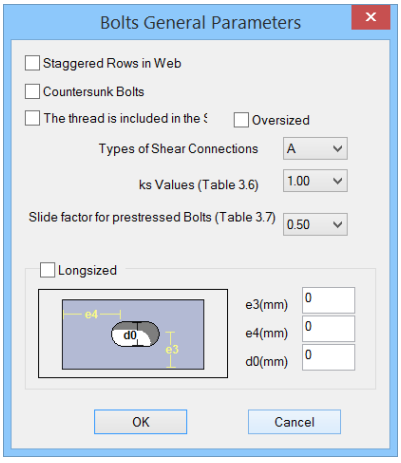
1. Main and Connected member are both trimmed
2. The Main member prevails to the Connected.

The other two options concerning the cutting of Connected members based on the priority:

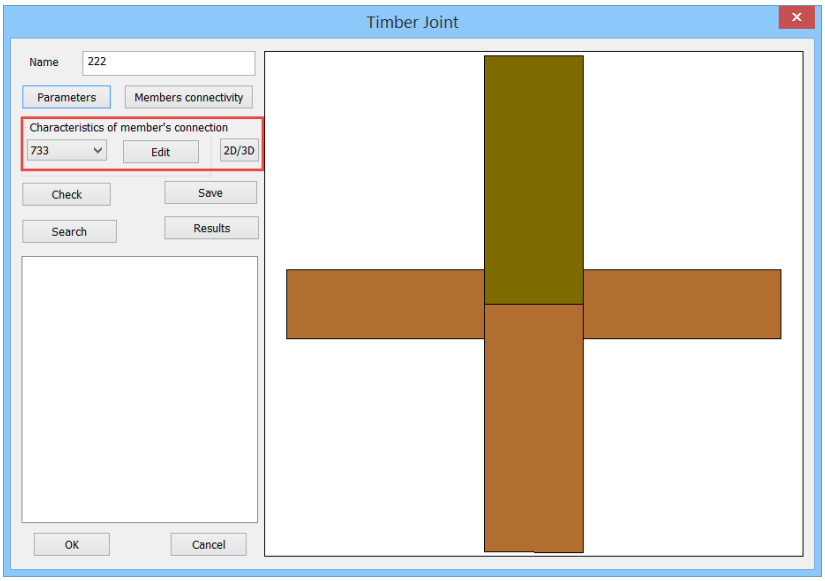


1. Connected member both cut
2. the Connected with bigger priority prevails.

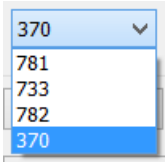
Bolt parameters to set additional parameters concerning bolts in steel plate.



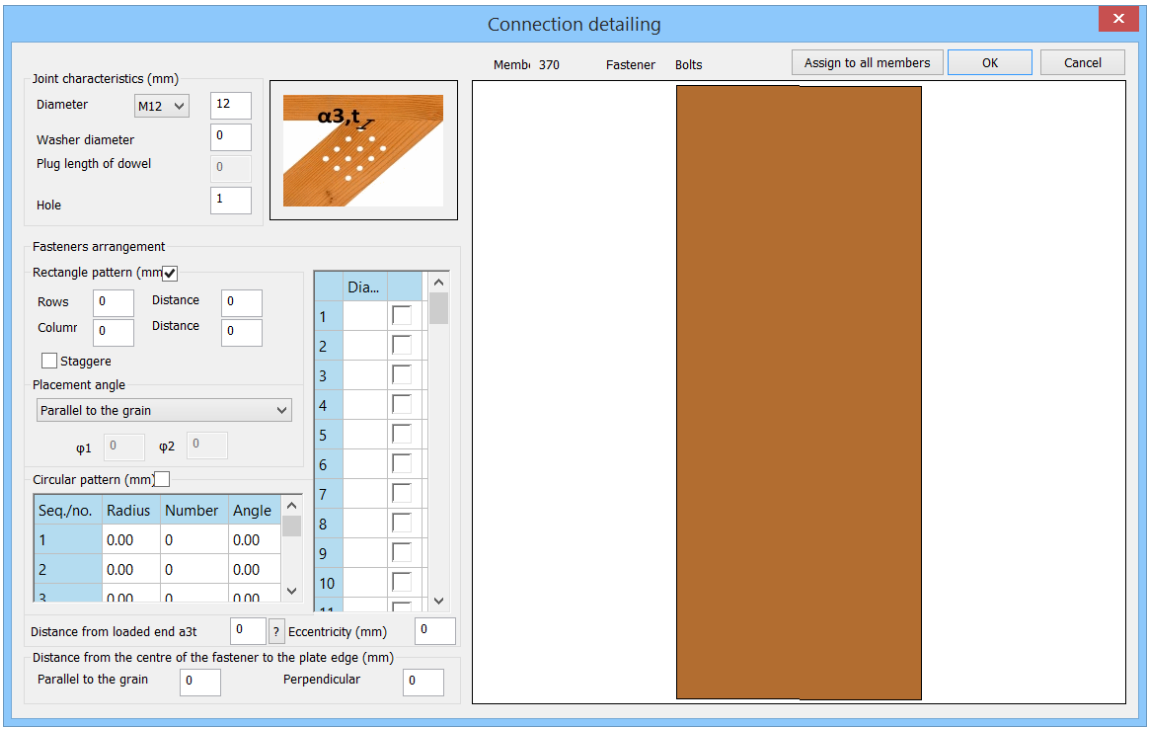
Characteristics of member’s connection

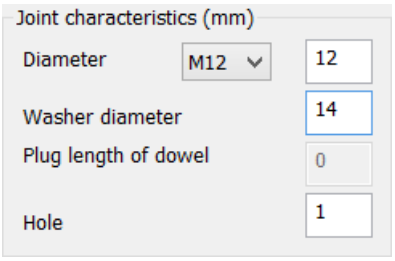


After Member’s connectivity and Parameters definition, the connection’s **Editing** follows. Choose a member, starting from the *Main*, either graphically in the right figure, or from the list

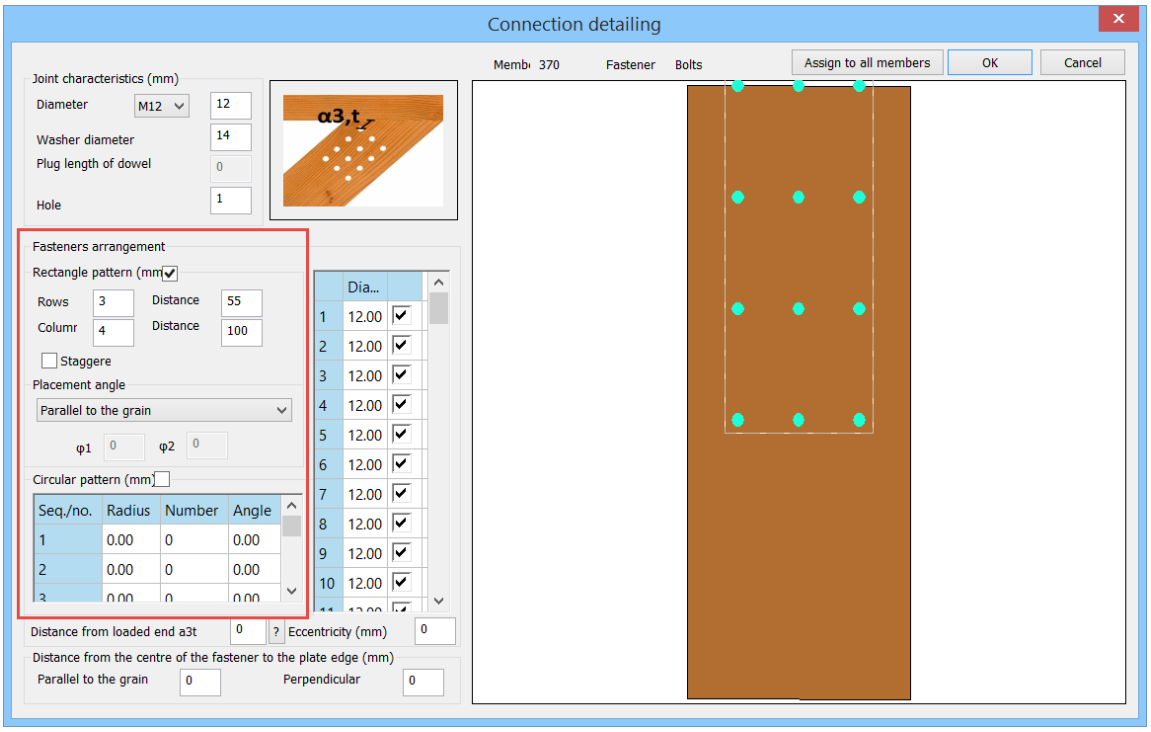


and press **Edit**.





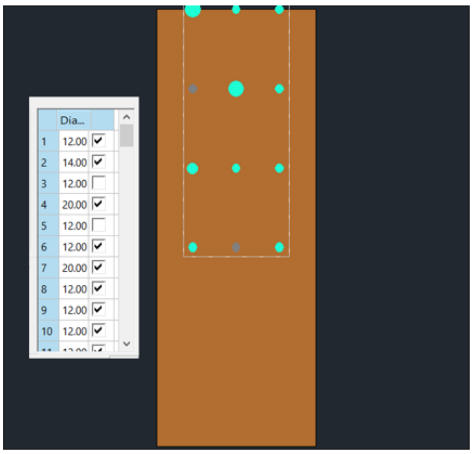
- Select the diameter of the bolt (or type the diameter of the dowel)
- Type Washer diameter (only for bolts)
- Define plug length (only for dowels)
- Type the hole tolerance in mm.



In the field **Fasteners arrangement**:

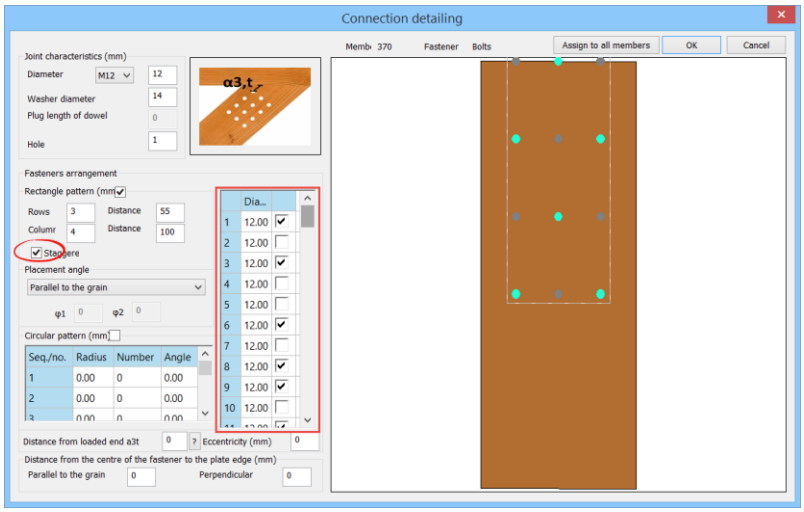
- Activated *Rectangle pattern*:

Type the number of the Rows (parallel to the wood fibers) and the Columns (perpendicular to the wood fibers) and the respective distances. The figure on the right is updated showing the connectors and the outline of the plate.

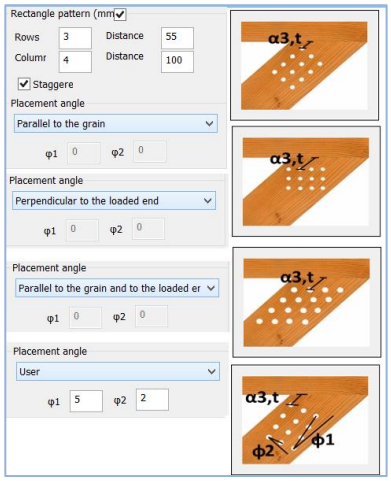


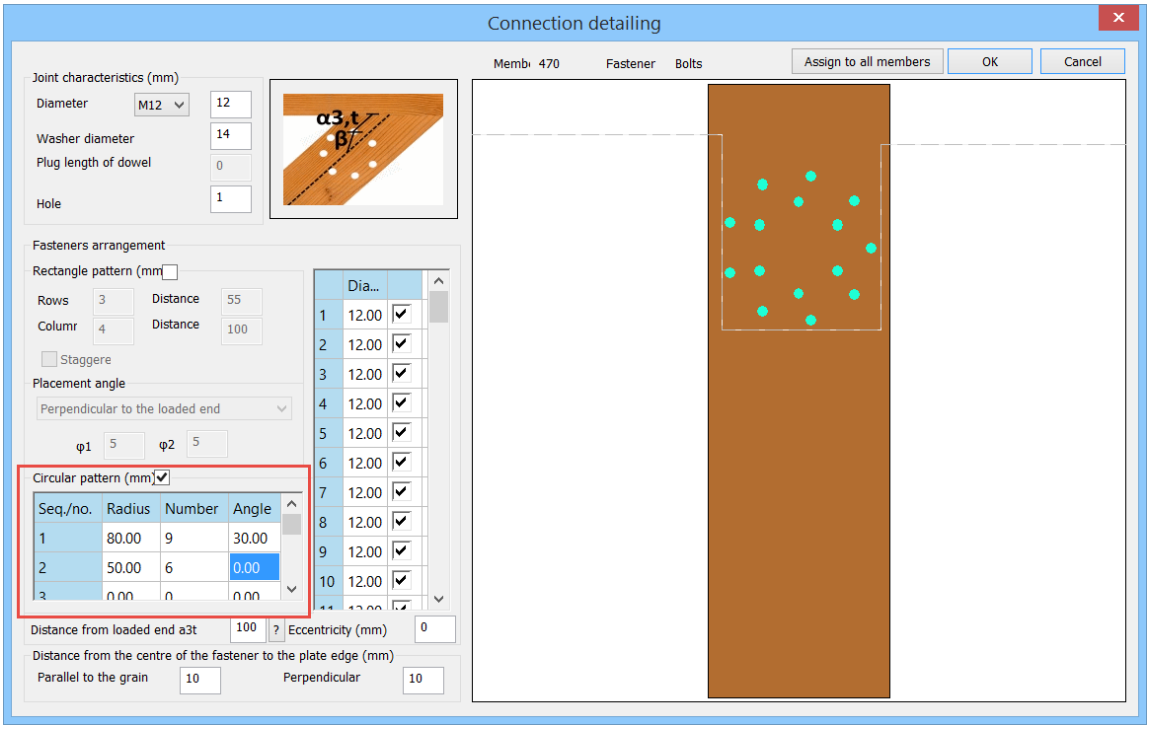
Through the column of the connectors, the user can modify the diameters by typing directly the diameter for the selected connector, either graphically in the figure or the column. There is also the possibility to exclude connectors by deactivating the checks.

The Staggered activation excludes all intermediate connectors.



The **Placement angle** of the connectors is selected from the list and is displayed in the figure.





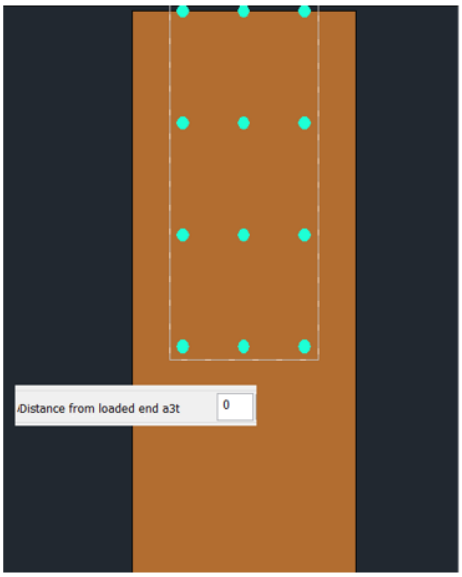
In **Fasteners arrangement** field

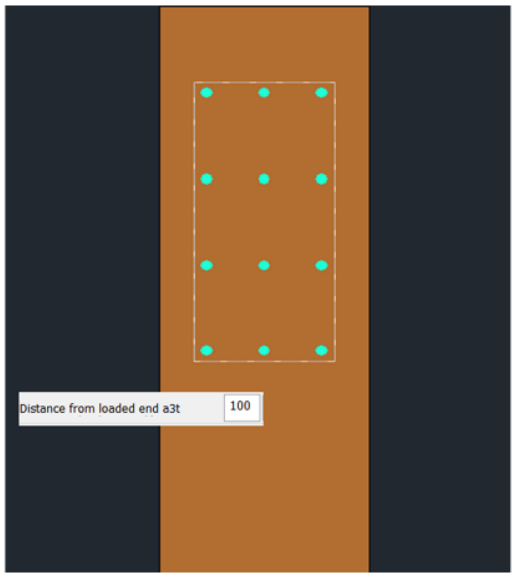
- Activated *Circular pattern*:

Complete the table defining for each circle of connectors the Radius and the Number of the connectors. The angle rotates the corresponding cycle from + x in the counterclockwise direction.

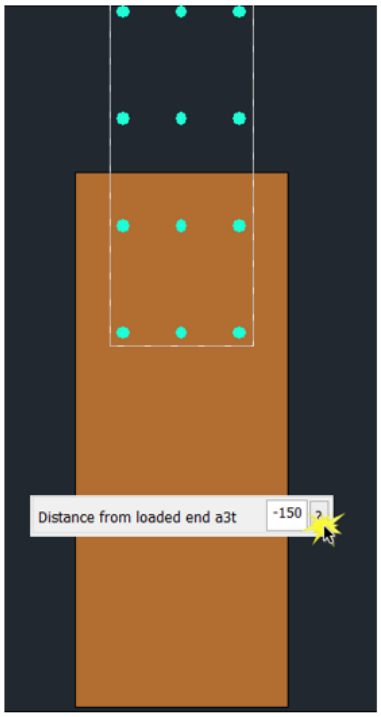
At the bottom of the remaining window, the user can :

- Set the distance a3t (the distance from the edge of the member of the closest to this edge connector, parallel to the fibers of the member)

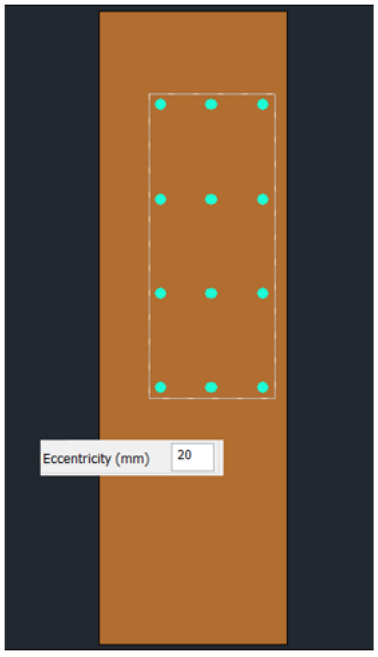




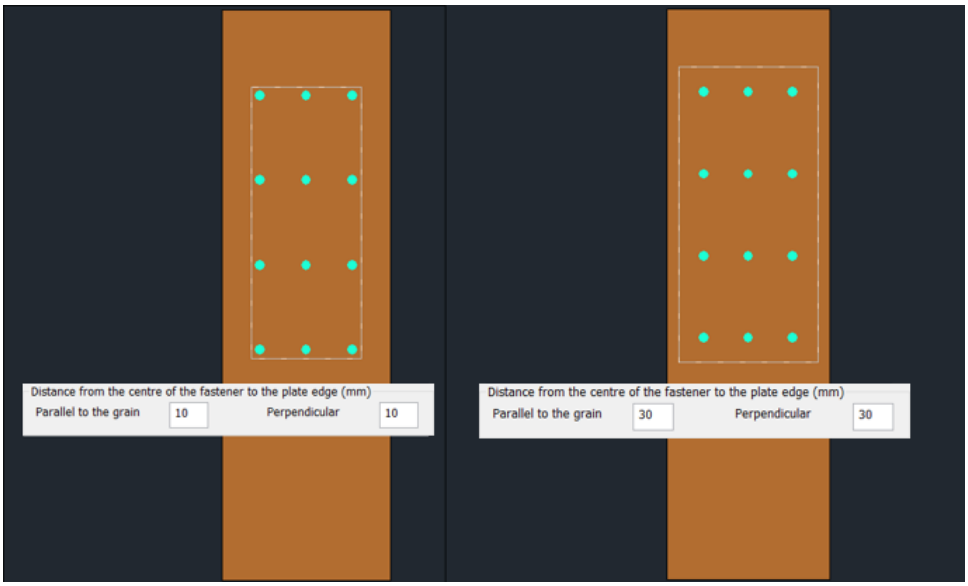
- By pressing**?** The connection center is automatically transferred to the edge of the member.



- **Eccentricity**: transfers the connection by the eccentricity, perpendicular to the fibers of the member.

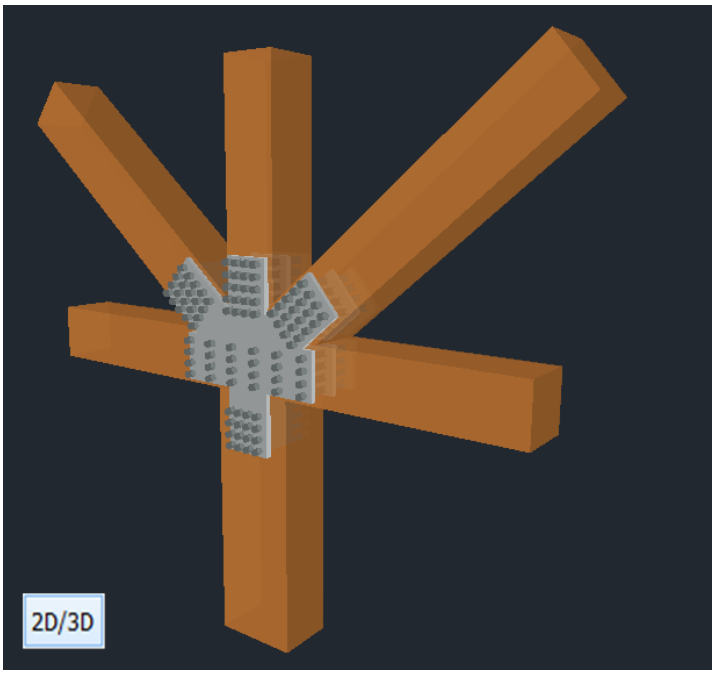


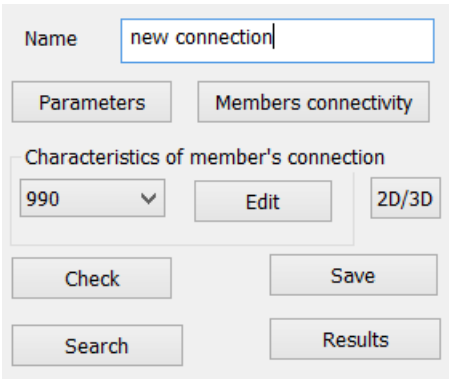
- Modify the vertical and parallel to the fibers of the two ends of the plate distances from the center of the connection.



Assign to all members By selecting the command, all the connecting member details, are applied to the other members of the connection as well.
Click OK to save the choices and close the window.

You can create complex connections with a large number of members and display them in 2D/3D representation.



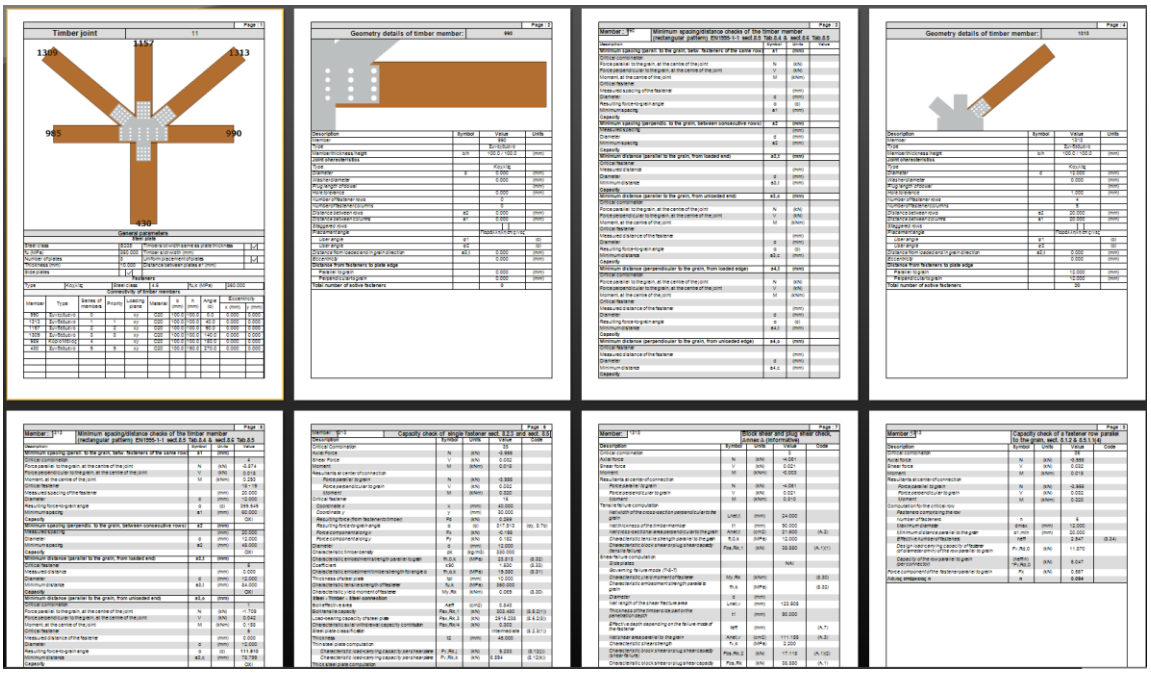


Save : saves connection

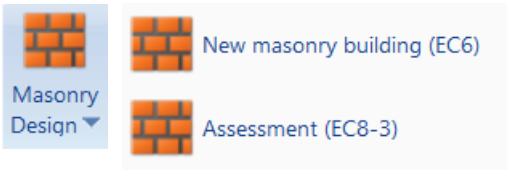
Check : makes the necessary checks according to EC5 (for wood) EC3 (for steel)

Search : displays the checks results in detail

Results : displays the checks results in tables



8. Masonry Design

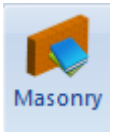


Command to design masonry structures.
Command for the assessment of masonry structures.

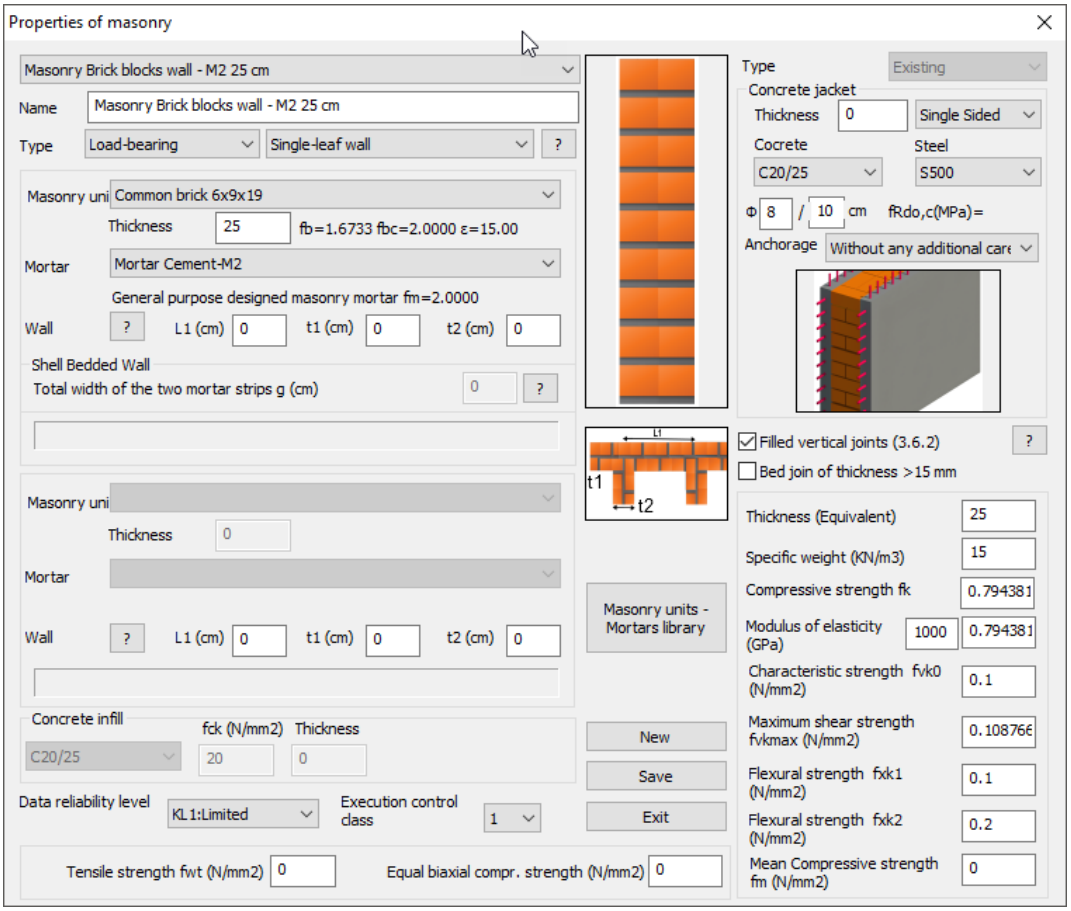
⚠ Basic requirement either to **desin**, or for the **assessment** of masonry structure is to perform first:

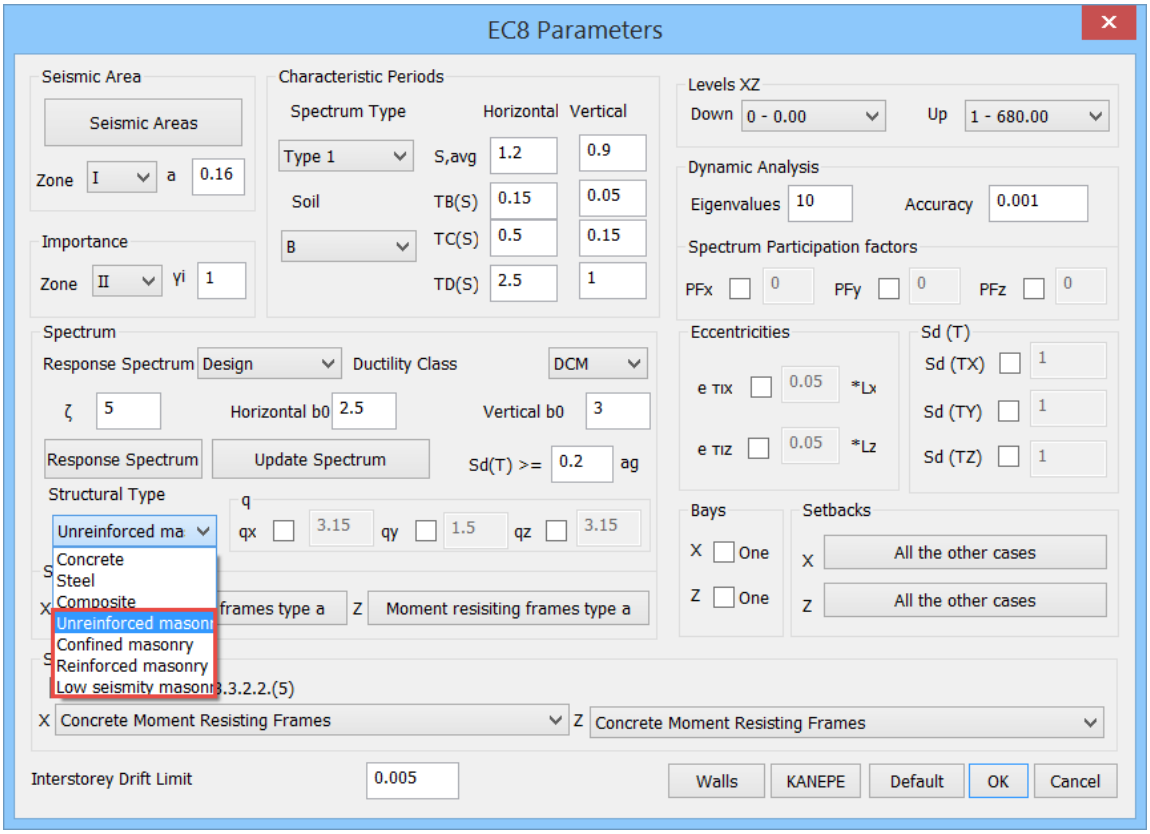
1. The modeling of the structure by using either 3D surfaces finite elements or the templates tool (with or without using command 

)

2. Define the parameters of the masonry 

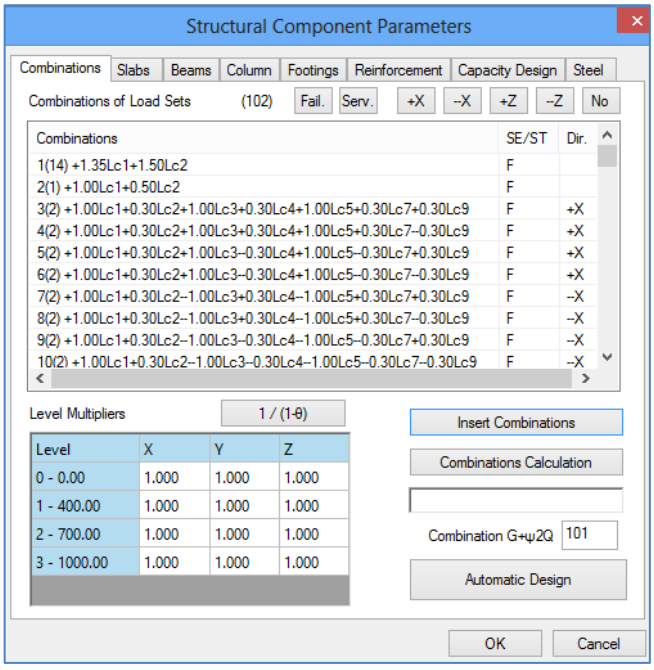
and select the respective "Structural Type" in the analysis "Parameters" for the automatic calculation of "q" factor:





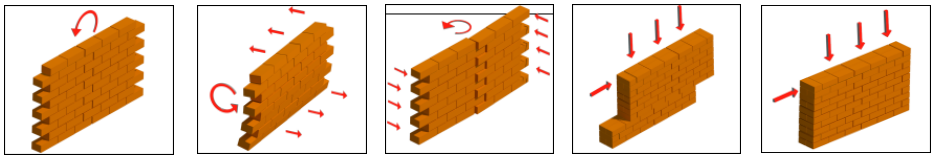
3. A design scenario by Eurocode is produced so all the required checks are incorporated

Open "Members Design" unit and in the "Scenarios" command group choose the "EC2" from the drop-down list as the Active Scenario.
Click on the command "Parameters" and load the combinations file that is automatically saved by the analysis. Insert Combinations, click Calculation and "OK" to close.



8.1 New masonry building (EC6)

Masonry design according to Eurocode 6 includes seven checks:

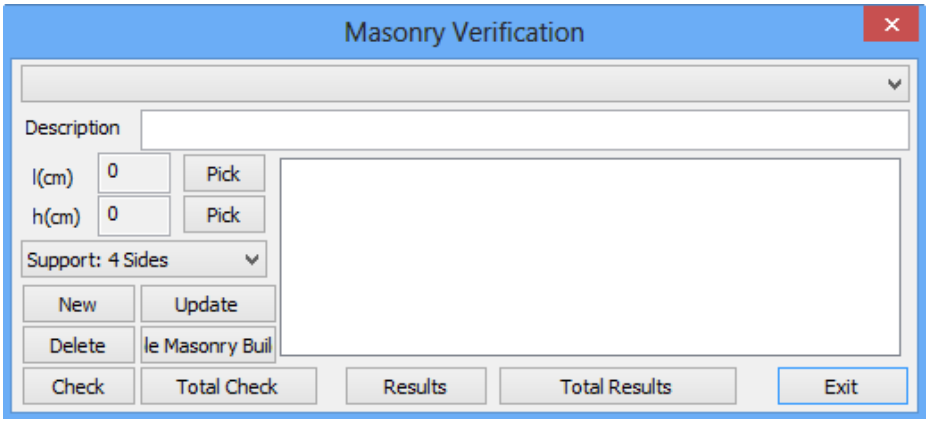


1. Wall subjected to in-plane bending
2. Wall subjected to out-of-plane bending across an axis parallel to the bed joints
3. Wall subjected to out-of-plane bending across an axis perpendicular to the bed joints
4. Wall subjected to shear loading
5. Wall subjected to mainly vertical loading, top
6. Wall subjected to mainly vertical loading, middle
7. Wall subjected to mainly vertical loading, bottom

These seven adequacy checks are defined for each wall or each wall section (spandrel), according to the user defined division.

Buildings that meet the requirements to be identified as "Simple" are excluded from all the above adequacy checks.

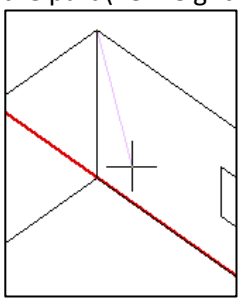
In the dialog box that opens, the user has to identify the parts of the walls to make the required checks:

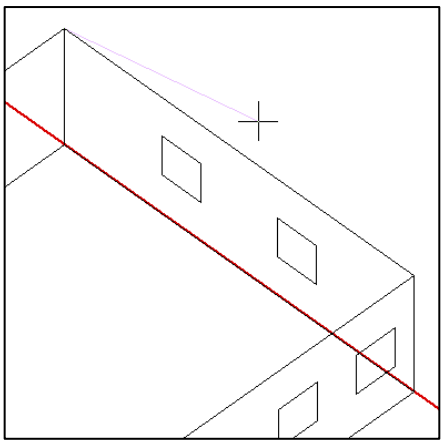


First, define the wall area for the checks:

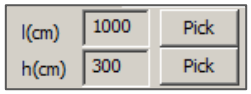
Type a name in the “**Description**” field (with at least three characters).
Press the button “**Pick**” (the first one) to define the x starting and ending points of the part (i.e. length definition). Since the starting point is clicked, an elastic chord emerges from it, waiting to link it with the ending point (second click).

In the same way, press the second button "Pick" (below the first one), to define the y starting and ending points of the part (i.e. height definition).

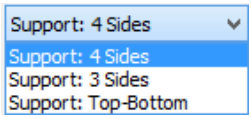




The values are automatically assigned to the fields "l" and "h".



Next, choose the type of the wall

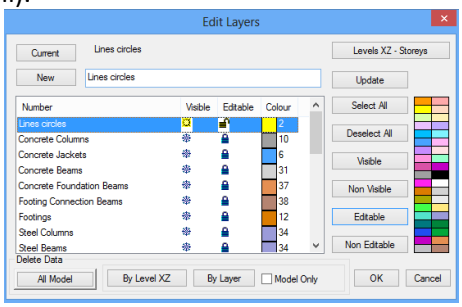


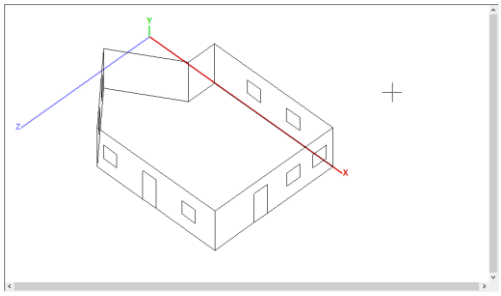
constraint and press the button "New"

to save.

NOTES:

⚠ In order to pick the points easily, take advantage of the object snap utility in a most efficient way by deactivating any layers that "confuse" the picking procedure (e.g. when a whole wall is to be picked, keep active only the layers of "Lines circles" to pick the corner points of the wall).

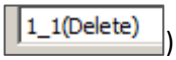




⚠ A selected wall can be recalled from the list and:

- Be modified. Make any change in the name, geometry, type of constraint and then press

to save it.

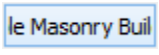
- Be deleted by pressing Delete command. The wall will still be visible in the list but with a Delete marking (e.g. 

).
-

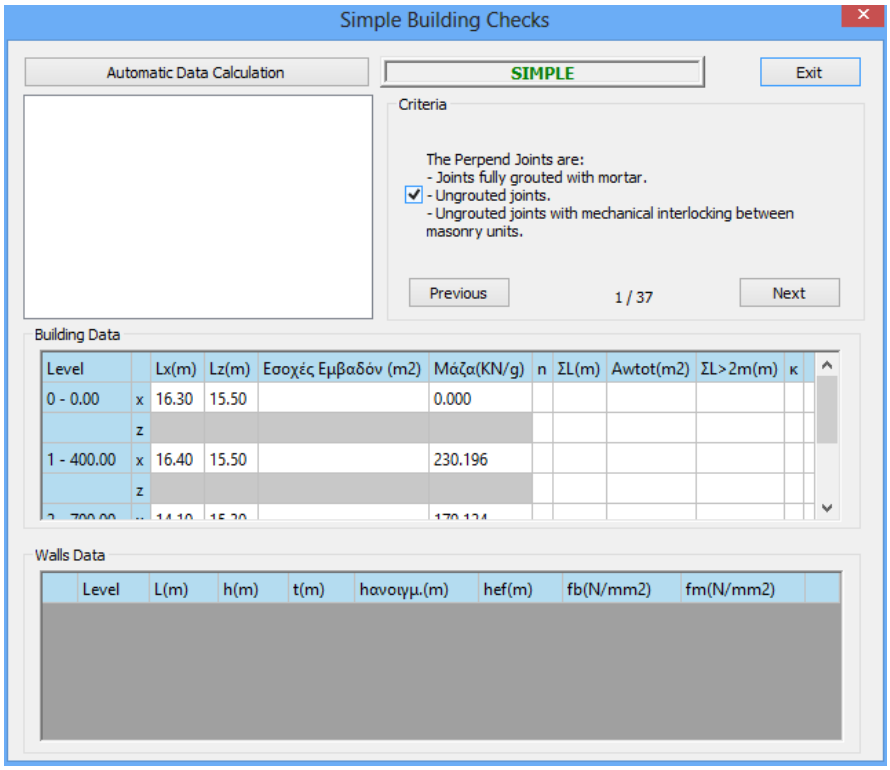
⚠ This is an iterative procedure and is not accomplished until all the walls or parts of walls are defined.

⚠ Before executing the checks, examine the possibility that the building can be defined as "Simple building". In such case, a verification by the checks above is not mandatory (EC8 – 9.7.1).

Simple Masonry Building Checks



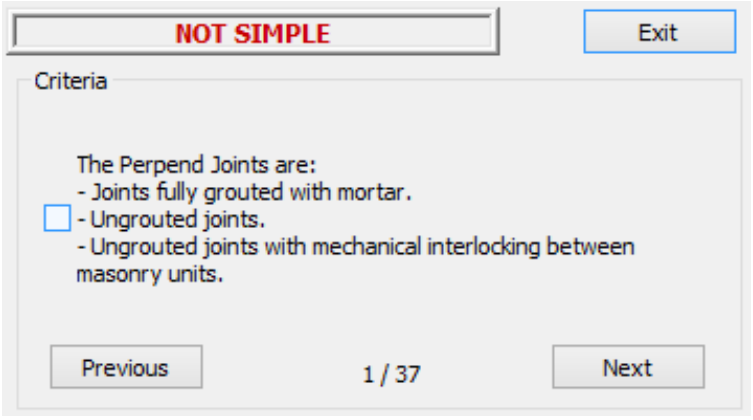
Press the corresponding button and in the dialog box, the following parameters are presented:



The field “Criteria” presents one by one, the 37 requirements according to EC8, for a building to be defined as “Simple”. The user should see a tick next to any of the requirements that are satisfied, and move on to the next one.

NOTE:

⚠ All the requirements must be satisfied, otherwise, the building cannot be defined as “Simple”. As previously mentioned, only in case of a “Simple building”, the design checks of EC6 are optional.



The 37 criteria of the previous stage are the initial steps of the “simple building” characterization procedure. It must also conform to the demands in Table 9.3 of EC8, for the characterization to

be finalized. These demands concern both the building in total and each wall consecutively, and the design check process starts with the command “Automatic Data Calculation” Automatic Data Calculation.

⚠ Again, a failed check means that the building cannot be characterized as “Simple”.

Building Data

Level	Lx(...	Lz(...	Εσοχές Εμβαδόν (...	Μάζα(KN...	n	ΣL(...	Awtot(...	ΣL>2m(...	κ	
0 - 0.00	13....	9.00		0.000	7	13.02	6.51	7.02	1....	NOT SIMP...
					1.	9.19	4.60	3.00	1....	NOT SIMP...
1 - 300...	13....	9.00		147.850	0	0	0	0		
					0	0	0	0		

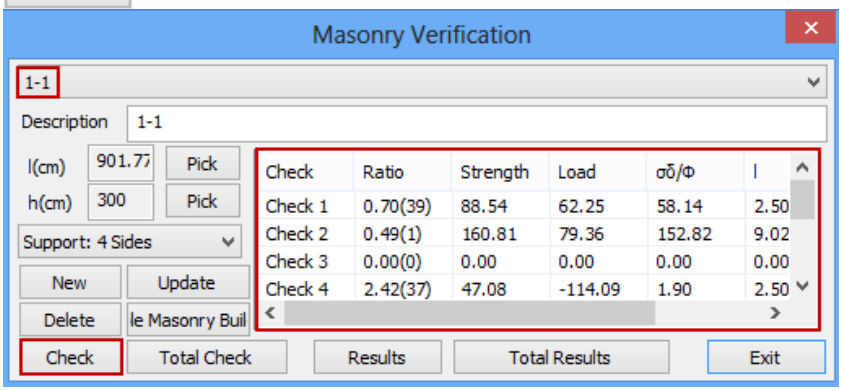
Walls Data

	Level	L(m)	h(m)	t(m)	ηανοιγμ.(m)	hef(m)	fb(N/mm2)	fm(N/mm2)	
1-1	0	9.02	3.00	0.50	1.00	2.70	1.68	2.00	NOT SIMPLE
1-2	0	9.00	3.00	0.50	2.20	2.70	1.68	2.00	NOT SIMPLE
1-3	0	9.00	3.00	0.50	2.20	2.70	1.68	2.00	NOT SIMPLE
1-4	0	6.02	3.00	0.50	2.79	2.40	1.68	2.00	NOT SIMPLE

Check

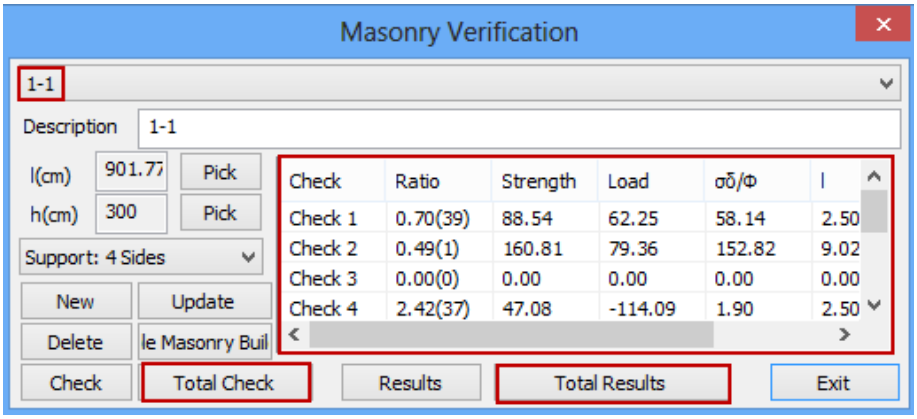
Automatic application of the seven design checks for a selected part of the wall.

Check



Total Check

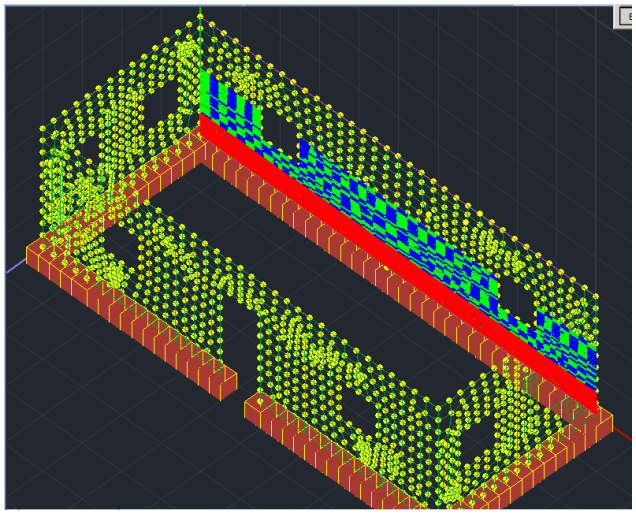
Total Check Automatic application of the seven checks in the structure in total.



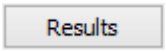
The design checks are applied to sections (horizontal and vertical) concerning the EC6 design code.

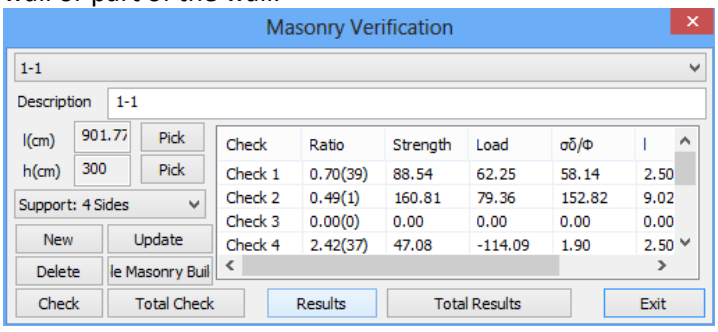
⚠ SCADA Pro scans each selected wall, at first horizontally and then vertically, the wall sections (strips of finite elements) are detected, and all the checks are applied to each section.

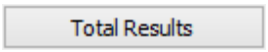
⚠ During the scan, each strip of finite elements is colored according to the results of the design checks; blue-green (all design checks of the section are satisfied) or red (one or more design checks of the sections are not satisfied).



Since the design checks’ procedure has been completed, the user can elaborate on the results.

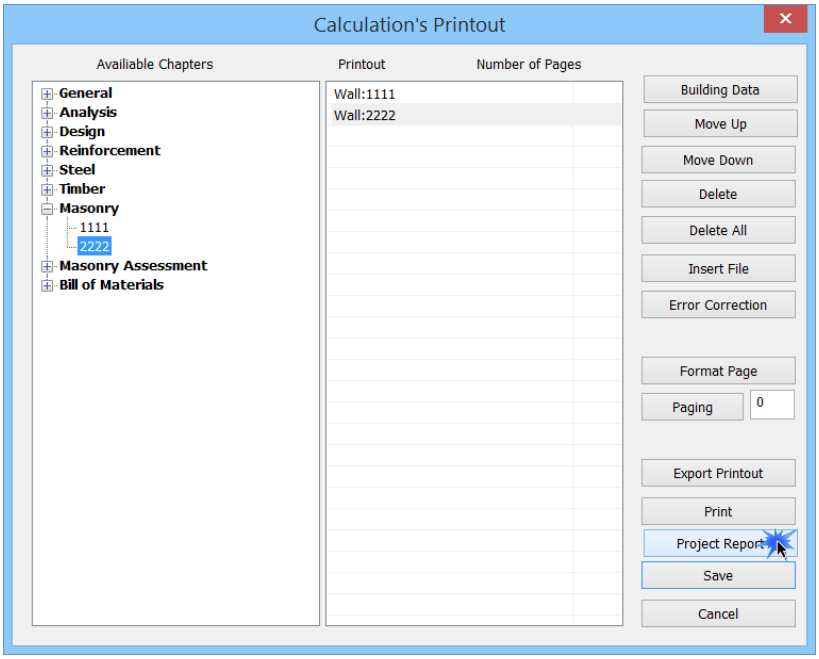
The command “Results”  presents the results of all design checks for the selected wall or part of the wall.

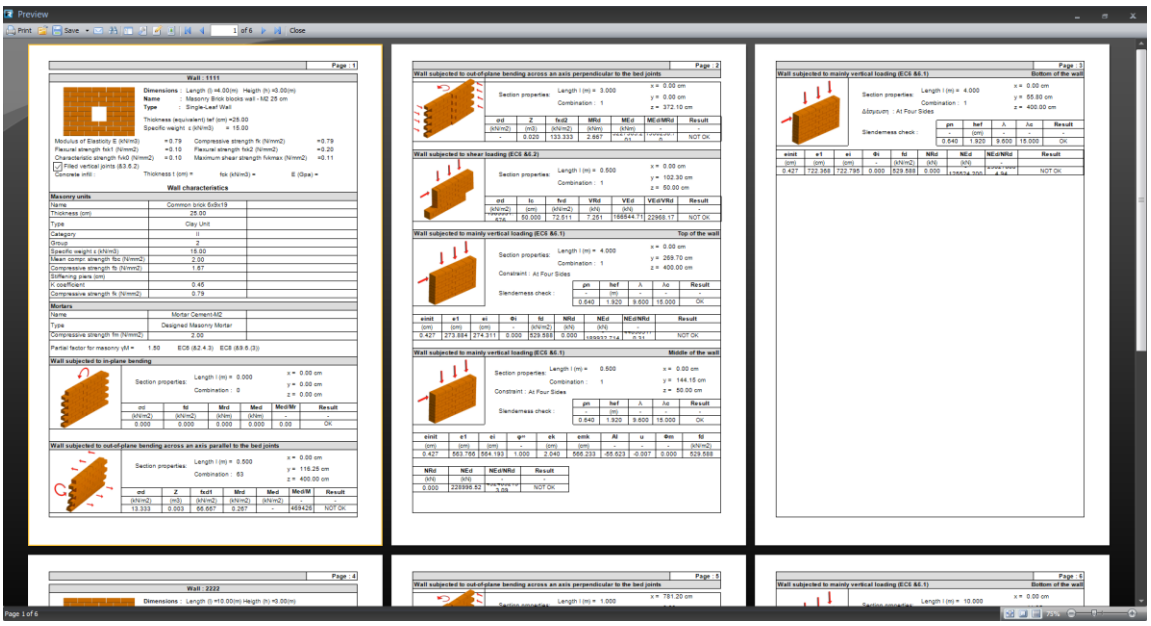


The command “Total Results”  presents the results for all the walls of the building.

Wall	Check 1	Check 2	Check 3	Check 4	Chec
1_1	0.60(1)	0.01(1)	0.02(1)	0.06(1)	1.29(1)
1_3	0.61(1)	0.01(1)	0.02(1)	0.11(1)	0.31(1)
1_4	0.61(1)	0.04(1)	0.01(1)	0.73(1)	0.13(1)
1_2	0.60(1)	0.06(1)	0.01(1)	0.51(1)	0.09(1)

⚠ For better and more detailed appearance of these results view the "Printout"



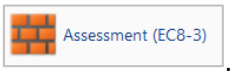


8.2 Assessment (EC8-3)

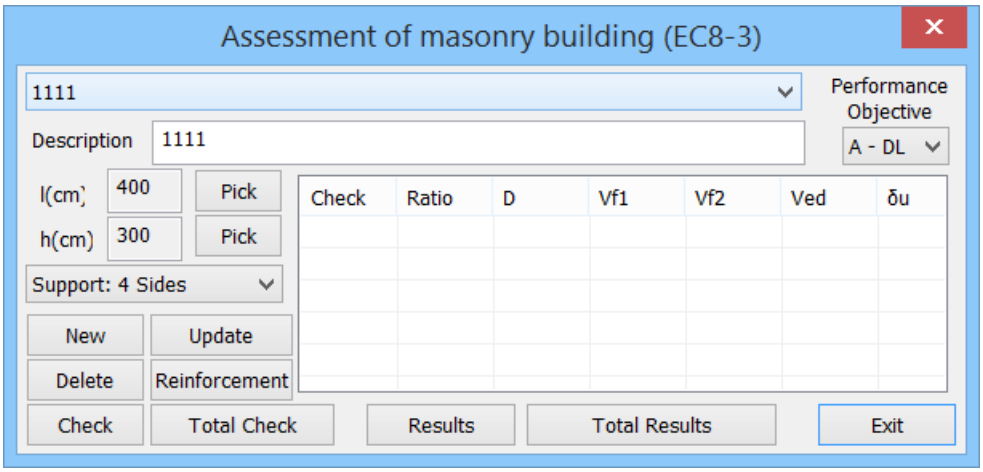
The design checks are applied to the cross section of the pier/spandrel, where the dominant stress resultant is either:

- The axial force and bending moment, or
- The shear force

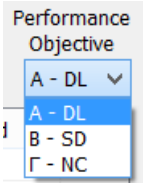
Consequently, the critical failure of the structural element is determined and the corresponding structural capacity is calculated for the three performance levels A, B, C.

After all the preliminary procedure select 

.
In the dialog box that opens, the user must identify the parts of the walls to make the required checks as for the **"New Masonry Building"**:

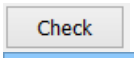


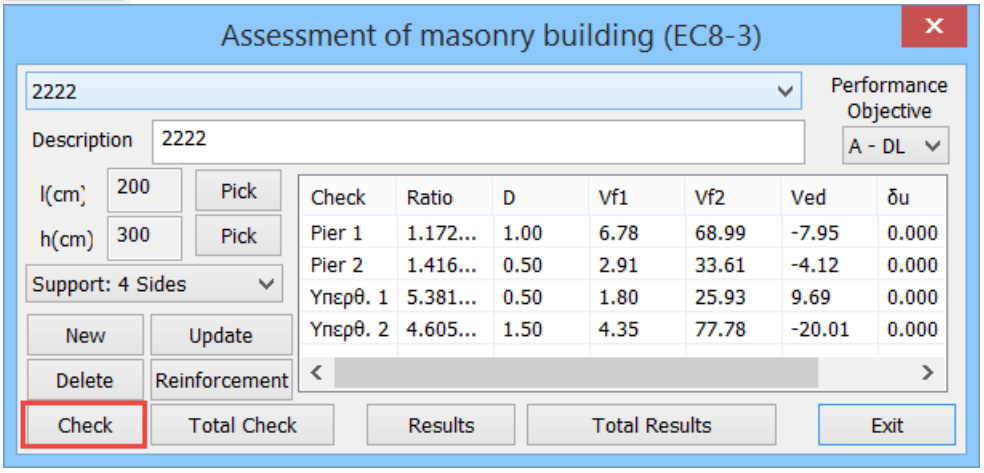
⚠ **NOTE** that:
The program automatically performs recognition of piers/spandrels. Therefore set the entire wall with the openings and the program checks automatically (separately) the piers and the spandrels (means wall sections above and below the openings)

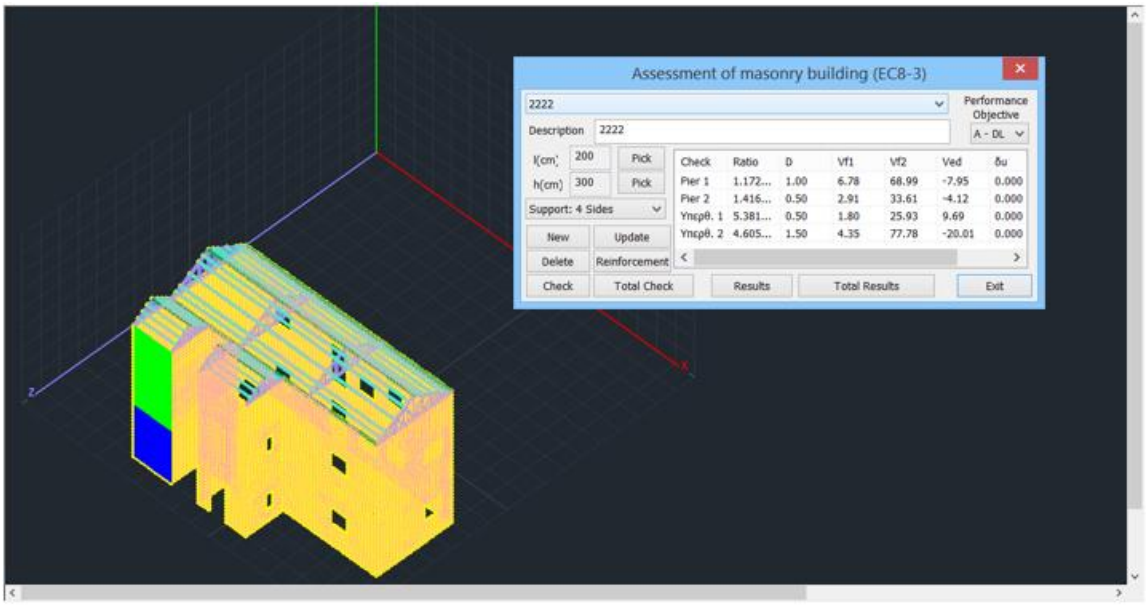
Choose the Performance level 

and then make the Checks.

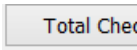
Check to make all the checks for the selected wall for each pier/spandrel section.

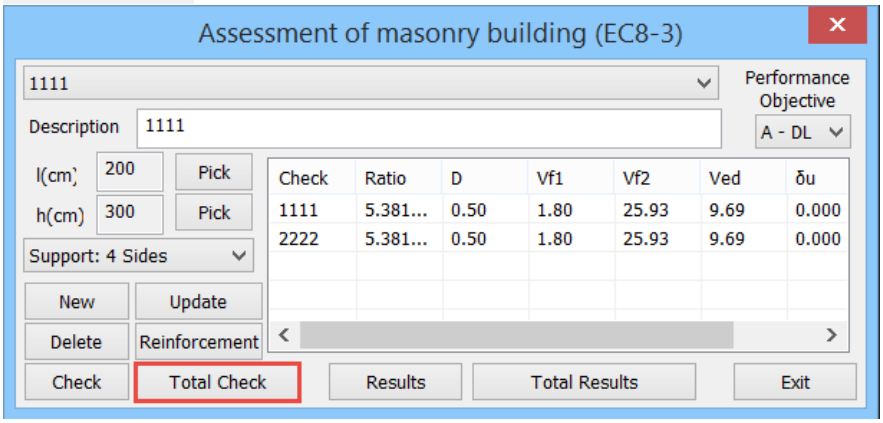


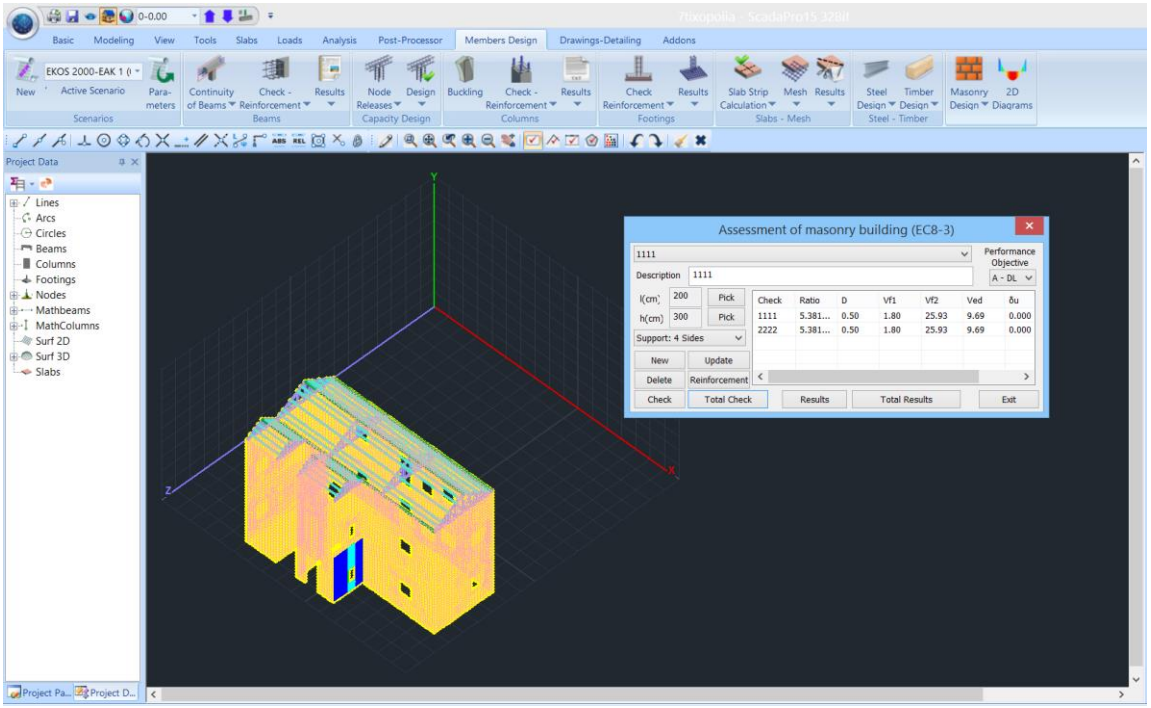




Total Check to make all the checks for all the defined walls and each pier/spandrel section.







⚠ The adequacy checks are performed per pier/spandrel section regarding forces and deformations depending on the Performance Level.

8.2.1 Masonry Reinforcement

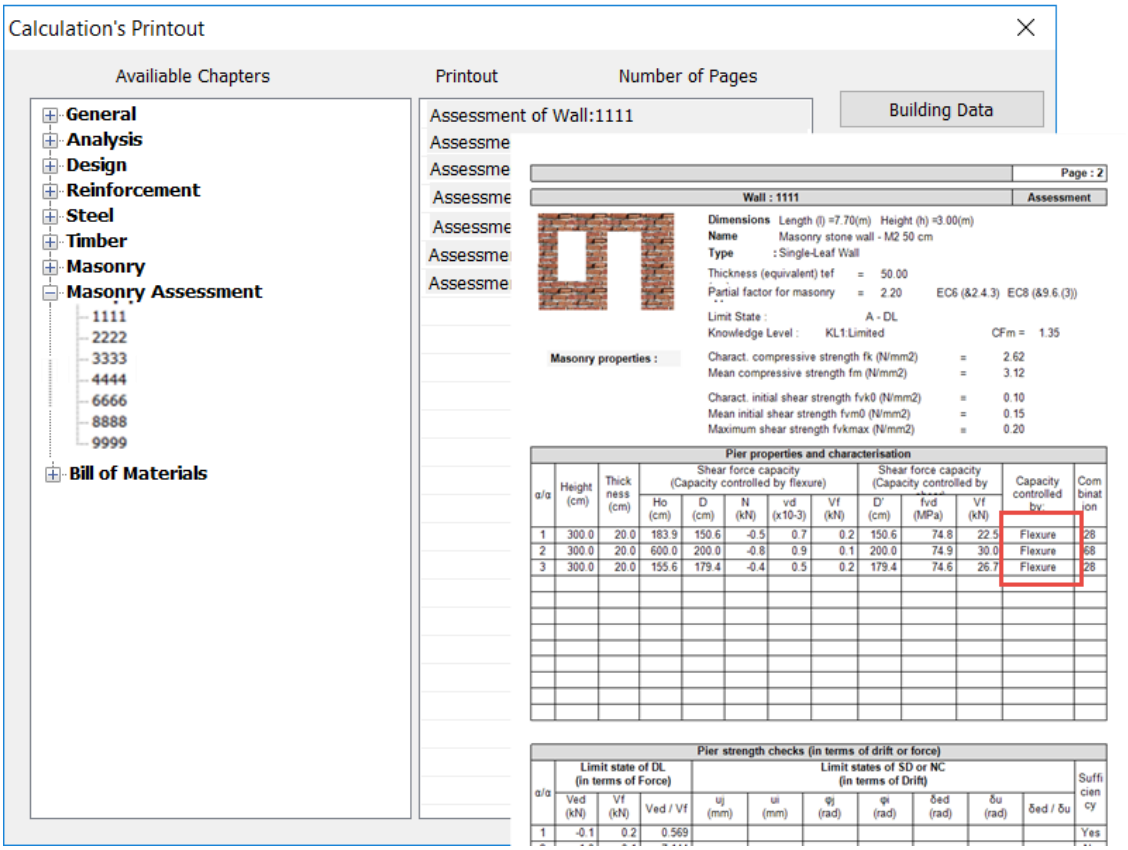
SCADA Pro offers the possibility of reinforcing the masonry with:

- simple or double of the reinforced concrete jacket to increase the compressive, shear and flexural strength of the element
- Textile Reinforced Mortar for shear reinforcement in a plane
- Also, in cases of reinforcing with **Deep Jointing** or **Grouting**, you define the compressive strength of the of the reinforced masonry according to the corresponding formulas:

$$f_{wc} = \frac{1}{\gamma_{Rd}} \cdot \zeta \cdot f_{wc,o}$$ (Deep Jointing)

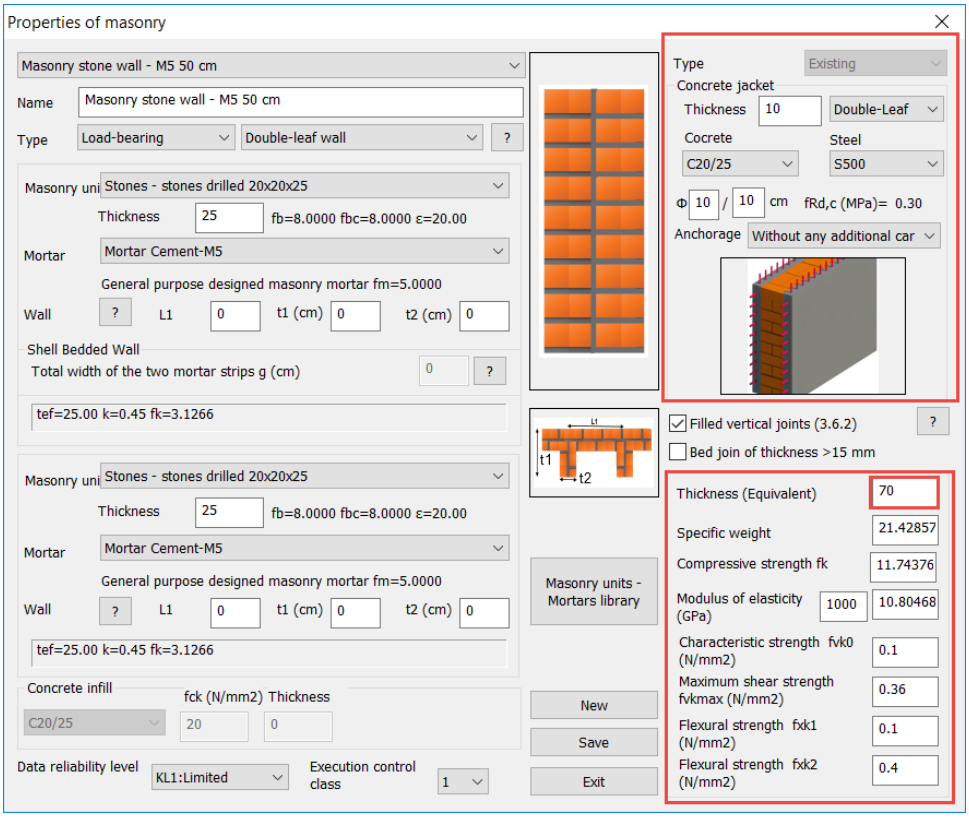
$$f_{wc,i} = f_{wc,0}\left(1 + \frac{V_i}{V_w}\frac{f_{c,in}}{f_{wc,0}}\right)$$ (Grouting)

After completing the checks through the "Masonry Assessment" printing files, you can read the characterization of fault resulting and reinforce accordingly.

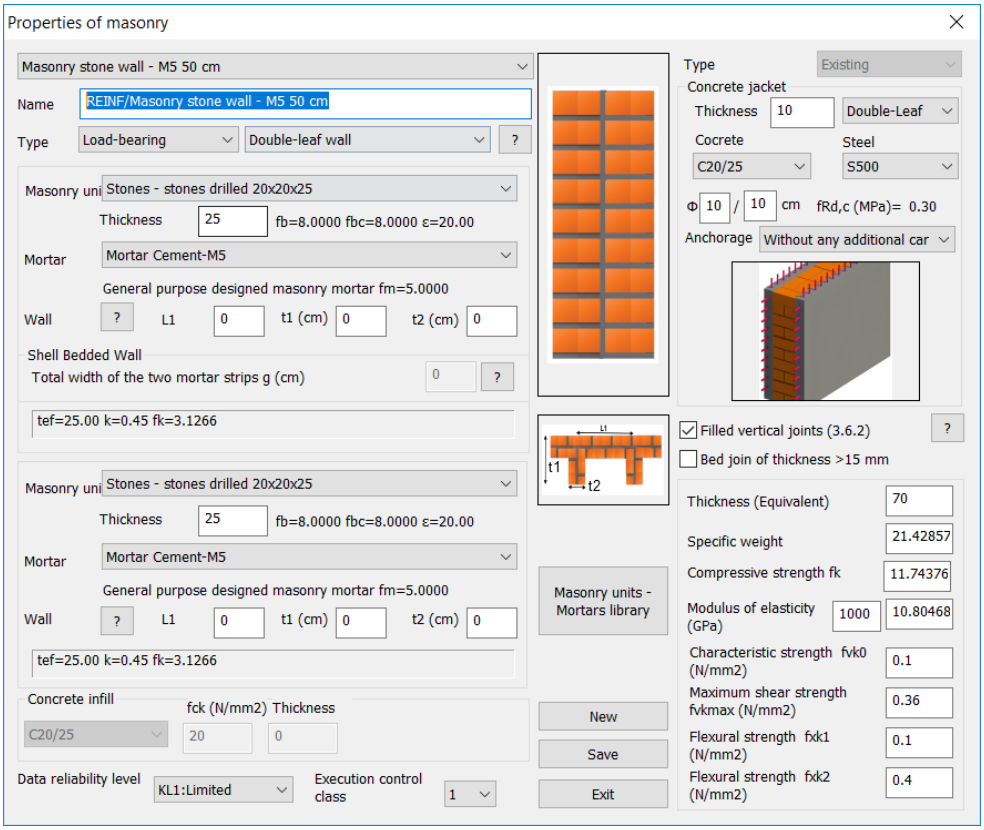


8.2.1.1 Reinforcement with a concrete jacket

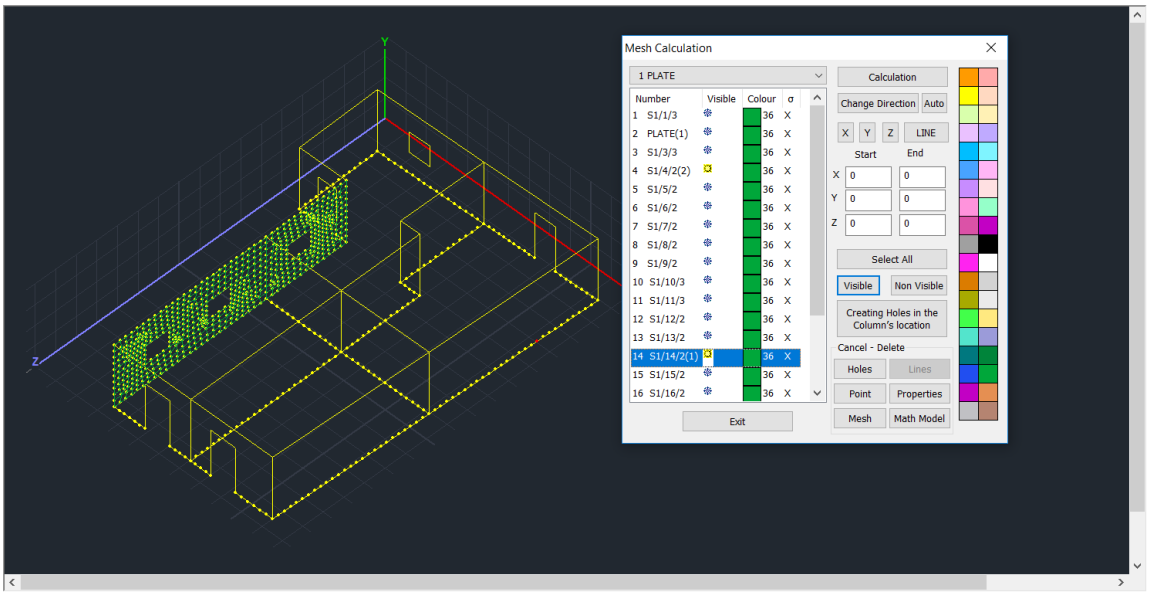
To reinforce a wall with a single or double jacket, in “Masonry” Library define the characteristics for the concrete jacket. Automatically change all the characteristics of the initial wall also.



Set a new name for the reinforced element and save it for using it later, defining the reinforced wall.

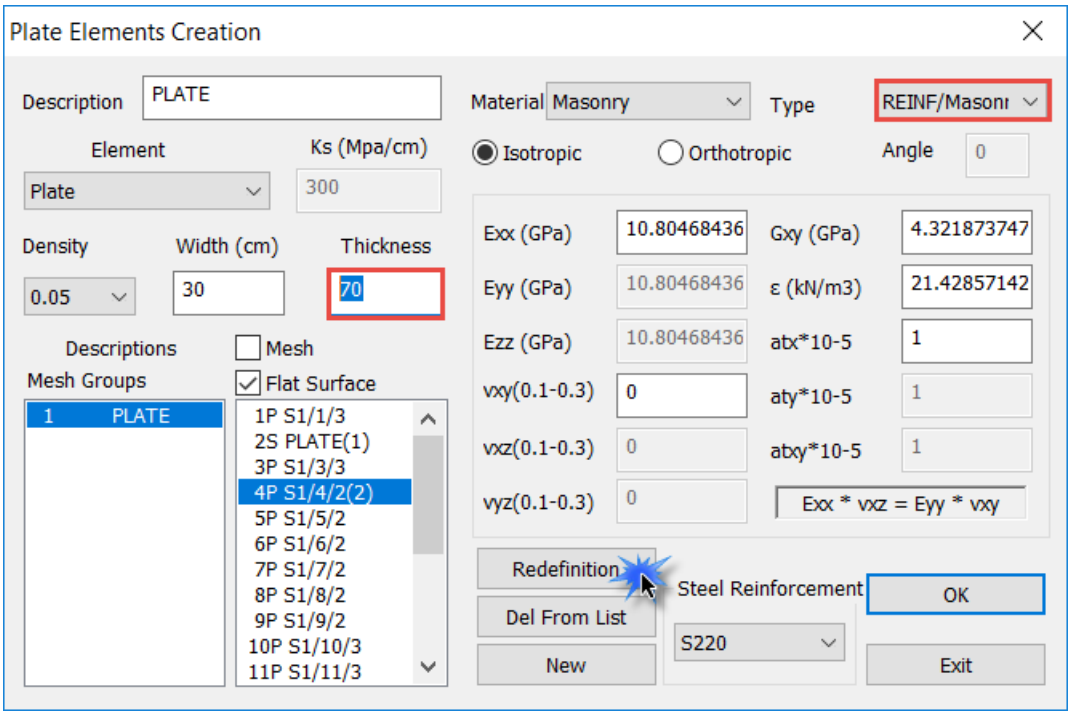


Open Mess 3D command and Calculation to identify the sub-groups that need reinforcement:



Then inside the window of the mesh group identify the sub-groups and modify their **quality** and **thickness:**

⚠ Remember to press **Redefinition** every time you modify something



Then, repeat the analysis process, updated with new data, and check again the reinforced wall to receive new adequacy ratios, until you manage to get ratios smaller than one. The process is iterative and can be done repeatedly.

§ WALLS WITH CONCRETE JACKET – NOTES:

What is affected?

The assignment of concrete jackets affects the following:

- The Equivalent Thickness
- The Specific Weight
- The Modulus of Elasticity
- The Characteristic Compressive Strength
- The Characteristic Shear Strength

Note: Since the equivalent thickness and the Modulus of Elasticity change, the tension of the elements is different than the non concrete jacket ones. As a result, you will have to change the thickness of the surface elements and run the analysis again.

What checks are being made?

The checks being made are the same as those on walls without a concrete jacket. That is to say, the provisions of Eurocode EC8-3 (annex C) which concern the following are applied:

- In-plane shear
- In-plane bending

Which parameters change?

The assignment of concrete jackets brings out the following changes:

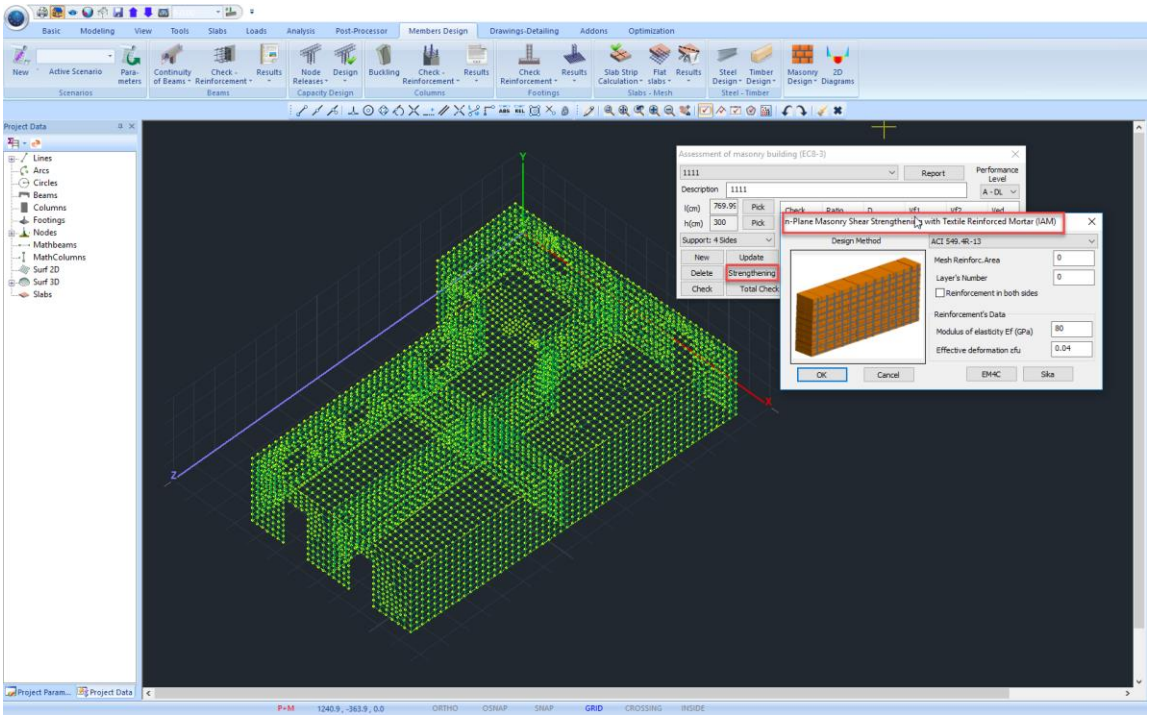
-Equivalent Thickness
-Specific Weight
-Compressive Strenght
-Characteristic Compressive Strenght
-Modulus of Elasticity

It is obvious that some parameters do not change. The reasons why this happens is because:

1. Either they are not used or they are not necessary for the checks of the EC8-3.
2. These are parameters which do not change (eg shear strength of a masonry without loading), but they are used or needed for the EC8-3 checks.

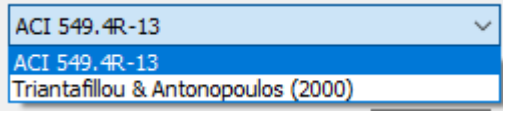
In the evaluation printout, we see similar differences.

8.2.1.2 Textile Reinforced Mortar (TRM)



Use Textile Reinforced Mortar for shear reinforcement in the plane, defined by the corresponding window for the selected wall from the list.

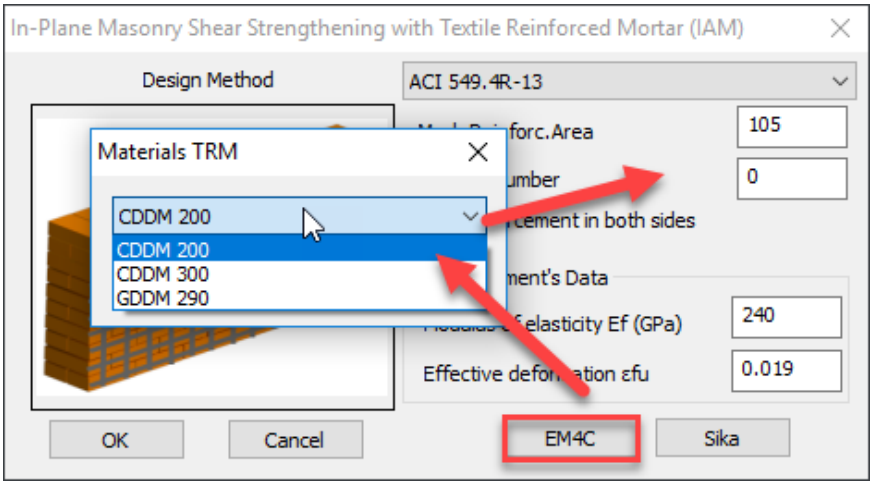
Select the “Design Method”.
To SCADA Pro contains two methods and you can select between

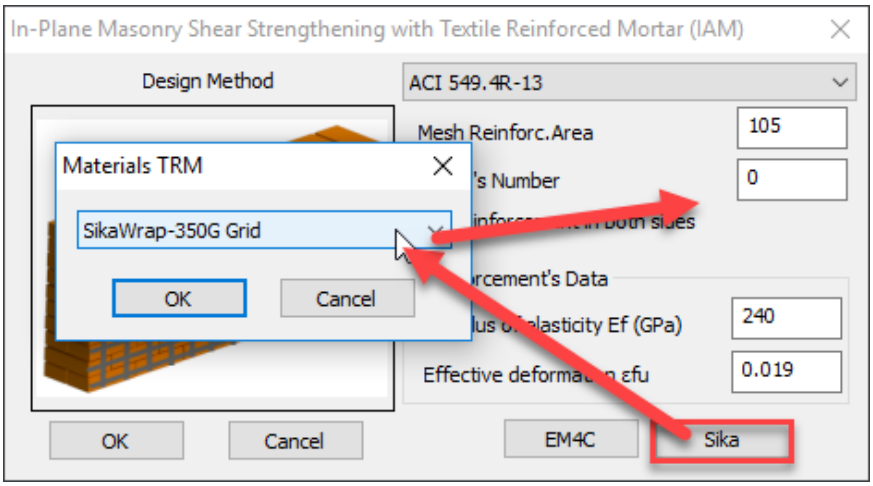


Specify the characteristics of the mesh, based on catalogs and commercial materials.

⚠ In SCADA Pro company materials have been introduced 

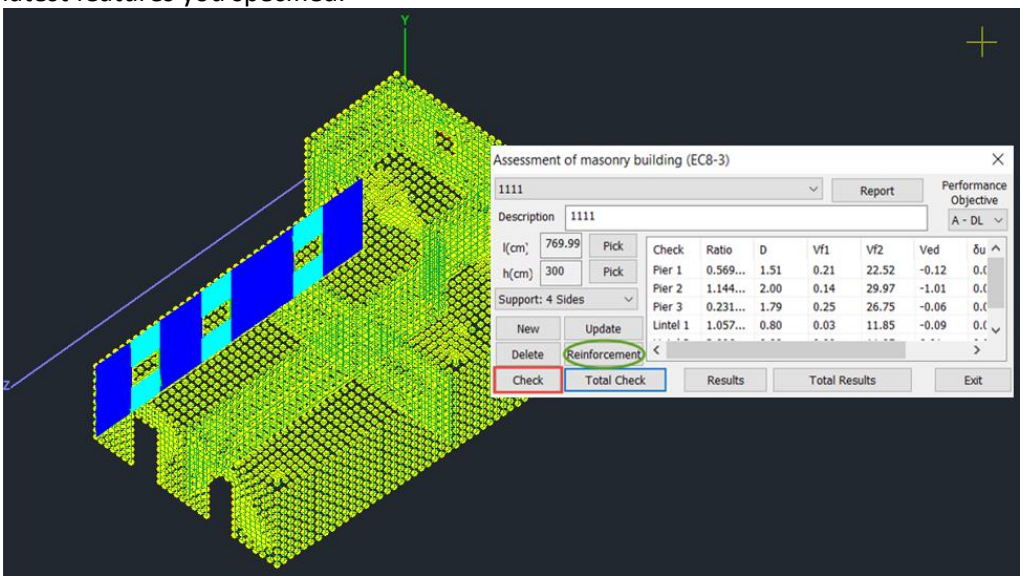
By selecting the company and the corresponding material the mesh features are automatically filled in by the program.



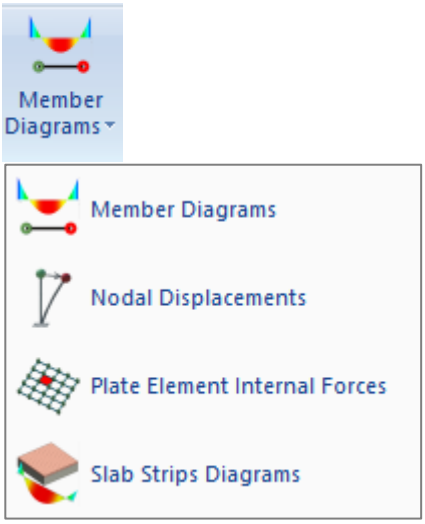


Then press again the "Checks" button and check the results obtained after the introduction of the grid.

You can repeat the process. The program is making checks each time to take into account the latest features you specified.



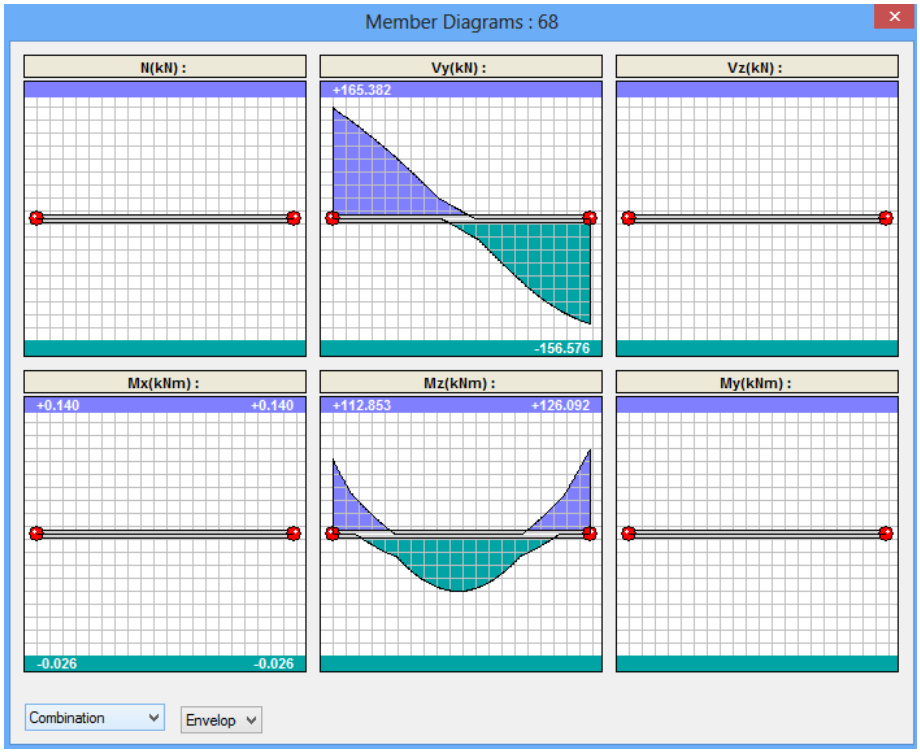
9. Member Diagrams



9.2 Member Diagrams

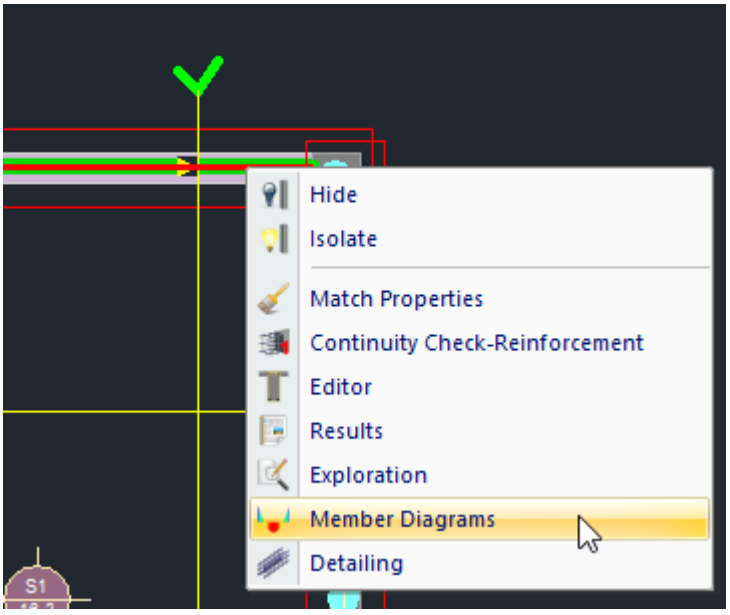
This command is used for the display of the structural members’ stress resultants diagrams in 2D.

Press the command and then left click to select a structural member. In the dialog box, all the diagrams of the stress resultants of the corresponding member are displayed. Select a load combination or a load case and move the mouse in the framework of the diagrams to read the corresponding values of the stresses resultant along the member.



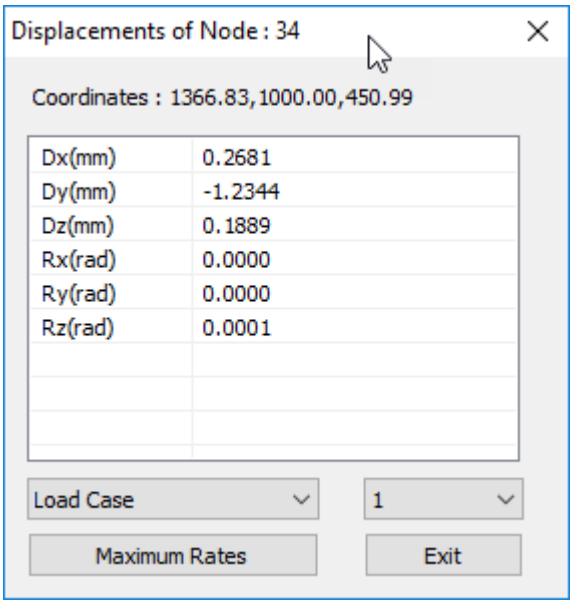
NOTE

Right mouse click on a member opens a command list containing also Members Diagrams.

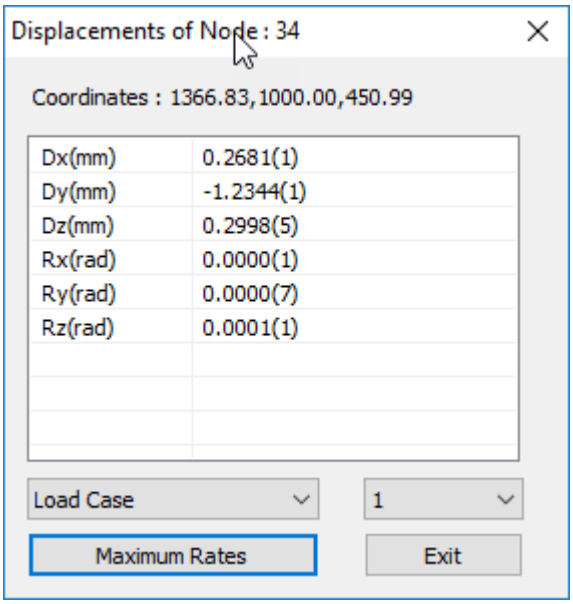


9.3 Nodes Displacements

There is now the possibility to see the nodes displacements.
Select the command and point the mouse on a node. In the dialog box that opens, all the movements of that node are displayed. Select a combination or load to read the corresponding values.

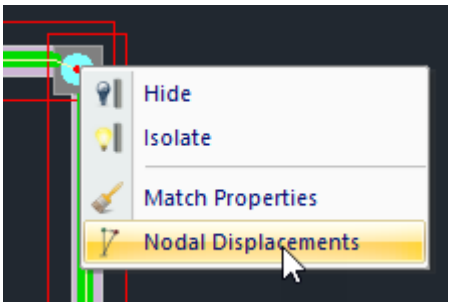


Press Maximum Rates to see the maximum value for each movement and rotation, as well as the combination from which it originates. The number of this combination is written in brackets next to the value.



NOTE

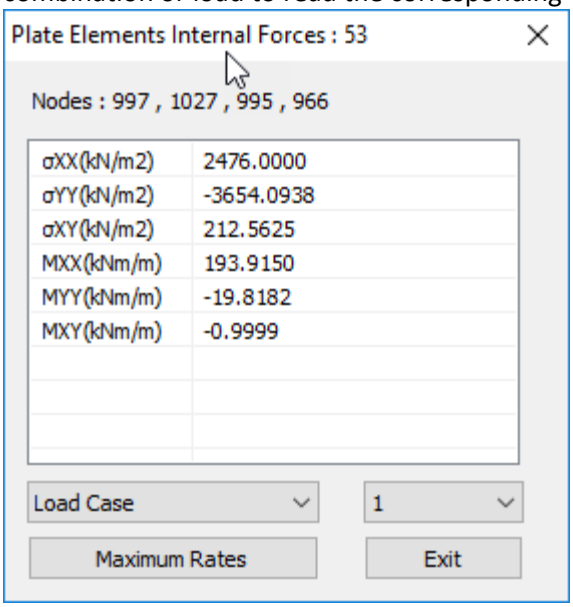
Right mouse click on a member opens a command list containing also Nodal Displacements.



9.4 Plate Elements Internal Forces

There is now the possibility to see the Internal Forces in Plate Elements.

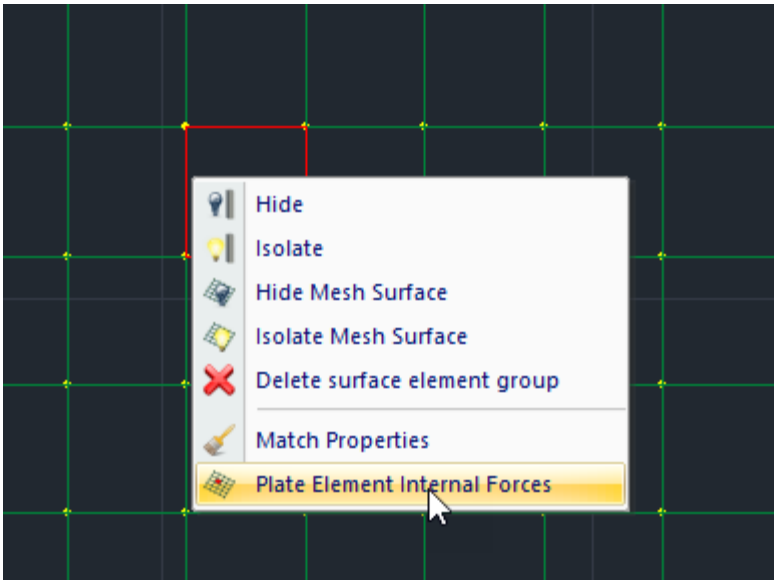
Select the command and point the mouse on a plate element.
The dialog box that opens display all stresses and moments for that plate element. Select a combination or load to read the corresponding values of stresses and moments.



Press Maximum Rates to see the maximum value for each stress and moment, as well as the combination from which it originates. The number of this combination is written in brackets next to the value.

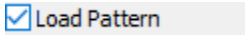
NOTE

Right mouse click on a node opens a command list containing also Plate Element Internal Forces.



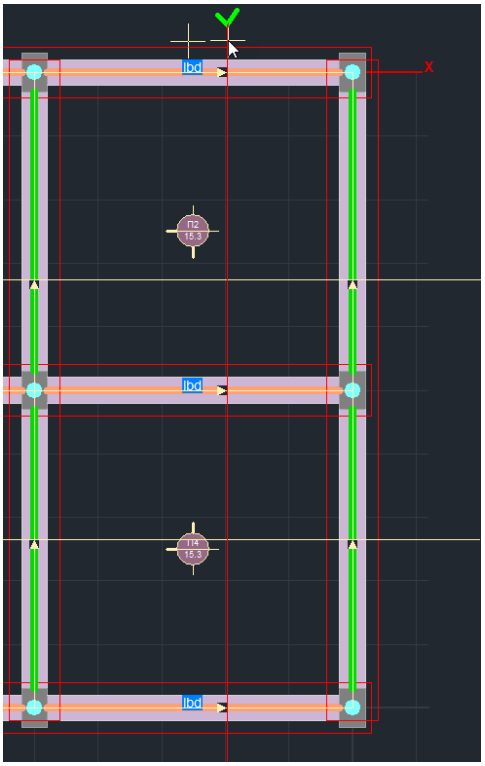
9.5 Slab Strips Diagrams

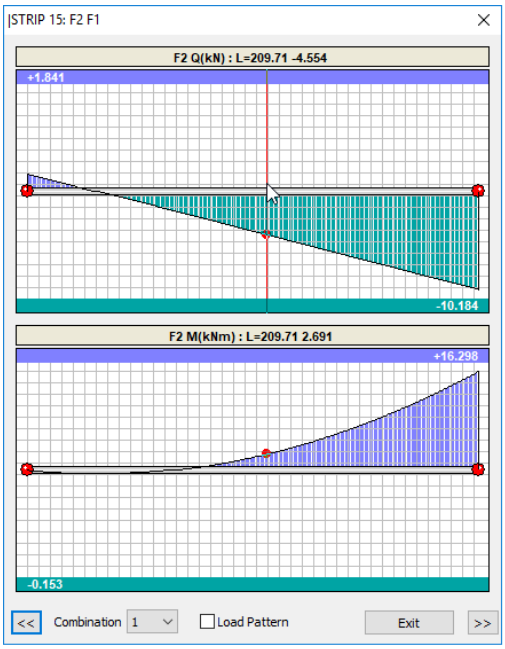
There is now the possibility to see the Slab Strips Diagrams.

Select the command and point the mouse on a Slab Strip.
The dialog box that opens display all diagrams of the internal forces for each span. Select a combination or load and move the mouse on the diagrams to read the corresponding values along the section.
In case of Load Pattern, activate 

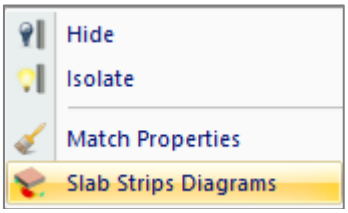
to see the diagrams resulting.

Move to the next span using the arrows on the bottom. 





NOTE



Right mouse click on a node opens a command list containing also the Slab Strips Diagrams.