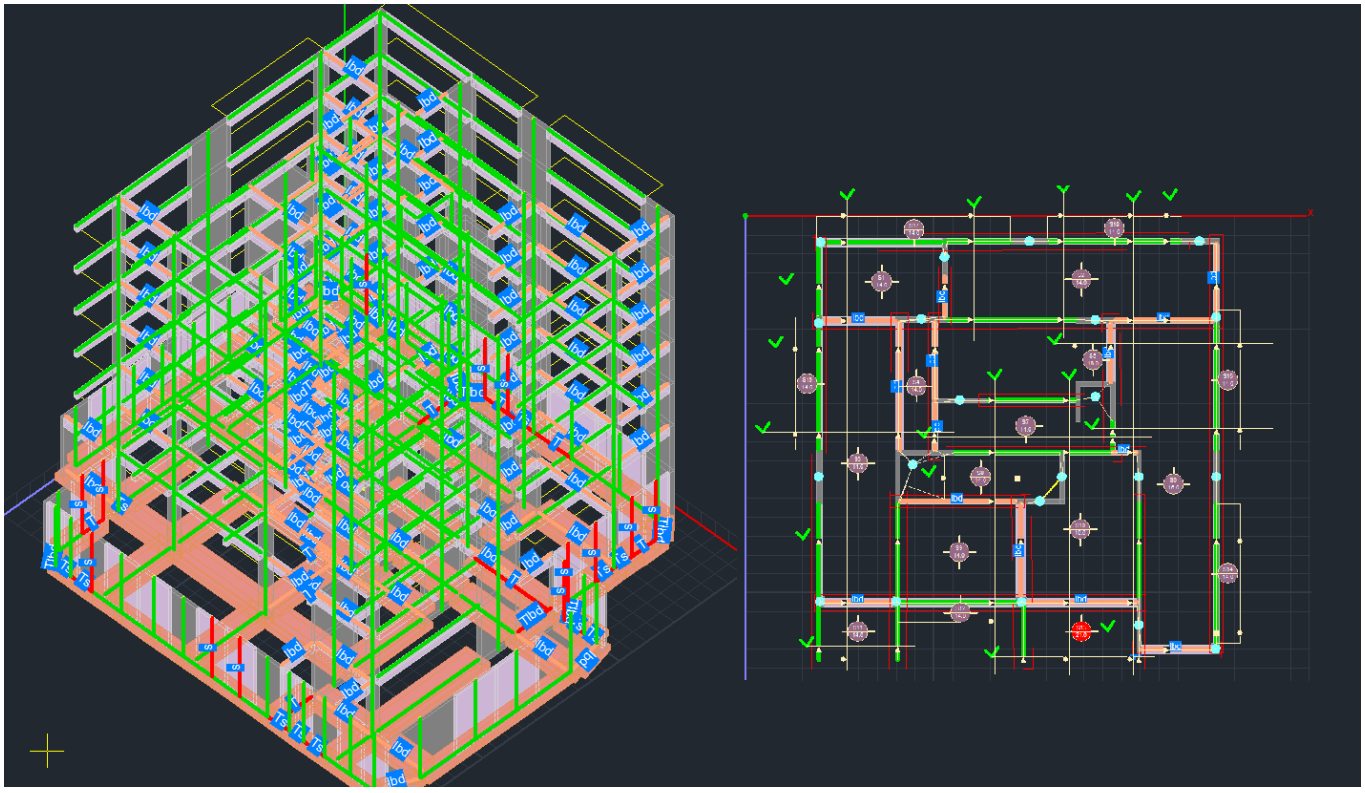
SCADA Pro™ 18
Structural Analysis & Design

User's Manual

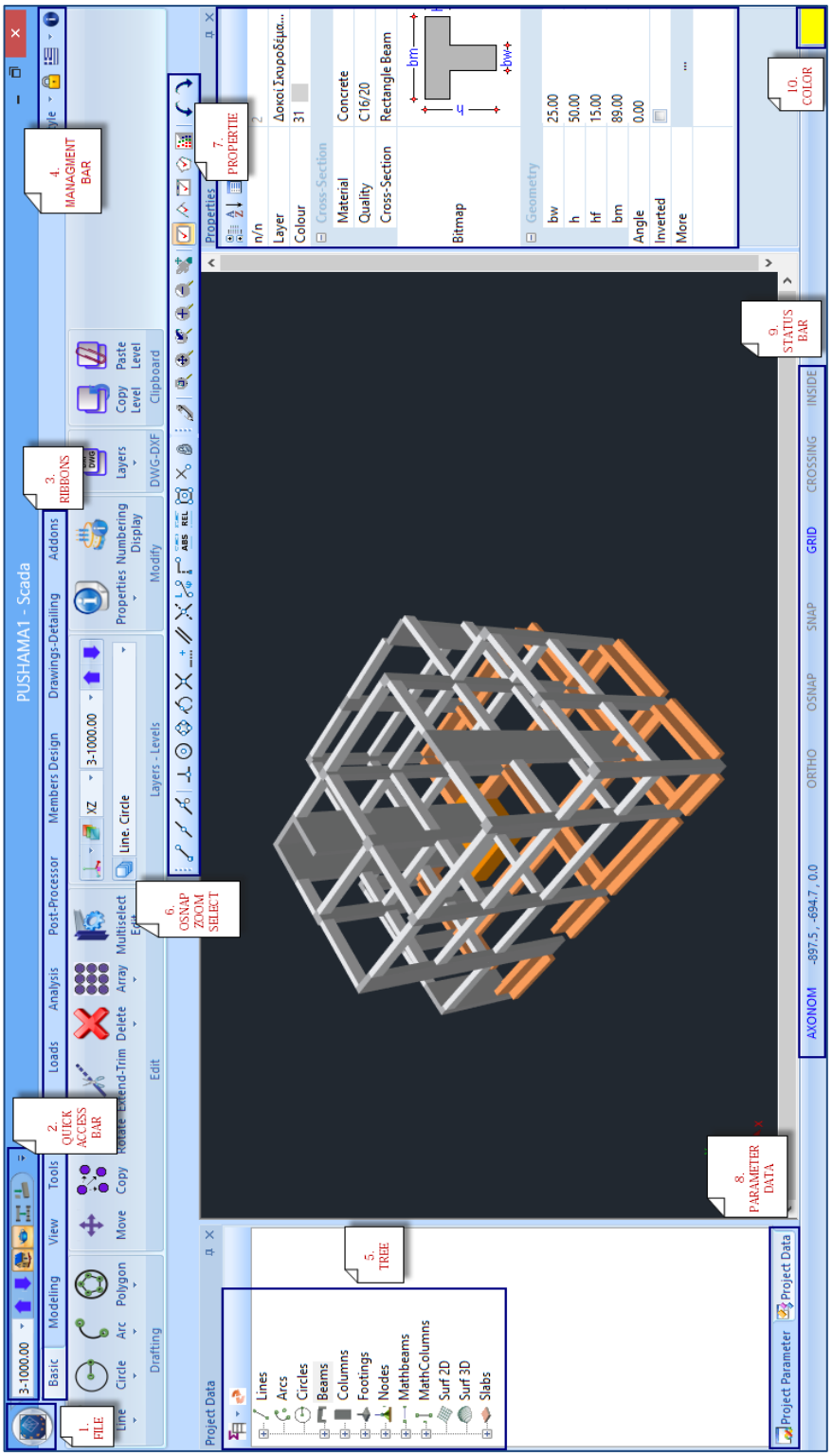
9A.MEMBERS DESIGN
SBC304_17_CONCRETE



ACE HELLAS
INTEGRATED SOLUTIONS

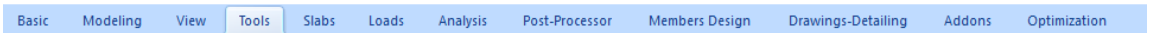
CONTENTS

I. THE NEW UPGRADED INTERFACE of SCADA Pro

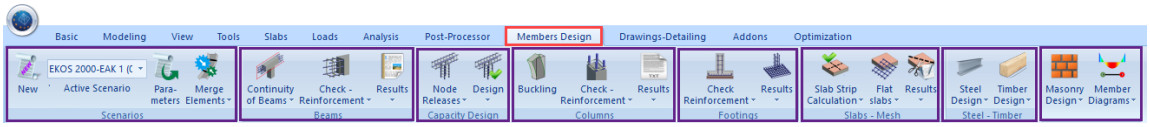


II. DETAILED DESCRIPTION OF THE NEW INTERFACE

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



Members Design



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

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

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

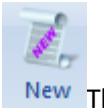
---

1. Scenarios

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

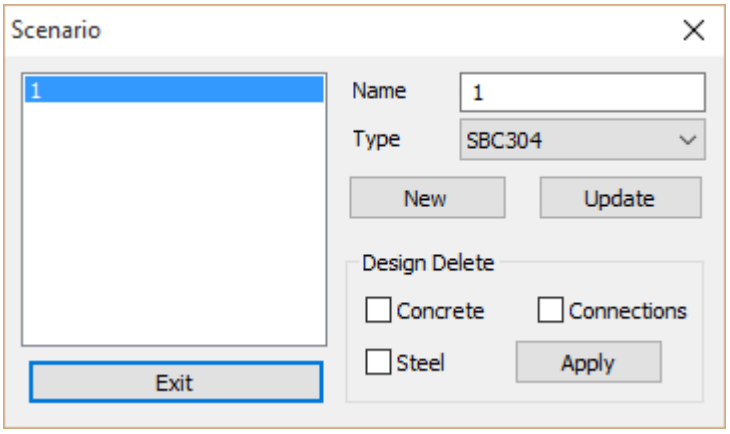
Furthermore, a new group of commands is added, concerning merging of elements (steel and timber)

1.1 New

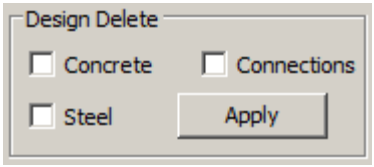


This command is used in order to create a new scenario. Type a name, select the corresponding design regulation and then press the button 

.
The selection of the design code corresponds to the design checks of the structural elements and the calculation of the steel reinforcement. For the SBC 304, you must create the SBC304 scenario for the design and the checks of the concrete structures.



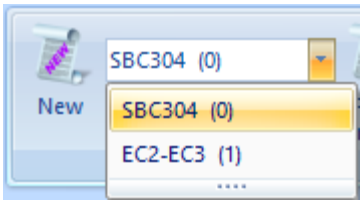
In order to modify an existing scenario press the button "Update"**/.**



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

1.2 List

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

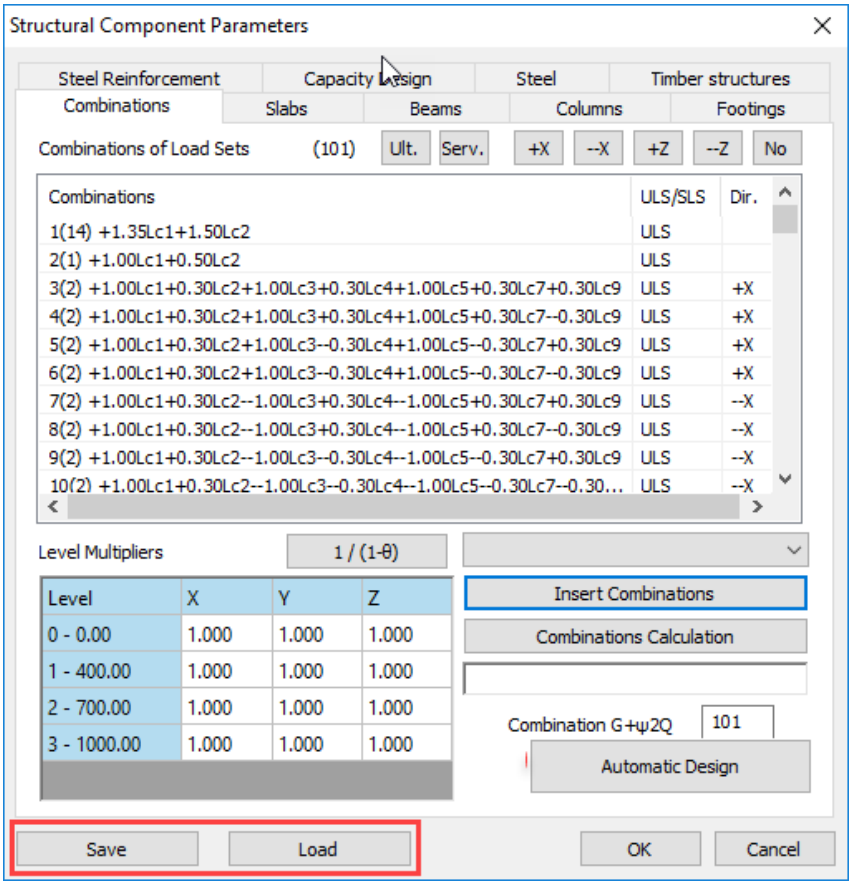


1.3 Parameters

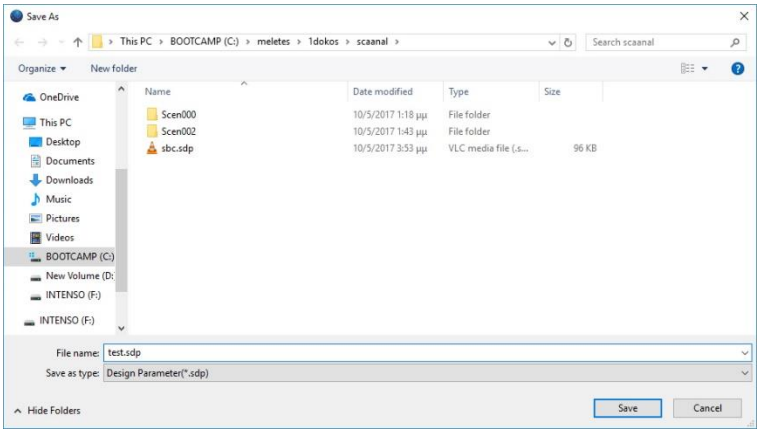
This command is used for the definition of all design check parameters:

NOTE

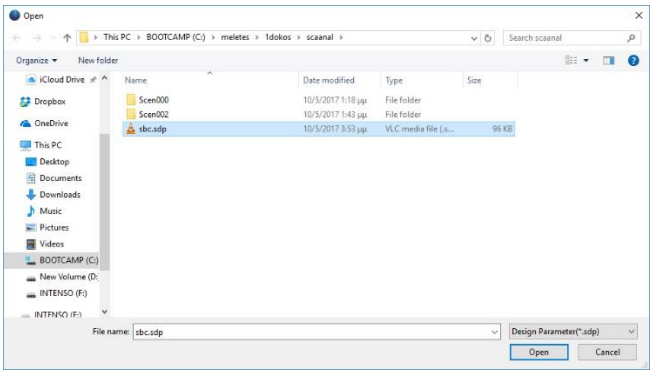
The parameters dialog box in the new SCADA Pro version contains two more commands for saving and reading the design parameters of the active scenario.



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

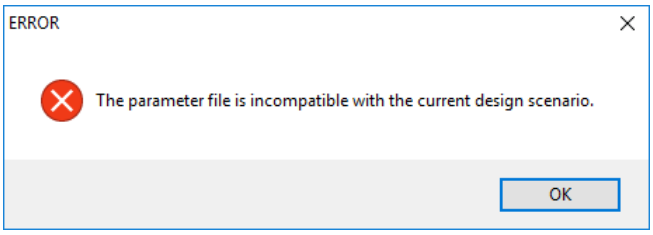


The file extension is sdp scenery design parameters.
Use “Load” command to apply the parameters that are already saved.



ATTENTION

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

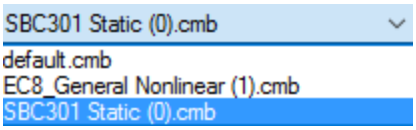


1.3.1 Combinations

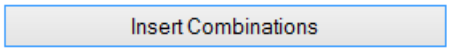
Regardless of the material, the calculation of combinations is a condition for designing.

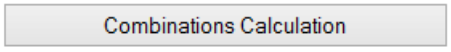
Combinations

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

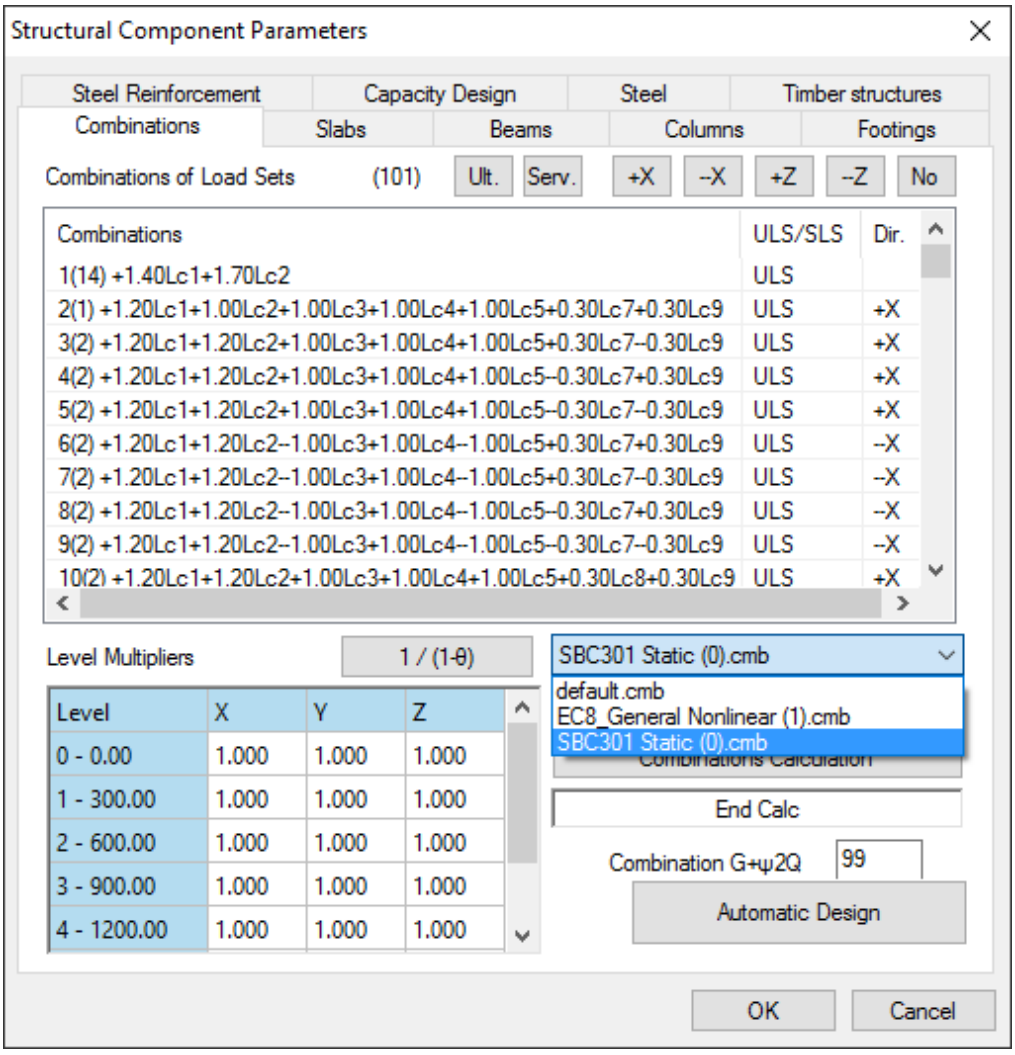
-from the dropdown list 

with automatic calculation or

-through the command 

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

.

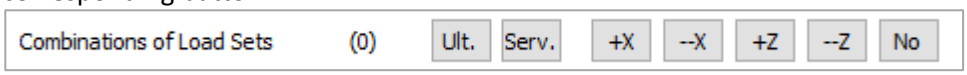


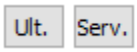
⚠ **NOTES:**

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

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

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



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

.

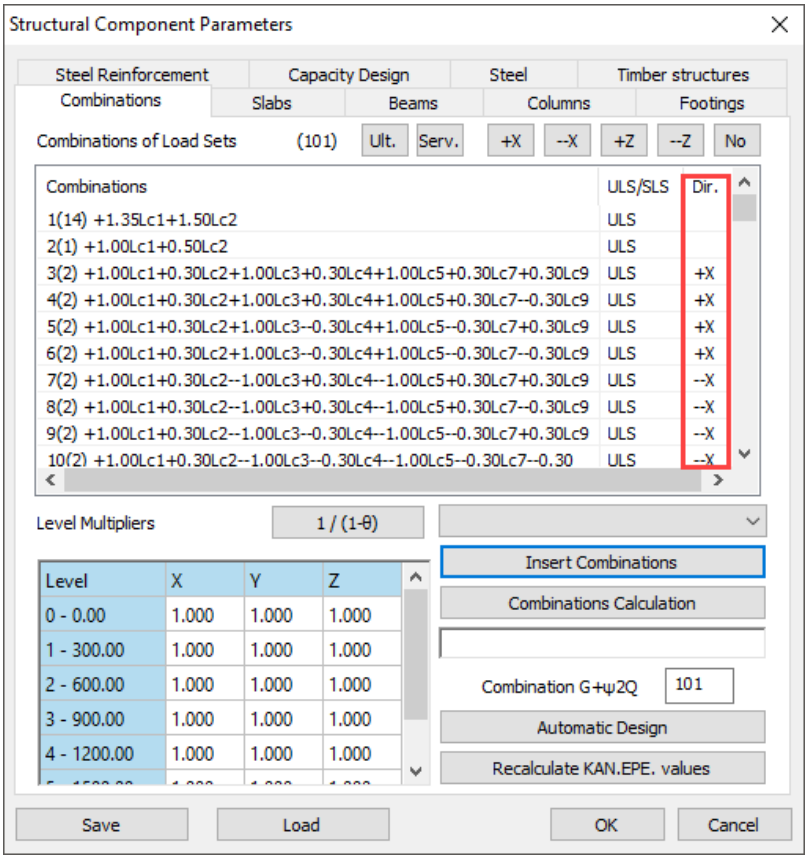
In the column “Dir.” That indicates the direction of the participation for the specific capacity design combination, by selecting the corresponding button 

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

NOTES:

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

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



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

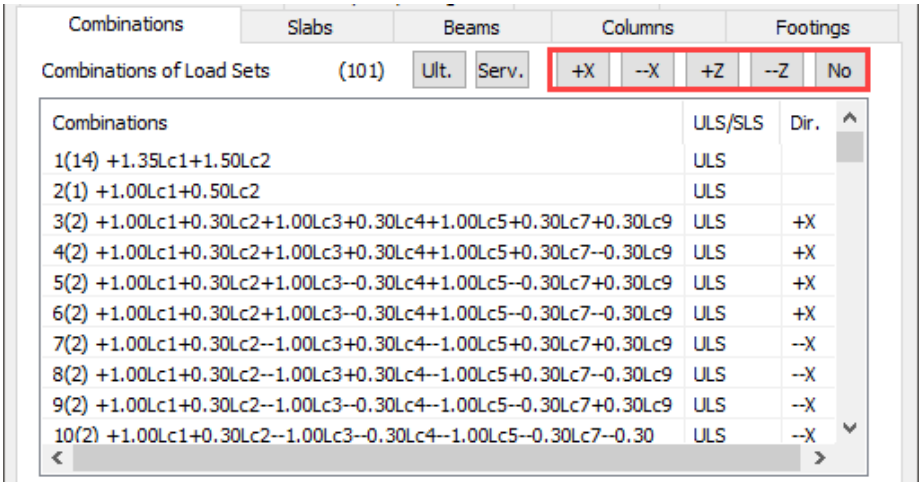
In conclusion, we would assume that in case we want to exclude one direction from one column in order to avoid the capacity design check, we move to the definition and choose the direction of the local axis that is parallel to the direction of the earthquake we want to exclude.
In case that either the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

Indicatevely, in the following printout:

Node = 15
Col. bottom = 14

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

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



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

SBC 304

SECTION 9.2

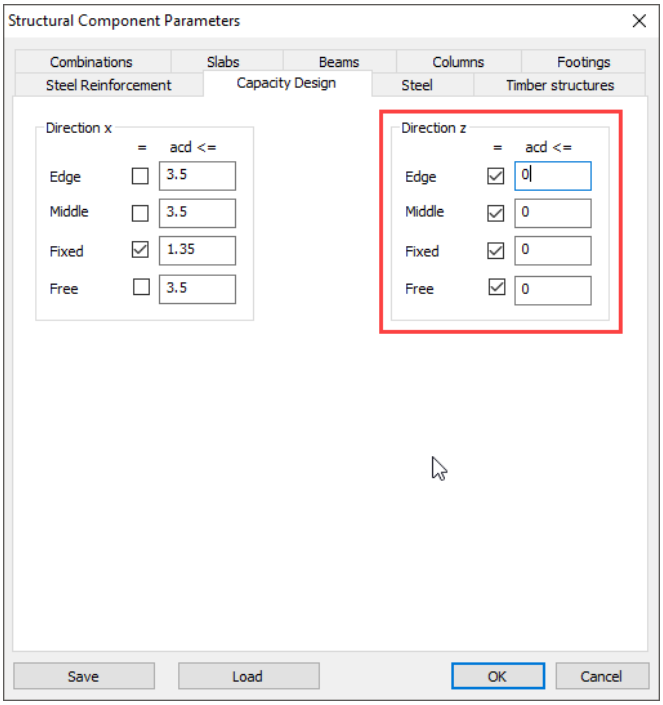
REQUIRED STRENGTH

9.2.1 *Required strength U shall be at least equal to the effects of factored loads in Eq. (9-1) through (9-7). The effect of one or more loads not acting simultaneously shall be investigated.*

U = 1.4 (D + F) (9-1)
U = 1.4 (D + F + T) + 1.7(L + H) + 0.5 (Lr or R) (9-2)
U = 1.2D + 1.6 (Lr or R) + (1.0 L or 0.8 W) (9-3)
U = 1.2D + 1.6W + 1.0L + 0.5(Lr or R) (9-4)
U = 1.2D + 1.0E + 1.0L (9-5)
U = 0.9D + 1.6W + 1.6H (9-6)
U = 0.9D + 1.0E + 1.6H (9-7)

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

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 1 / (1-θ) in order to take into account the P-Delta effect during the design check. The stress resultants will be increased automatically at the corresponding levels, where 0.1 < θ <θmax.

SBC 301

10.9.7.2

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

ATTENTION:

For modification purposes, press the following button Combinations Calculation.

The following field/concerns only the Greek EKOS.

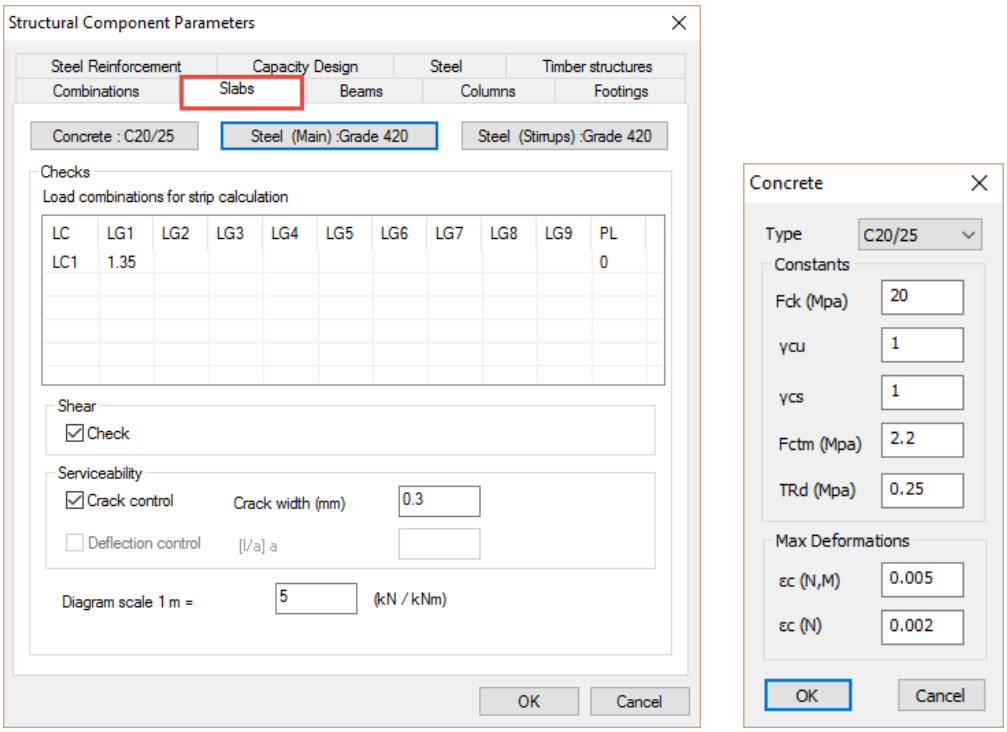
Automatic Design:

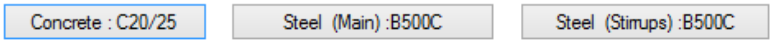
Automatic Design

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 with respect to the fulfillment of the design checks.

1.3.2 Slabs



Press the following buttons 

to change the material of the slabs: concrete and/or 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_{cu}=1 because they are not applied in SBC)

"γ_{cs}": the partial factors for concrete in the serviceability limit state (γ_{cs} =1 because they are not applied in SBC)

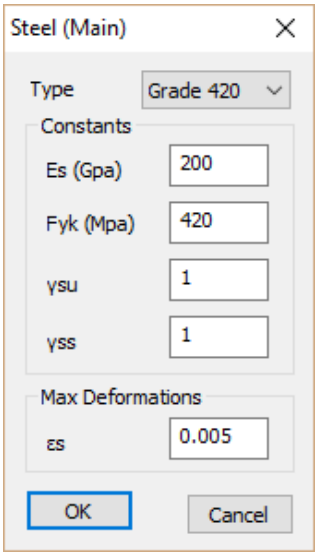
Fctm: the tensile strength of concrete in MPa

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 only 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.



"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 the ultimate limit state (γ_{su}=1 because they are not applied in SBC).
"γ_{ss}": the partial factors for the steel reinforcement in serviceability limit state (γ_{su}=1 because they are not applied in SBC).

In the "Max Deformations" field indicating the maximum deformations of the steel reinforcement.

According to SBC 306, Section 3.8:

The Standards of the American Society for Testing and Materials are declared to be part of SBC 304.

So the table below defines the strength of the rebars, according to those standards

SBC 306

Table 2-1 Inch-Pound and Soft Metric Bar Sizes

Inch-Pound		Metric	
Size No.	Dia. (in.)	Size No.	Dia. (mm)
3 4 5	0.375 0.500 0.625	10 13 16	9.5 12.7 15.9
6 7 8	0.750 0.875 1.000	19 22 25	19.1 22.2 25.4
9 10 11	1.128 1.270 1.410	29 32 36	28.7 32.3 35.8
14 18	1.693 2.257	43 57	43.0 57.3

Table 2-2 ASTM Specifications - Grade and Min Yield Strength

ASTM Specification	Grade/Minimum Yield Strength	
	Inch-Pound (psi)	Metric (MPa)
A615 and A615M	40/40,000 60/60,000 75/75,000	280/280 420/420 520/520
A955 and A955M	40/40,000 60/60,000 75/75,000	300/300 420/420 520/520
A996 and A996M	40/40,000 50/50,000 60/60,000	280/280 350/350 420/420
A706/A706M	60/60,000	420/420

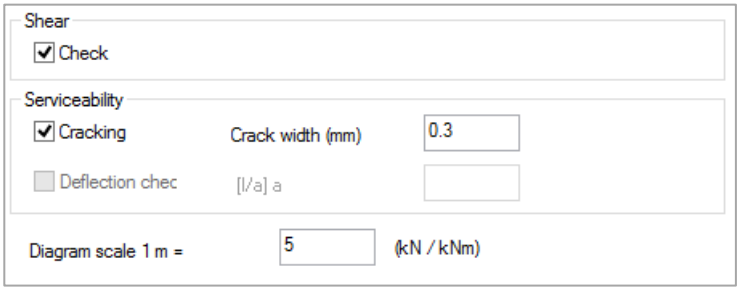
In the "Checks" field the following dialog box can be edited as appropriate.

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 take them into consideration.

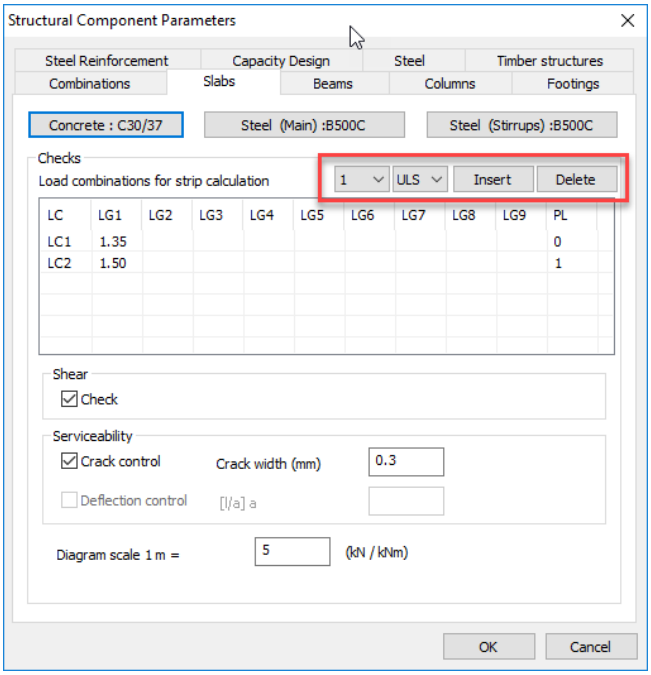


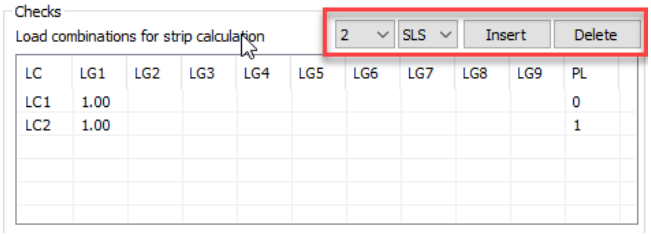
Activate the relative checks for the slab verification.
Define the value in the field "Crack width" for the concrete cracking check.
Define scale for diagrams' display.

NEW FEATURES

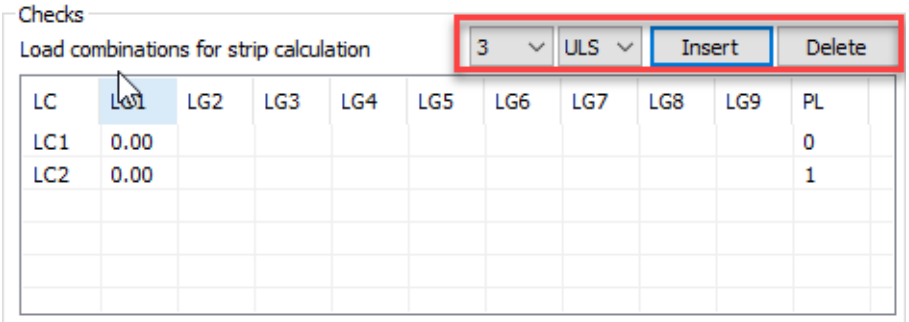
§ More combinations for the slab's design

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:





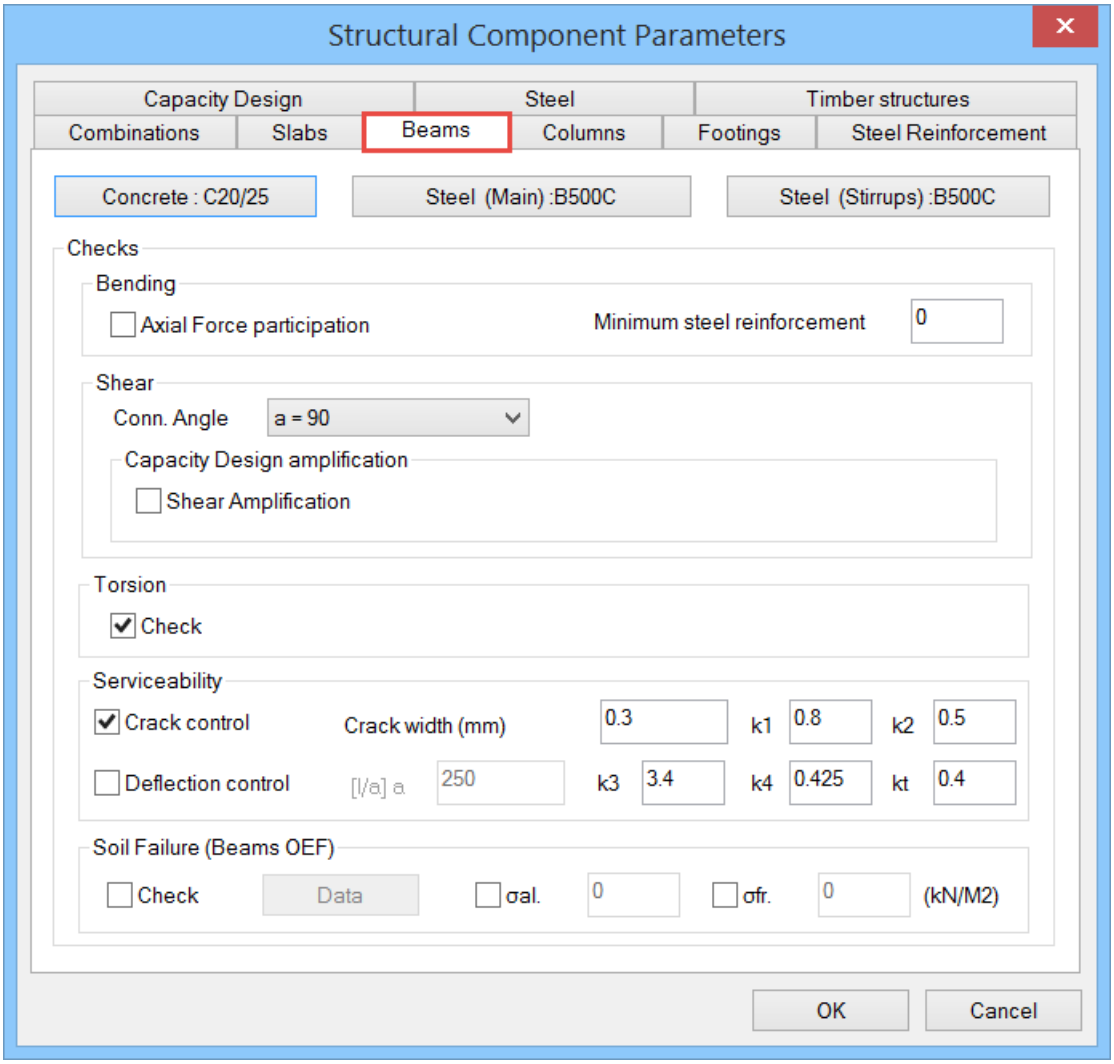
There are 2 default combinations, one for the Ultimate and one for the Serviceability Limit States. In order to create a new combination, press, "Insert". The new combination is the combination number 3 and the coefficients are all 0.



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.

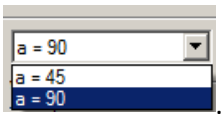
1.3.3 Beams

This field contains the design checks for the beams verifications.

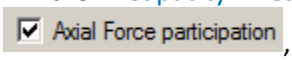


In the "Bending" field activate the checkbox "Axial Force participation" for considering the axial force in the bending design checks.

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

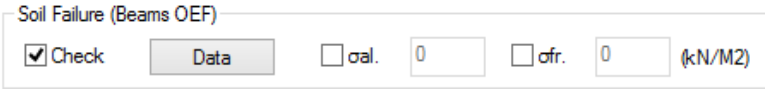
BEAM	1 5 b=0.25 h=0.50 c=0.03					
COMB	M	N	Du1	Du2		
100	-21.356	-0.000	0.00443	0.00011	0.00454	0.01500
101	-20.315	-0.000	0.00432	0.00012	0.00443	0.01500
102	-23.958	-0.000	0.00471	0.00010	0.00481	0.01500

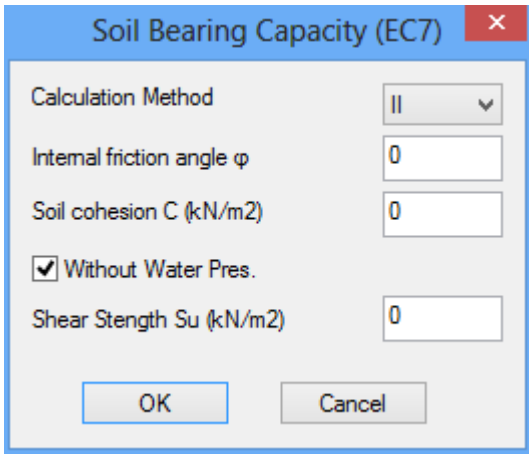
EXAMPLE:

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.





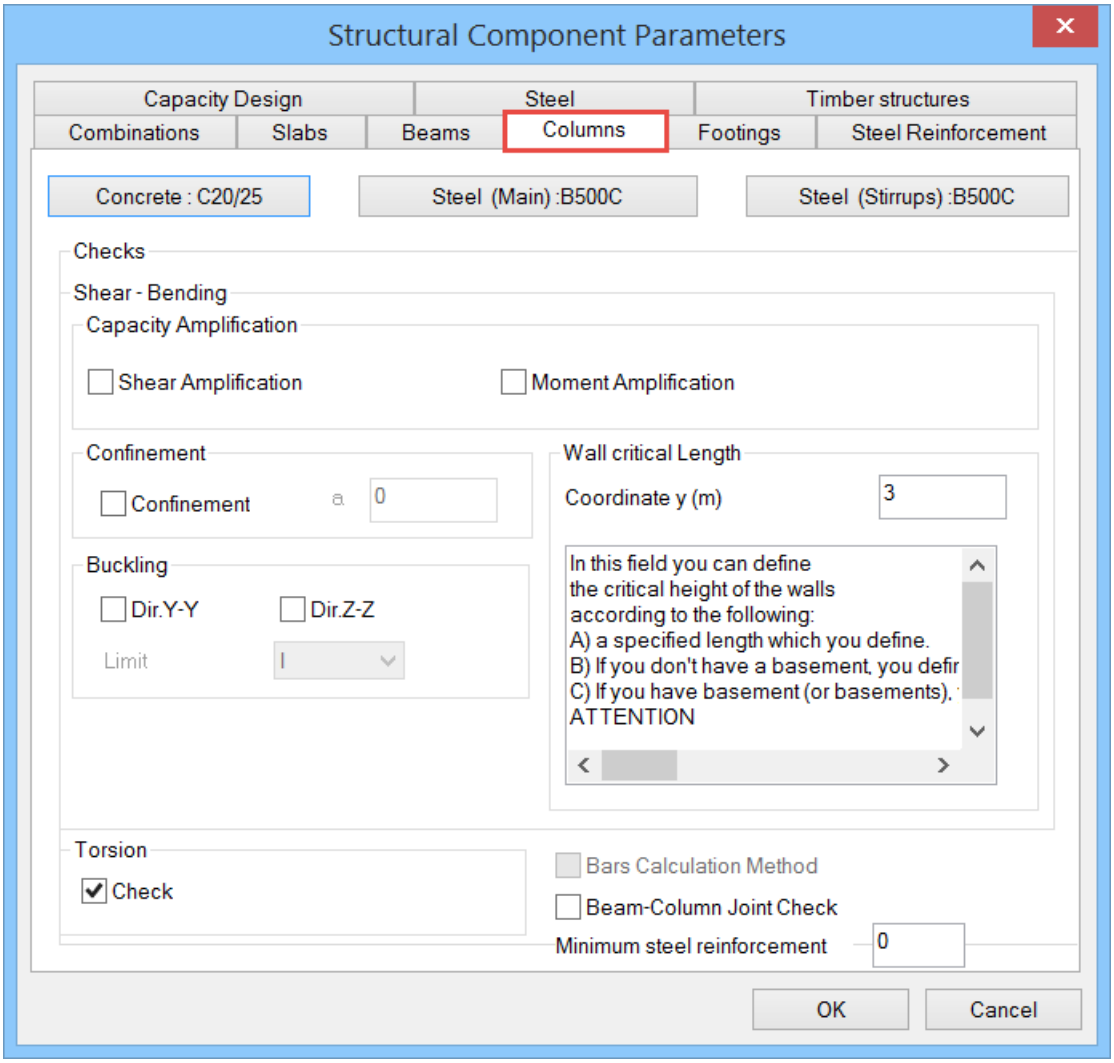
Activate the checkbox "Check" and click the button "Data". A dialog box opens entitled "Soil Bearing Capacity (EC7)" where you enter the soil data, in case you have a geotechnical study. Select a calculation method from the drop-down list and define an internal friction angle ϕ, the soil cohesion and 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 by means of the subsequent rules.

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

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

Where
$\eta=0.6\cdot(1-f_{ck}/250)$;
v_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;

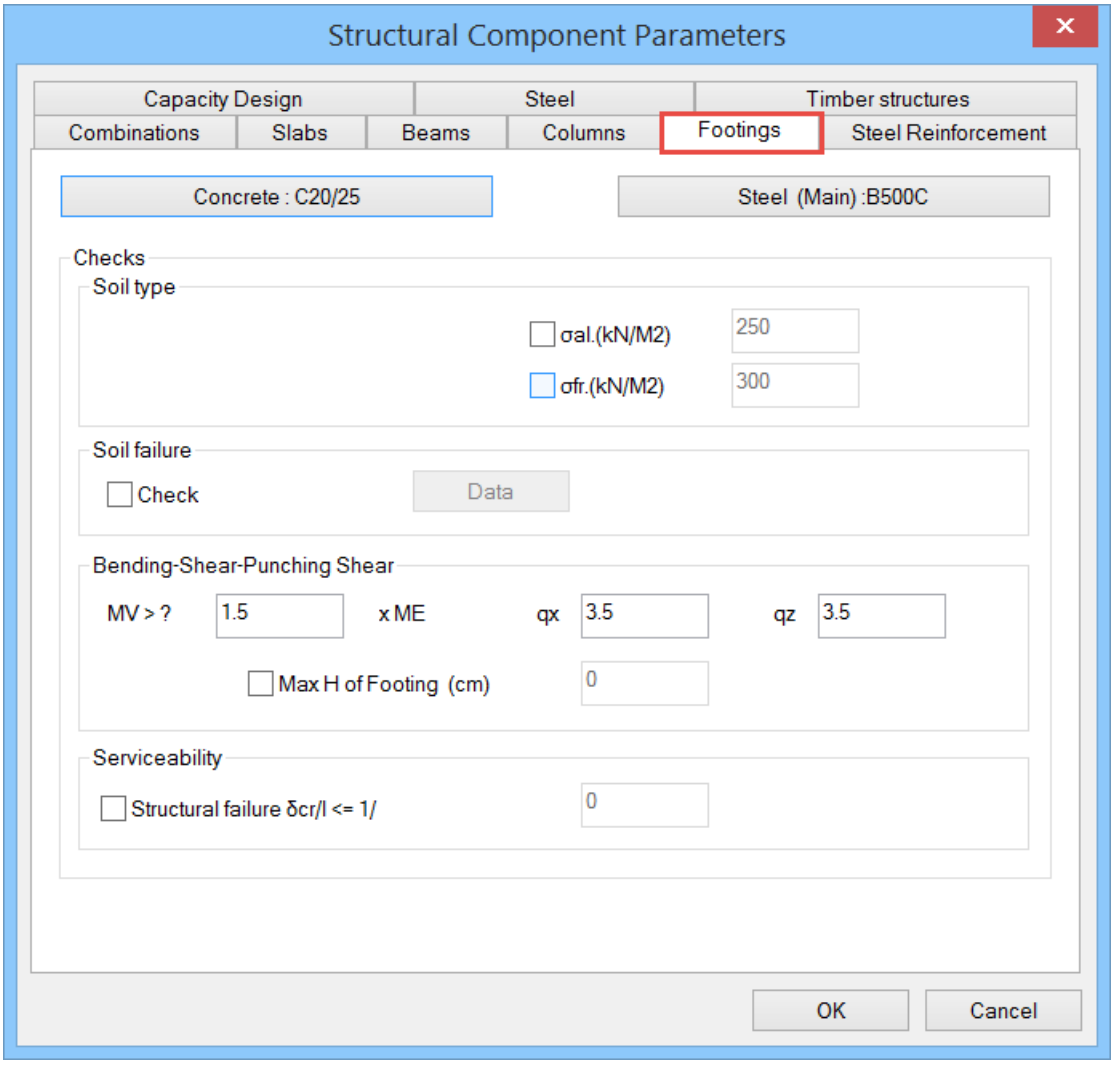
And the effective joint width b_j is:

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)*

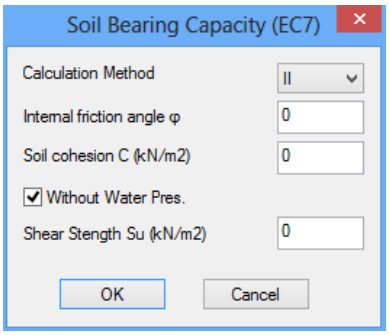
1.3.5 Footings

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 

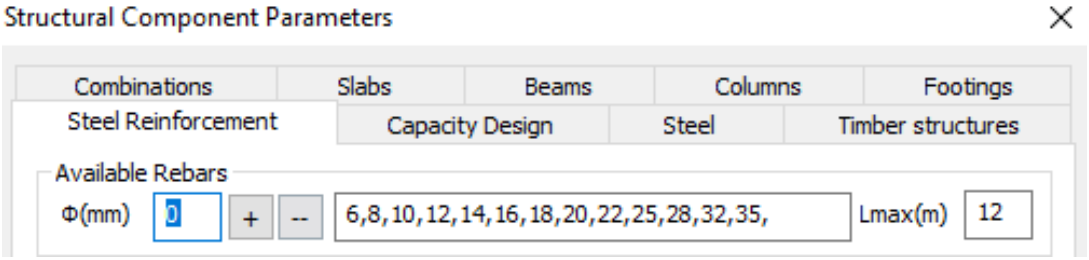
add or remove a new or an existing diameter, respectively. Type a value in the following field 

and press the button 

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

.

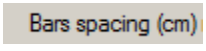
In the Lmax field, you type the maximum bar length to be used in the reinforcement of the structural elements.



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).

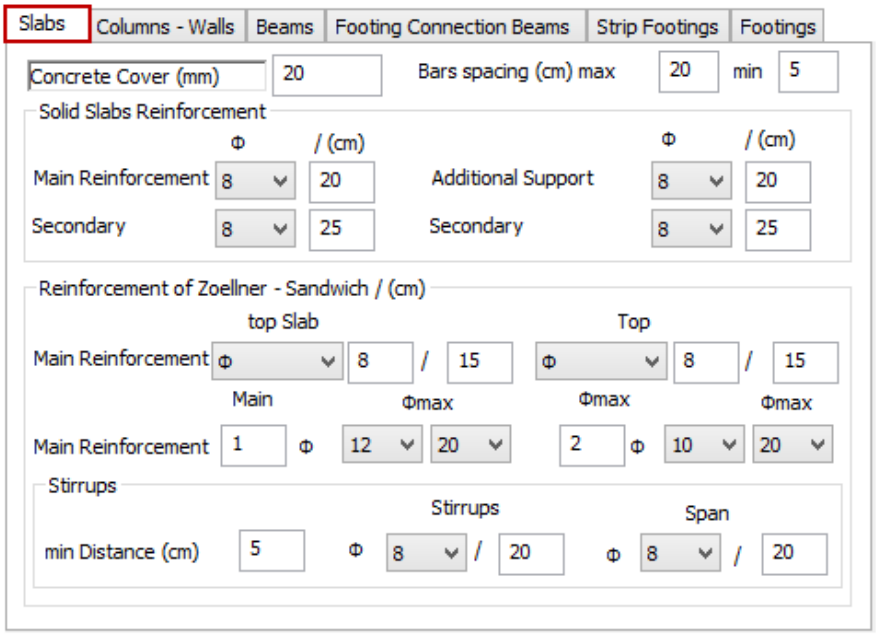
In the "Bar spacing" field 

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.

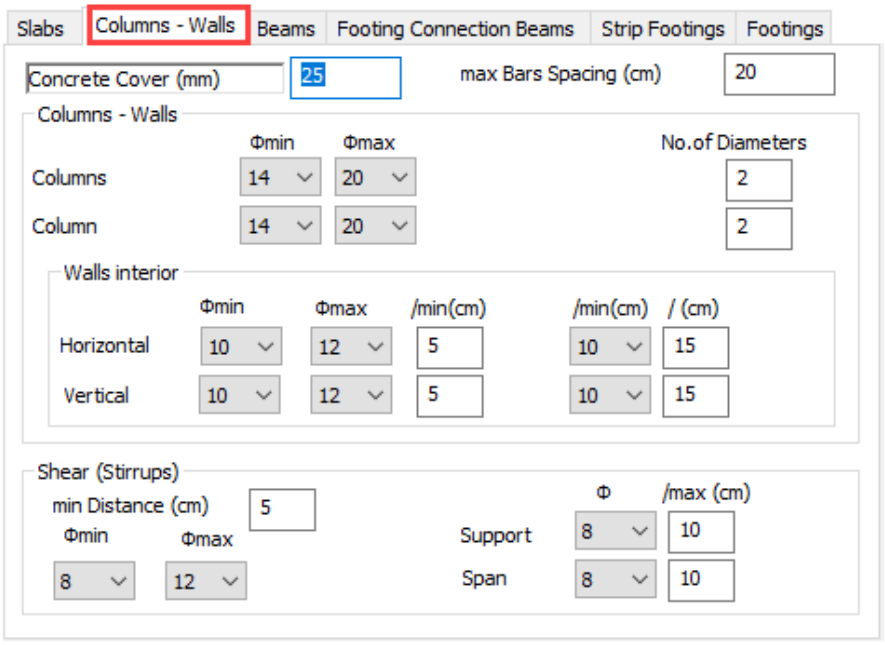


1.3.6.2 Columns-Walls

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 for 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).

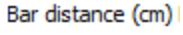
In the following fields 

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.

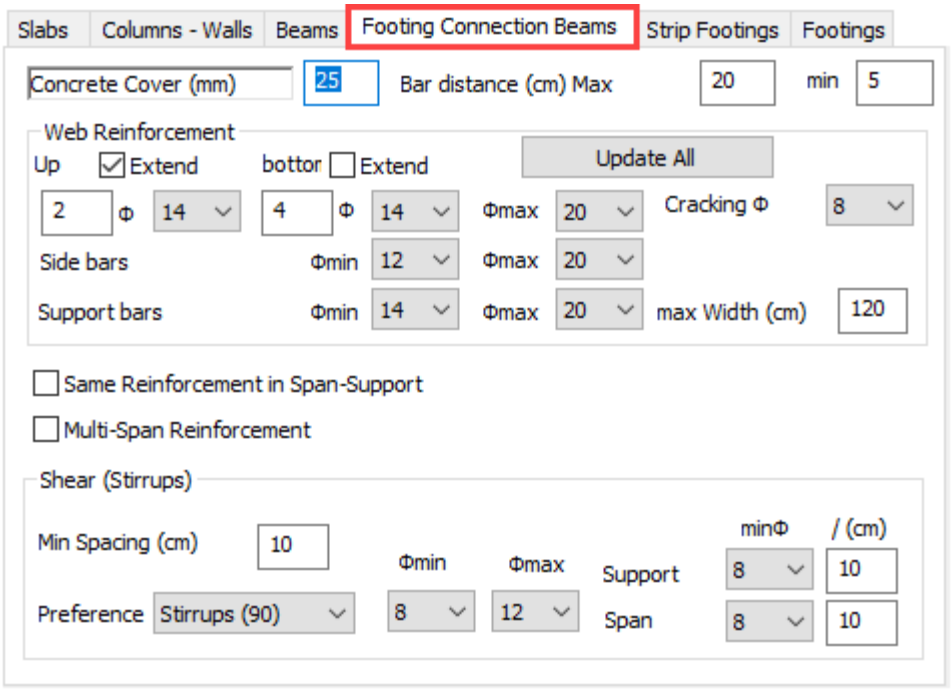
1.3.6.3 Beams/Footing Beams

In "**Beams**" and "**Footing Beams**" define reinforcement:

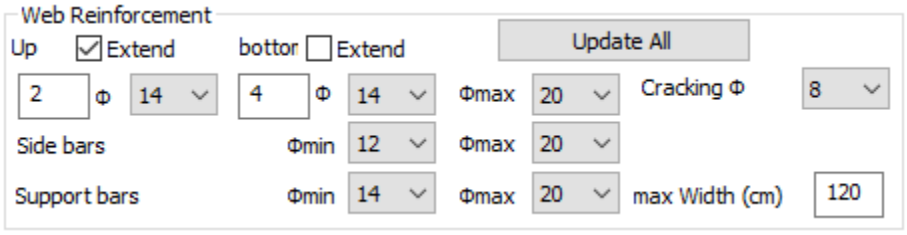
Type bar cover value according to the climatic conditions (mm).

In 

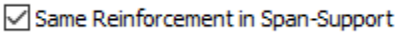
type the maximum and minimum limits for distance (cm) between the bars.



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.

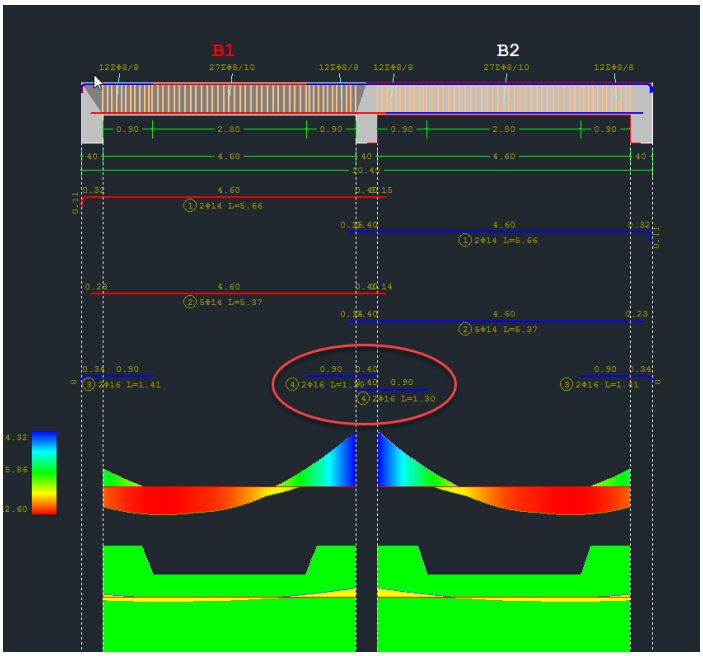


Figure 1

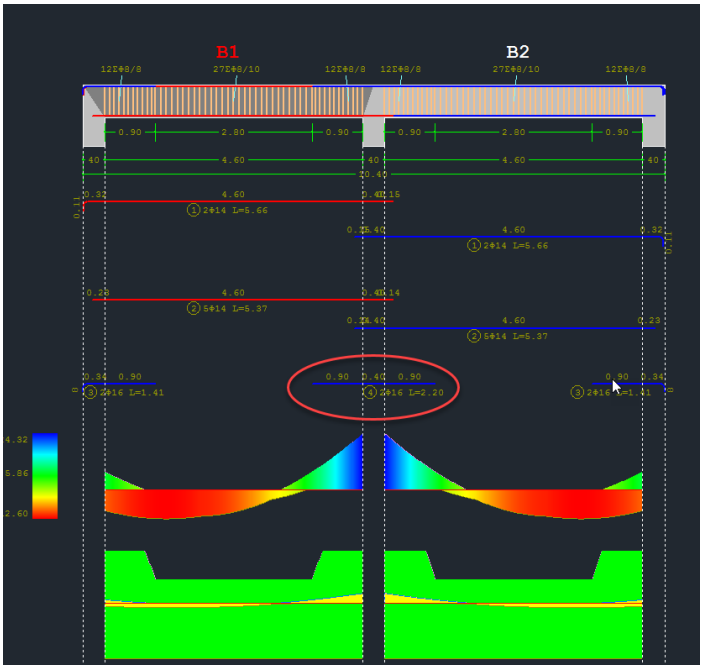
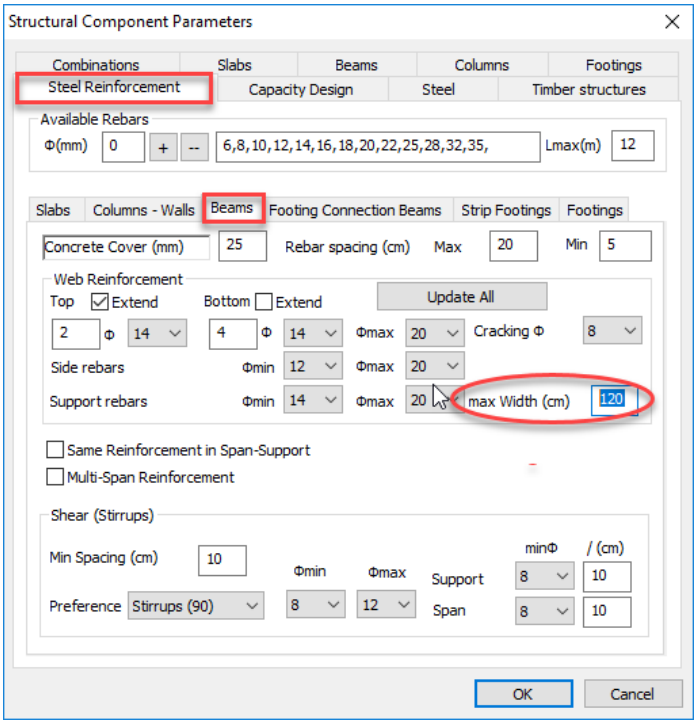


Figure 2

A. The first criterion is the <u>width of the support</u>, 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.

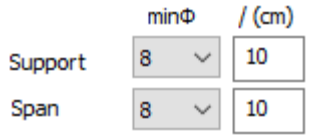
CONCLUSION:

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:

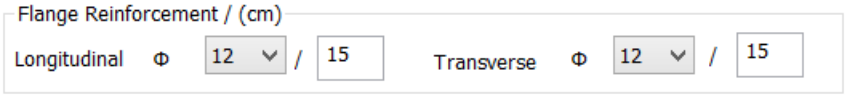
Update All: After completing the parameter selections and before closing the window, press the Update All button to update the parameters.

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 previously described procedure.

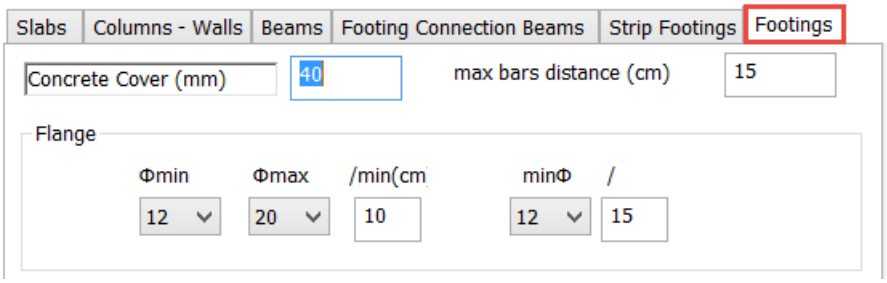
In addition, in the "Range Reinforcement" field define the limits for longitudinal and transversal flange reinforcement.



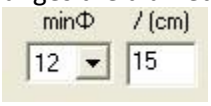
1.3.6.5 Footings

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

Type the bar cover value in accordance with 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.

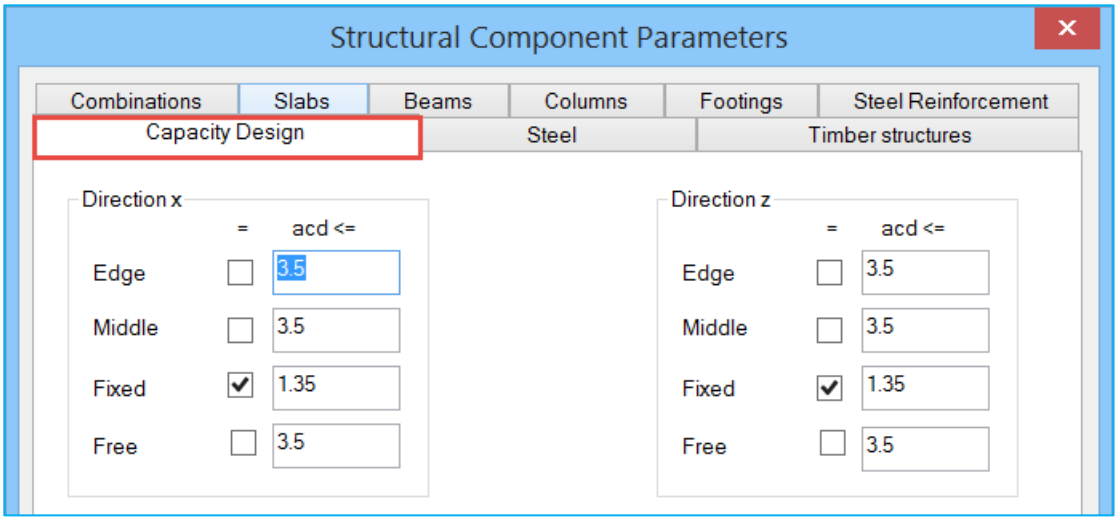


Specify the minimum and maximum diameter of the bars and the minimum distance. For lower distance, the program changes the diameter.

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.

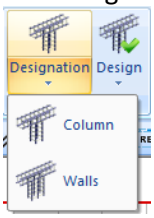
1.3.7 Capacity Design

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.

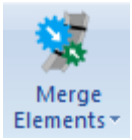
Generally, the value of α_{cd} should be less or equal to the seismic behavior factor q.
For fixed columns, α_{cd} = 1.35 is taken.
Activate the corresponding checkbox and type the value you want.
Without ticks, the program takes into consideration the calculated values.
The designation of a node as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



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

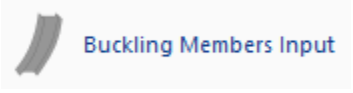
⚠ Finally, considering the exploration of the column, there are the results of the capacity design check (initial and incremented moments) for each combination. Of course, in the direction in which the capacity design check has not been taken into account, the initial moments are the same as the incremented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

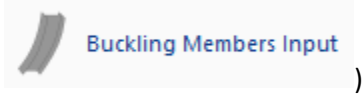
1.4 Merge Elements



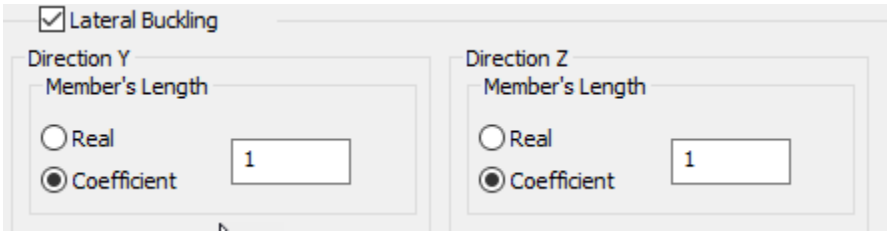
In the new version of the program, a new command group is added, which concerns merging of steel (and timber) members for the calculation as well as buckling and deformation checks display according to EC3.

IMPORTANT NOTES:

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

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

)



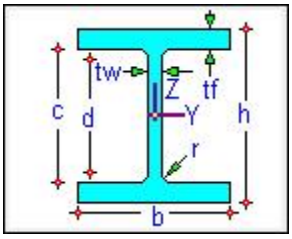
- Now, be using merging per direction, there is no need for the coefficient process, and merging will be achieved, in most cases, automatically.
- Also, note that through the merging process, the buckling length, is calculated correctly, and in the printouts of the results a merged element is printed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

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

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

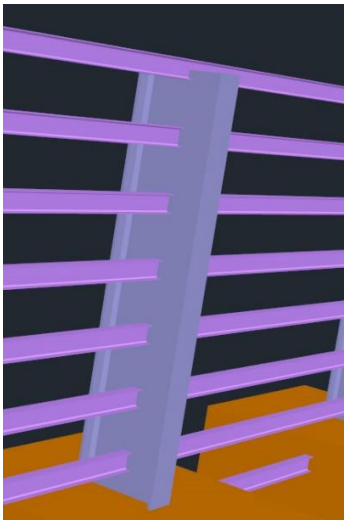
- **y-y** is the **major**, and
- **z-z** is the **minor**,

as in the figure below:

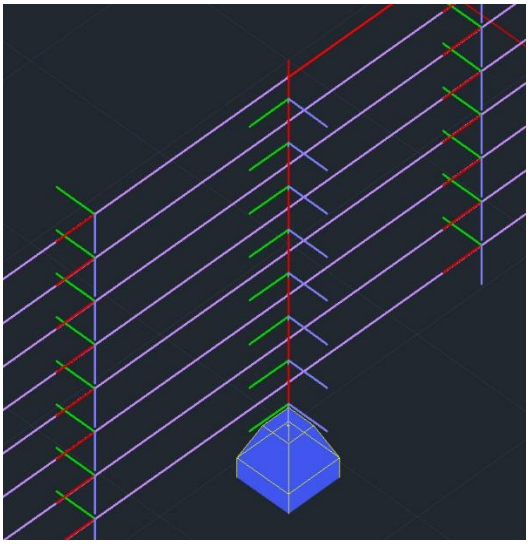


EXAMPLE:

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



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



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

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

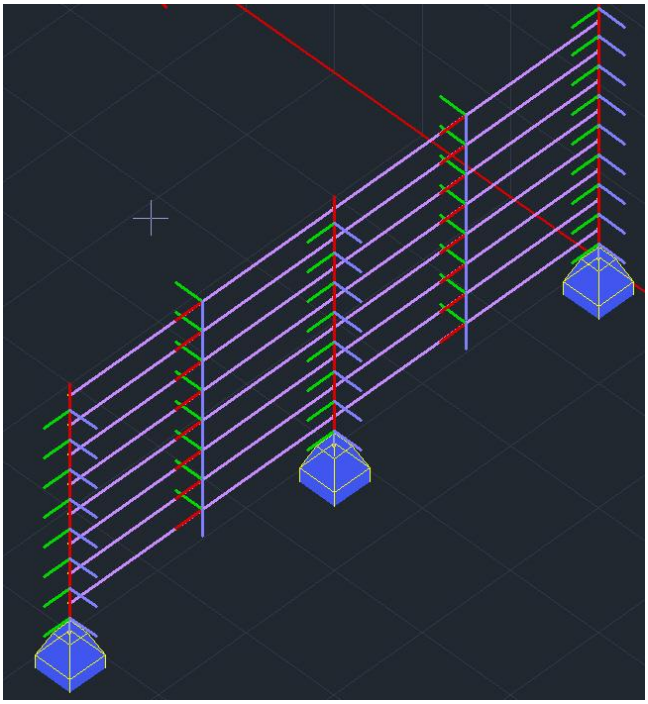
In the other direction, buckling along its **minor** axis **z-z** (blu) means:

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

NOTE:

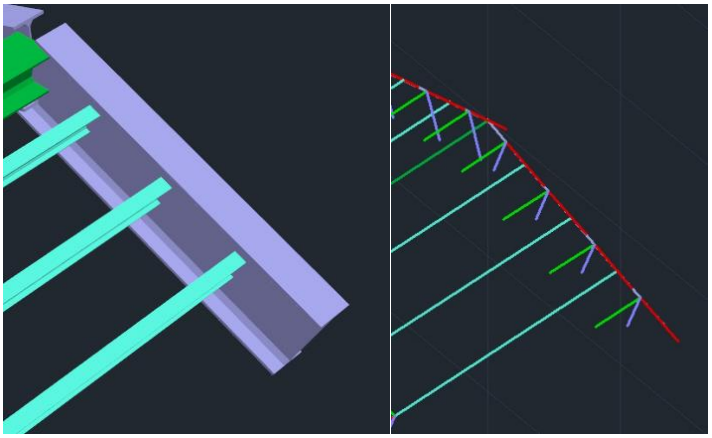
Generally, making a rule, we could say that, we consider the **merged length Ly** in the direction where the local axis y-y is parallel to the supporting elements. While in the other direction, if there are no supporting elements, **Lz** is **the length of each member**.

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



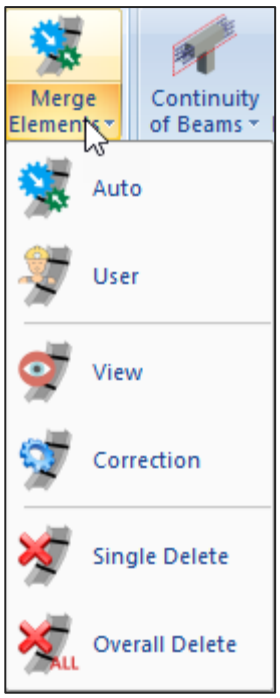
The supports from the columns are parallel to the local z-z axis (blue, out of plane) of the griders. So, merging will be in Lz (total length). While in the y-y direction (green, in the plate), Ly is the length of each member.

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



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

Merge group command, contains the list of commands below:



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

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

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

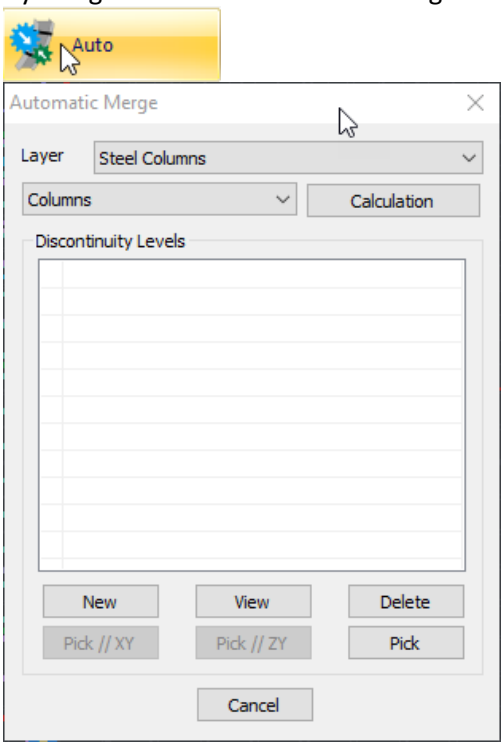
Finally, in automatic merging, there is the definition of discontinuity levels, horizontal or vertical, used as merging boundaries of a continuous element.

NOTE

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

1.4.1 Auto merge

By using this command the following dialog box is displayed:



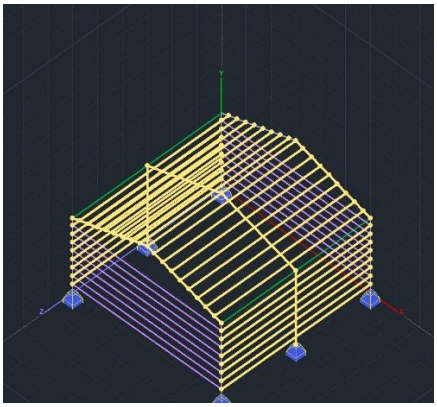
First, choose the layer of the elements to merge.

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

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

Merged elements are displayed with colors:

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



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

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

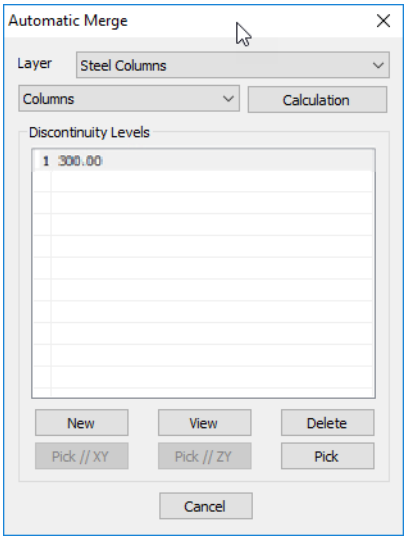
Predefined limits:

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

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

EXAMPLE:

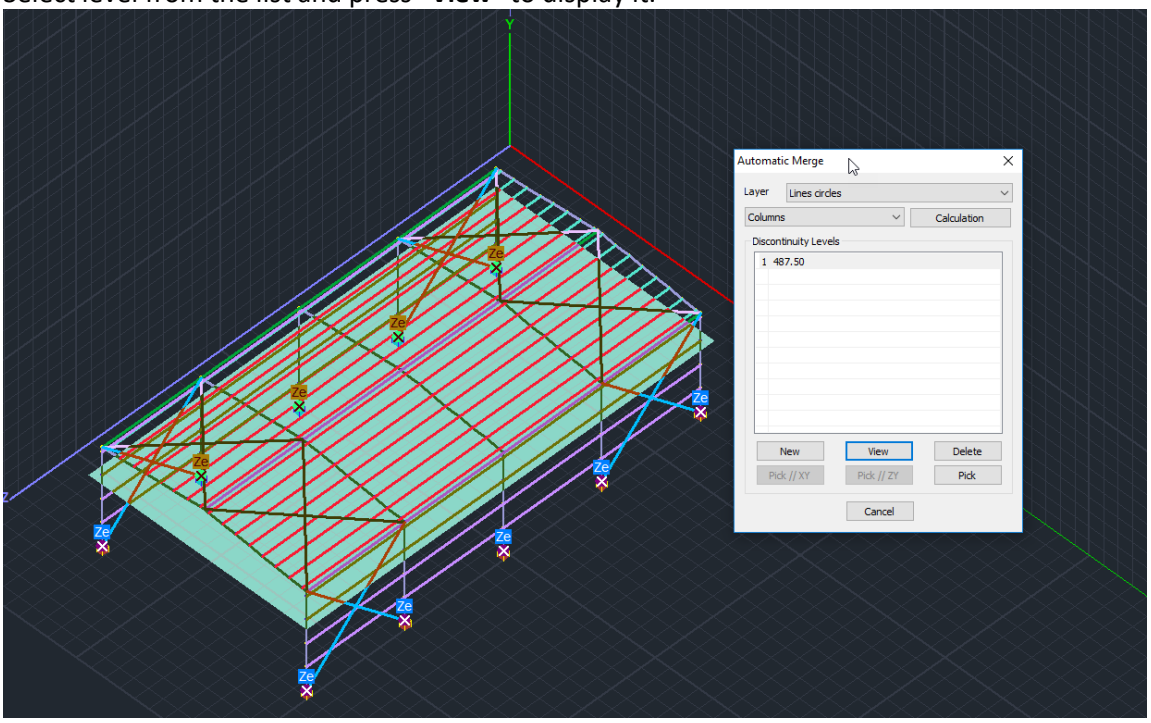
A three floor building with 0.00, 300.00 and 600.00 height levels, in **discontinuity levels** list of the columns, only the level 300.00 will be specified by default (that is, only the intermediate level without the limits)



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

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

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

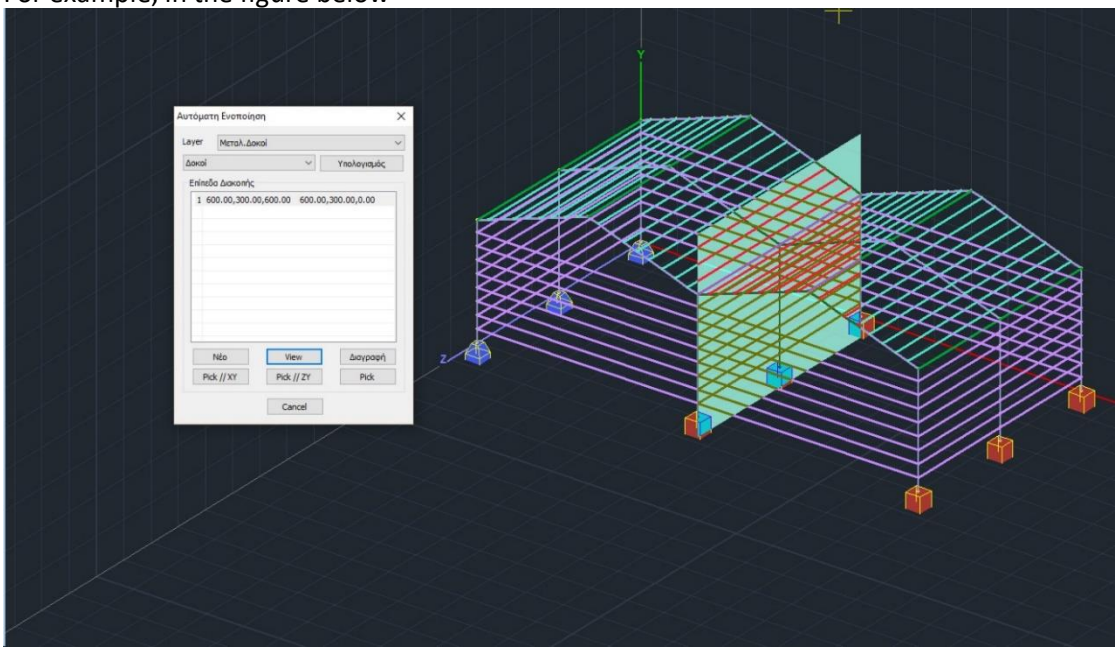


To delete a **discontinuity level,** select it from the list and press "**Delete**".

- For the **BEAMS**:

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

For example, in the figure below



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

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

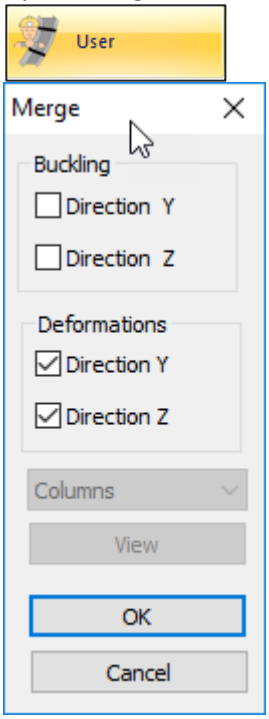
For Beam's and Column's **discontinuity levels,** *editing* can be achieved in two ways:

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

1.4.2 Users merging

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

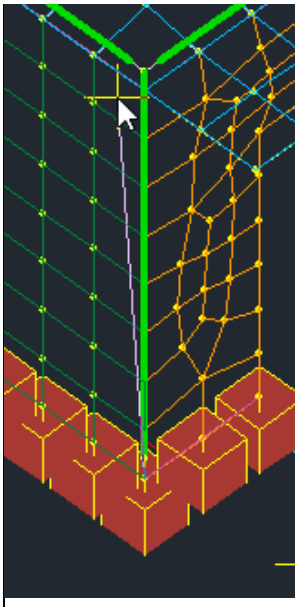
By selecting the second point (endpoint), the following dialog box appears:



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

1.4.2.1 Merge concrete columns

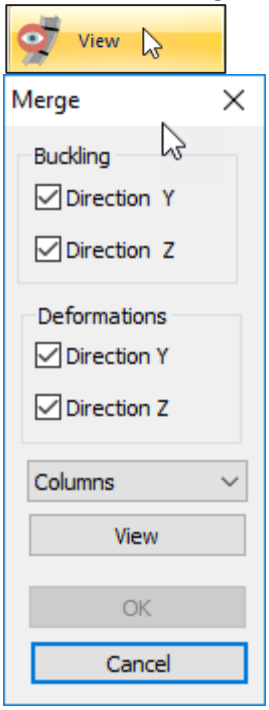
This command is mainly used in masonry buildings with vertical reinforced concrete elements which connect the nodes of the surface elements and which, in order to be designed, must be merged.



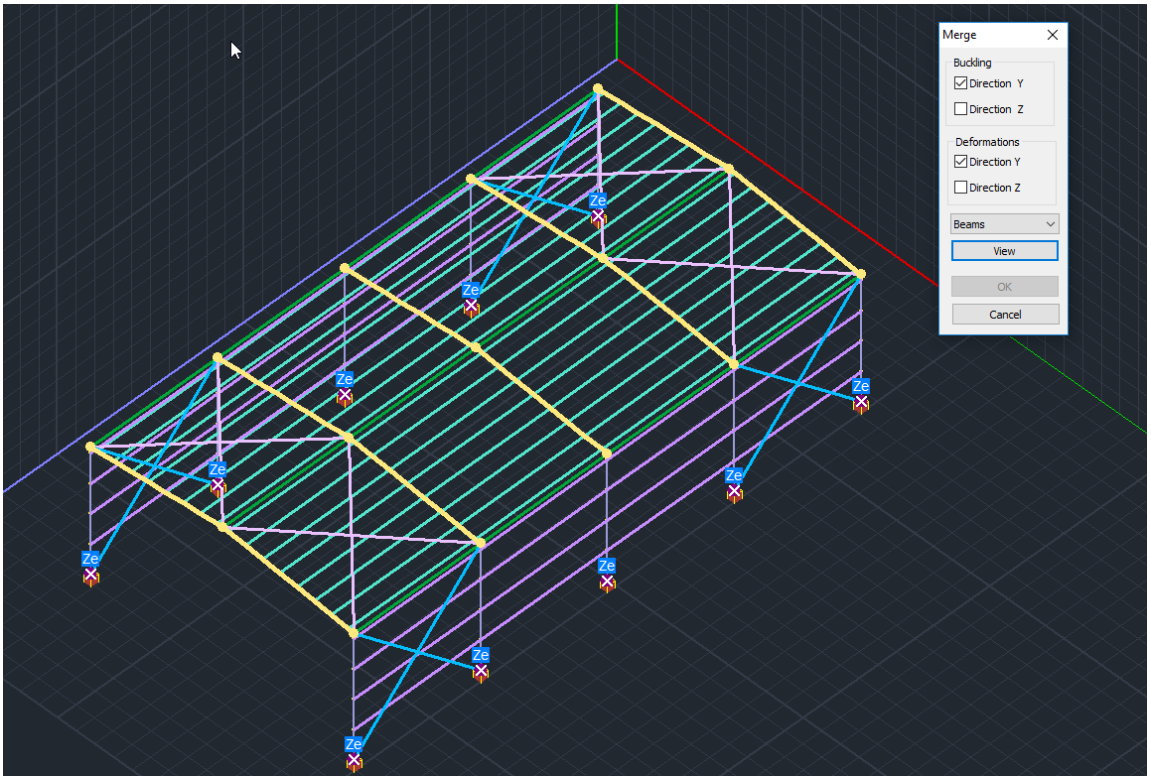
You select the command, and then you point the starting point and the end point of the members you want to merge.

1.4.3 View

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



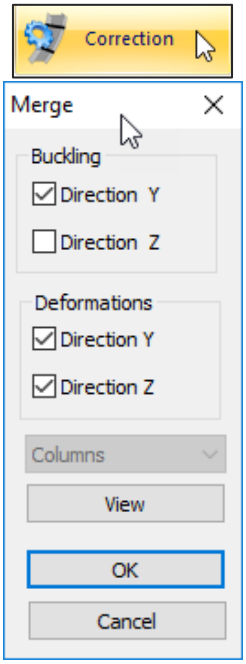
- Select element’s type from the list, then
- check in Buckling / Deformations the direction of merging in order to see the corresponding merged elements.



1.4.4 Correction

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

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



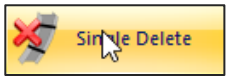
in which checks shows the merging direction.

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

ATTENTION

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

1.4.5 Single Delete

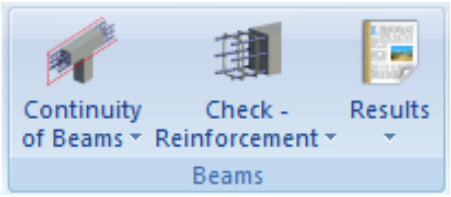


Select this command in order to delete a single merging.
Select the command and click a merged element. Right click to delete merging.

1.4.6 Overall Delete

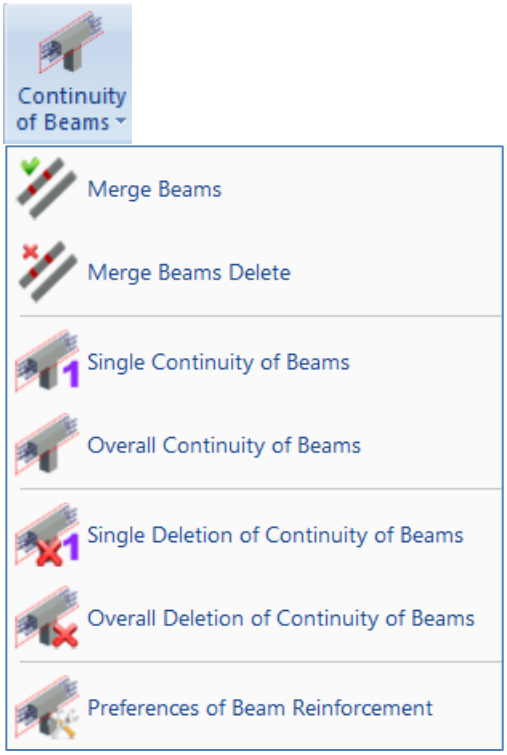
Select this command to delete all merging. Select to delete all merging from all the merged elements.

2. Beams



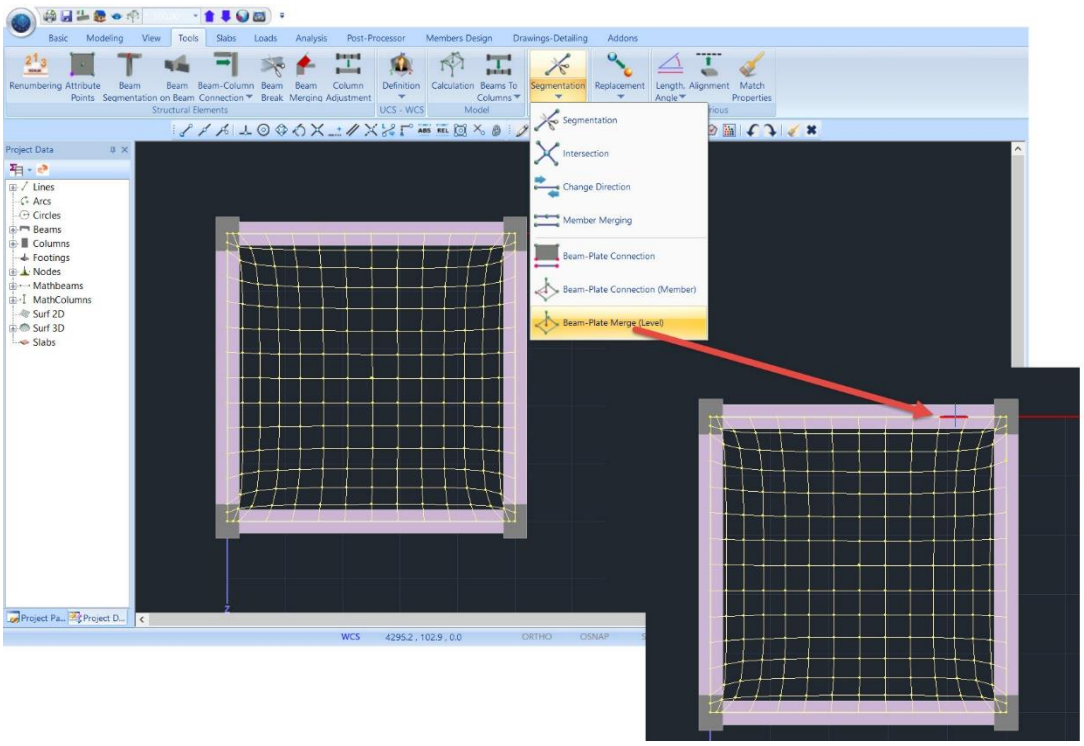
The "Beams" command group contains the following commands
- **"Continuity of Beams"**
- **"Check – Reinforcement"**
- **"Results"** related to the design checks and the checks of the steel reinforcement

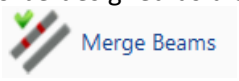
2.1 Continuity of Beams



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, in order 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”.

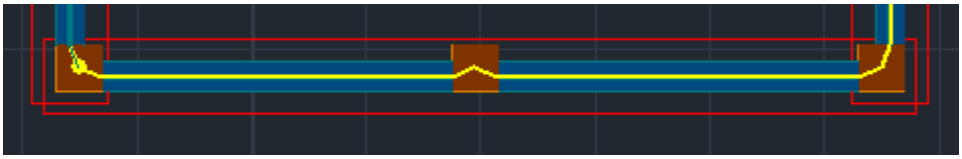
If for some reason you wish to delete a preliminary merging, select the command 

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

2.1.2 Single Continuity/ Overall Continuity of Beams

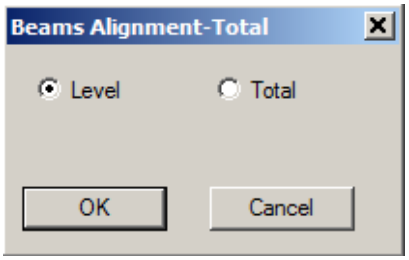
Single Continuity of Beams: This command is used to (selectively) define the beams that participate in a beam continuity.

- ✓ 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.

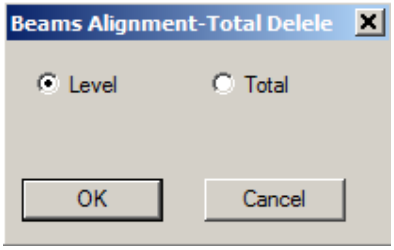
2.1.3 Single/Overall Deletion of Continuity

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.

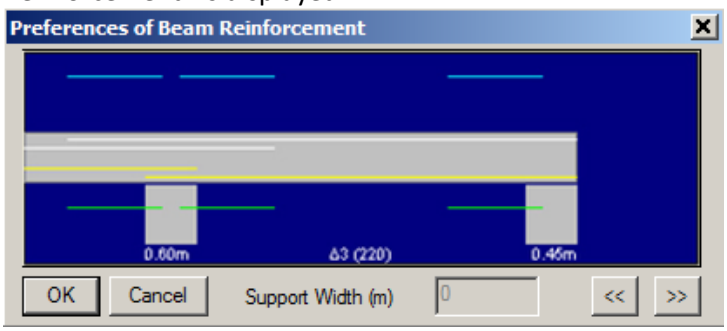
2.1.4 Preferences of Beams Reinforcement

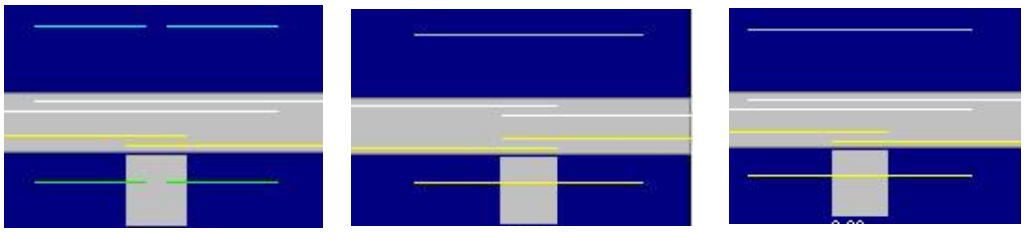
This command is used in order to:

- 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.

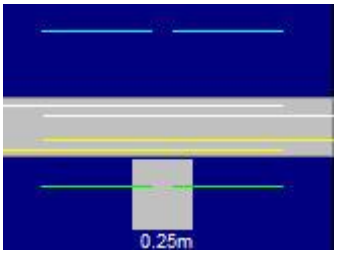
EXAMPLES:

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

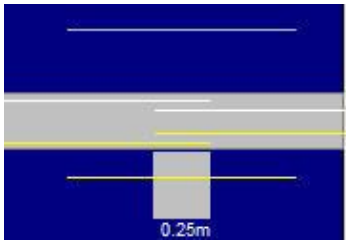




On the common support, the positioned bars consist of two different bars.
By clicking on the two bars, they are converted into one common bar.



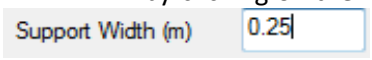
In the figure on the left, the positioned bars to the corresponding spans, pass through the spans on both sides, on the top and bottom of the beam.
This means that during the calculation, the program takes into account the bars from both spans (top and bottom).



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

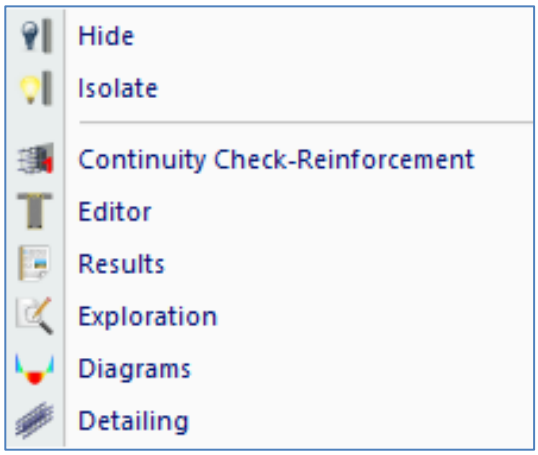
In order to change the bars on the left side (top and bottom) use the arrows  to view the left span.

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.

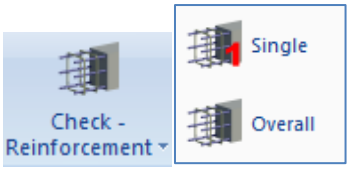
§ Mouse right button function

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



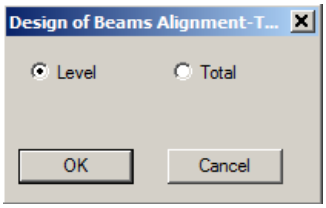
2.2.1 Single

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:

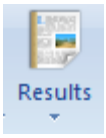
Failure in Bending	M
Failure in Shear	V
Failure in Torsion	T
Dense positioned Stirrups	Asw
It has exceeded the maximum steel reinforcement ratio	ρ
It has exceeded the maximum anchorage length	ldb
Capacity Design	αcd
It has exceeded the maximum crack width	Wk
Deflection failure	L/d

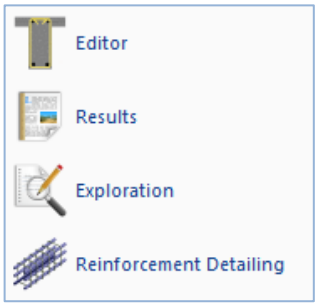
2.2.2 Overall

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.

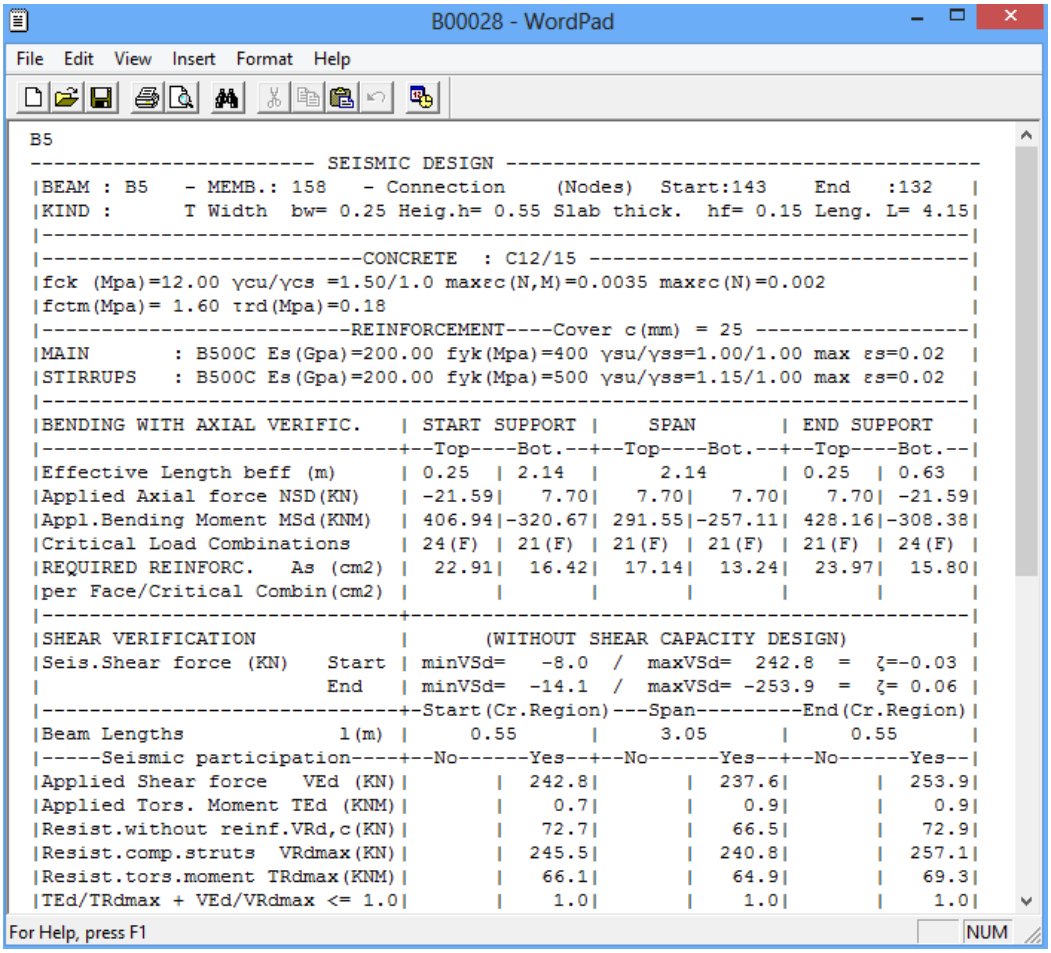
2.2.3.1 Editor

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.

⚠ In addition, 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

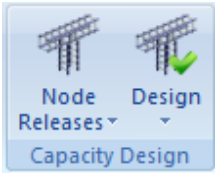
 This command is used to display the TXT file that contains all design checks and calculation results of a beam. Select the command and left click on the centroid axis of the beam to open the TXT file and read the results.

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.

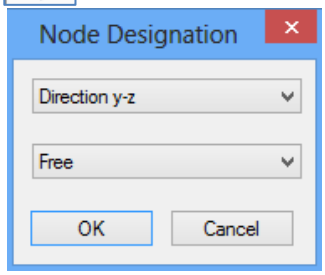
⚠ Capacity Design always precedes columns and walls design if necessary.

3.1 Node Releases

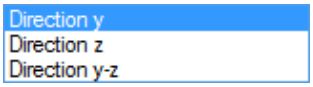
This command is used to determine the node releases of the columns and walls.

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

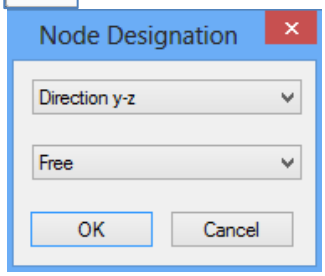
the node releases for the capacity design application.

ATTENTION:

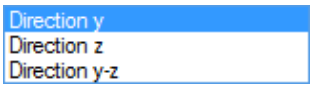
⚠ Note that this command is used only for **columns'** nodes.

3.1.2 Walls

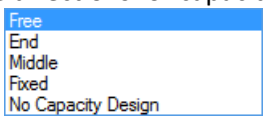
Specify the node support releases of the wall for the capacity design application.
Select the command and the column's nodes for changing the support conditions. Right click to end selection and the dialog box opens:



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.

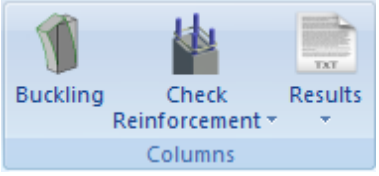
3.2.3 Results

 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.

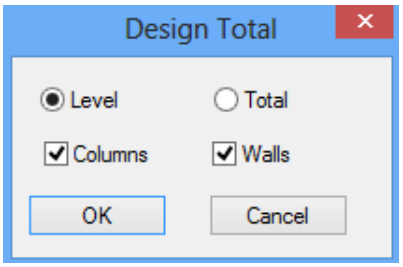
The initially indicated type of failure appears above the element as well:

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:



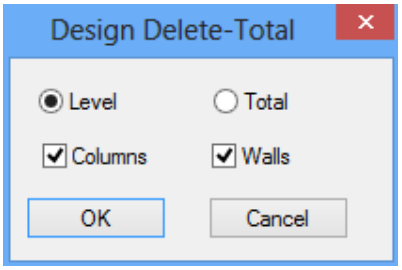
Select whether to design all columns/walls of the current level or the building in total.

4.2.3 Single Delete

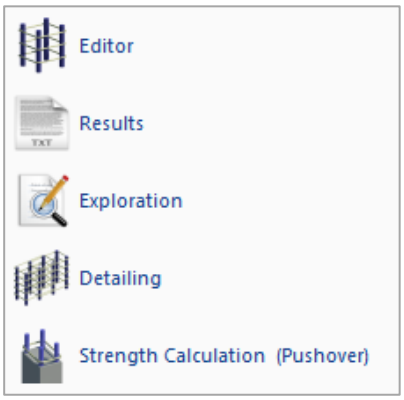

This command is used to delete the previous calculations from one or more columns or walls.
Press the command and left click to select the columns and the walls.

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.

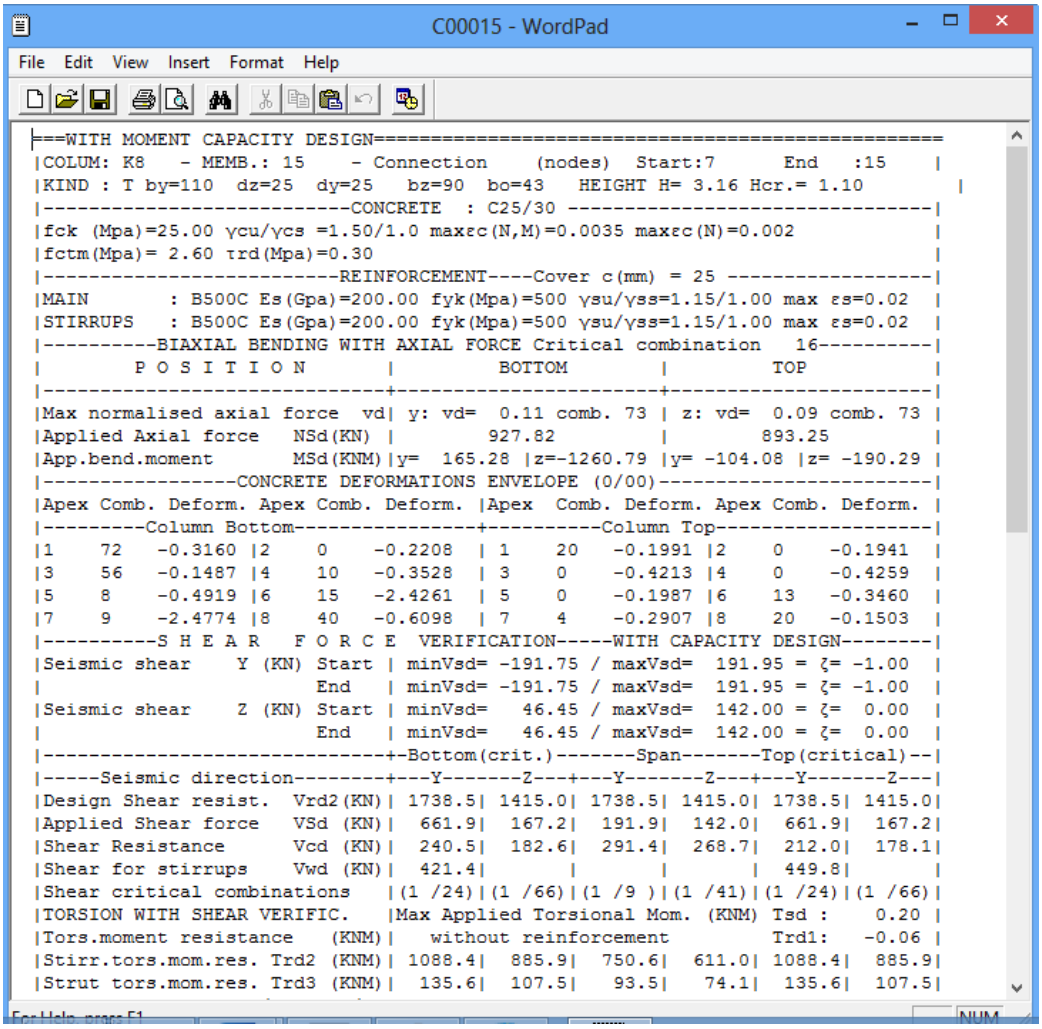
4.3.1 Editor


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

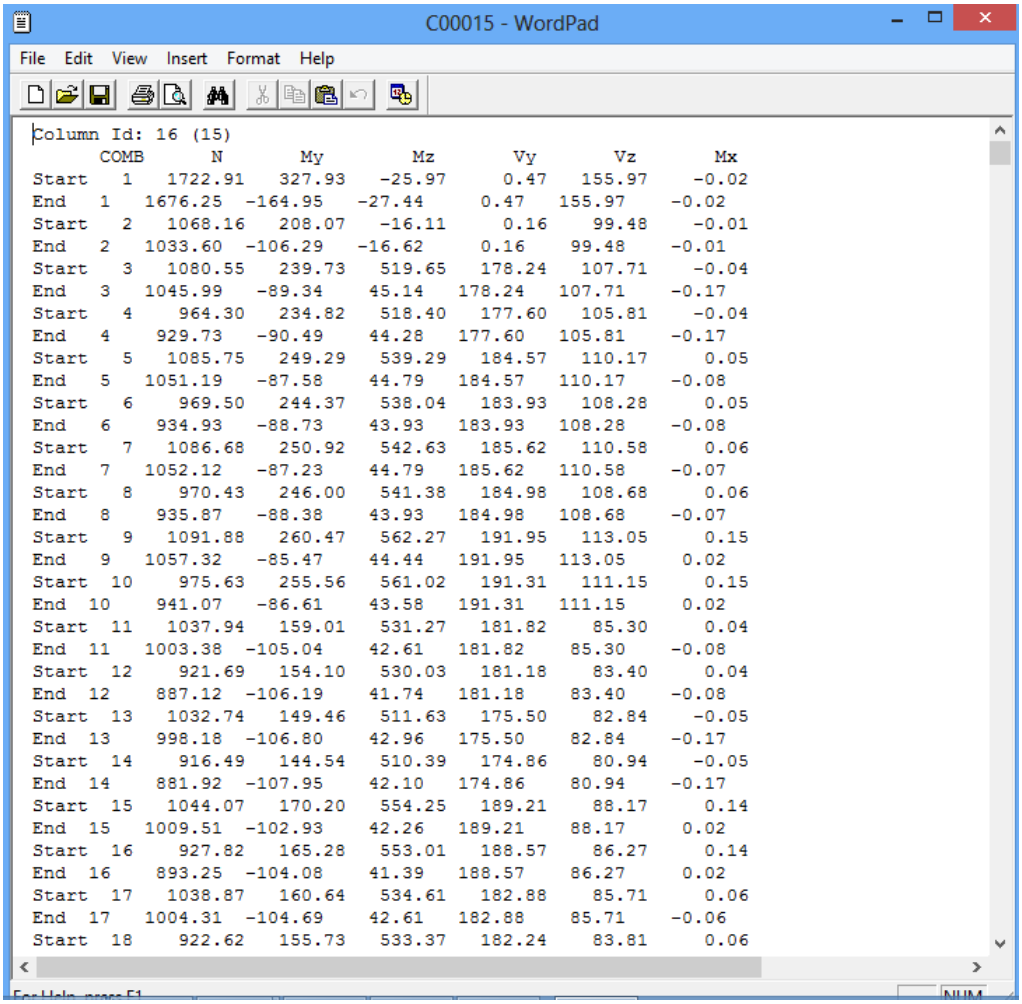

Select to display the .txt file containing the main checks and design results of columns and walls. Select the command and left click on beam’s axis to open the .txt file and read the results. The *txt files displayed are those generated by the program for the printout.




```
|==WITH MOMENT CAPACITY DESIGN=================================================
|COLUM: K8    - MEMB.: 15     - Connection     (nodes)  Start:7       End   :15    |
|KIND : T by=110  dz=25  dy=25   bz=90  bo=43   HEIGHT H= 3.16 Hcr.= 1.10        |
|----------------------------CONCRETE  : C25/30 ---------------------------------|
|fck (Mpa)=25.00 γcu/γcs =1.50/1.0 maxεc(N,M)=0.0035 maxεc(N)=0.002              |
|fctm(Mpa)= 2.60 τrd(Mpa)=0.30                                                   |
|-------------------------REINFORCEMENT----Cover c(mm) = 25 -------------------|
|MAIN       : B500C Es(Gpa)=200.00 fyk(Mpa)=500 γsu/γss=1.15/1.00 max εs=0.02   |
|STIRRUPS   : B500C Es(Gpa)=200.00 fyk(Mpa)=500 γsu/γss=1.15/1.00 max εs=0.02   |
|----------BIAXIAL BENDING WITH AXIAL FORCE Critical combination   16----------|
|        P O S I T I O N       |         BOTTOM        |          TOP            |
|------------------------------+-----------------------+-----------------------|
|Max normalised axial force  vd| y: vd=  0.11 comb. 73 | z: vd=  0.09 comb. 73 |
|Applied Axial force   NSd(KN) |        927.82         |        893.25         |
|App.bend.moment       MSd(KNM)|y=  165.28 |z=-1260.79 |y= -104.08 |z= -190.29 |
|-----------------CONCRETE DEFORMATIONS ENVELOPE (0/00)-----------------------|
|Apex Comb. Deform. Apex Comb. Deform. |Apex  Comb. Deform. Apex Comb. Deform. |
|---------Column Bottom----------------+----------Column Top-------------------|
|1    72   -0.3160 |2    0    -0.2208  | 1    20   -0.1991 |2    0    -0.1941  |
|3    56   -0.1487 |4    10   -0.3528  | 3    0    -0.4213 |4    0    -0.4259  |
|5    8    -0.4919 |6    15   -2.4261  | 5    0    -0.1987 |6    13   -0.3460  |
|7    9    -2.4774 |8    40   -0.6098  | 7    4    -0.2907 |8    20   -0.1503  |
|----------S H E A R   F O R C E  VERIFICATION-----WITH CAPACITY DESIGN--------|
|Seismic shear    Y (KN) Start | minVsd= -191.75 / maxVsd=  191.95 = ζ= -1.00   |
|                        End   | minVsd= -191.75 / maxVsd=  191.95 = ζ= -1.00   |
|Seismic shear    Z (KN) Start | minVsd=   46.45 / maxVsd=  142.00 = ζ=  0.00   |
|                        End   | minVsd=   46.45 / maxVsd=  142.00 = ζ=  0.00   |
|------------------------------+-Bottom(crit.)-------Span-------Top(critical)--|
|-----Seismic direction---------+---Y-------Z---+---Y-------Z---+---Y-------Z---|
|Design Shear resist.  Vrd2(KN)| 1738.5| 1415.0| 1738.5| 1415.0| 1738.5| 1415.0|
|Applied Shear force   VSd (KN)|  661.9|  167.2|  191.9|  142.0|  661.9|  167.2|
|Shear Resistance      Vcd (KN)|  240.5|  182.6|  291.4|  268.7|  212.0|  178.1|
|Shear for stirrups    Vwd (KN)|  421.4|       |       |       |  449.8|       |
|Shear critical combinations   |(1 /24)|(1 /66)|(1 /9 )|(1 /41)|(1 /24)|(1 /66)|
|TORSION WITH SHEAR VERIFIC.   |Max Applied Torsional Mom. (KNM) Tsd :    0.20 |
|Tors.moment resistance   (KNM)|   without reinforcement         Trd1:   -0.06 |
|Stirr.tors.mom.res. Trd2 (KNM)| 1088.4|  885.9|  750.6|  611.0| 1088.4|  885.9|
|Strut tors.mom.res. Trd3 (KNM)|  135.6|  107.5|   93.5|   74.1|  135.6|  107.5|
```

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

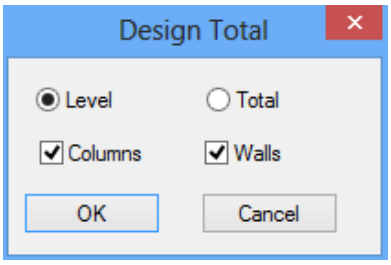
⚠ Use this command for editing a beam continuity in an integrated environment of verification and design. You can also apply Retrofitting methods and calculate the new moment diagrams.

⚠ 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)

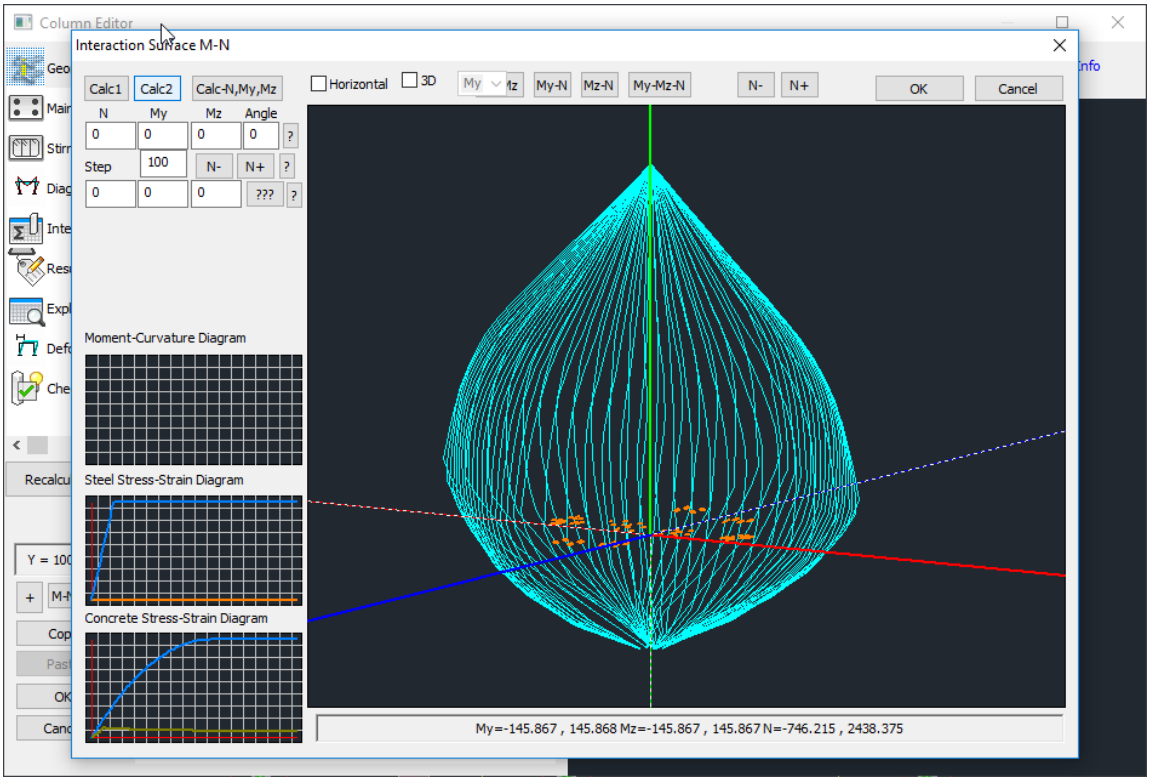
This command refers to the nonlinear static analyses (Pushover analyses). First, the existing structure is designed in accordance with the design provisions that were active the time of the construction of the structure. This means that the dimensions of the structural elements as well as the existing 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.

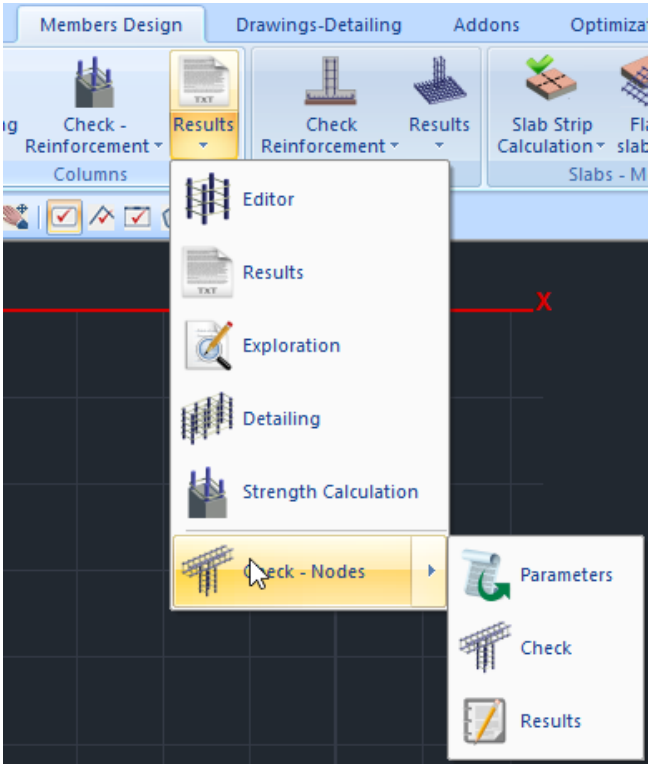


⚠ The calculation of the interaction diagram M-N can also be performed for a single column or wall, using the command "Detailing" (see the corresponding **chapter B. Column's Detailing**).

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

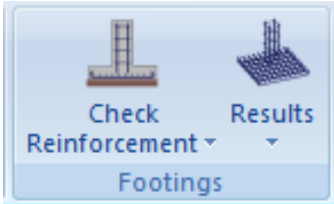


NOTE:

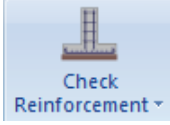
- This check concerns only the Greek KAN.EPE norm and is no valid in other countries.
- It works only in the Greek version.

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).

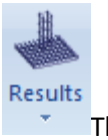
5.1.3 Single Deletion

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.

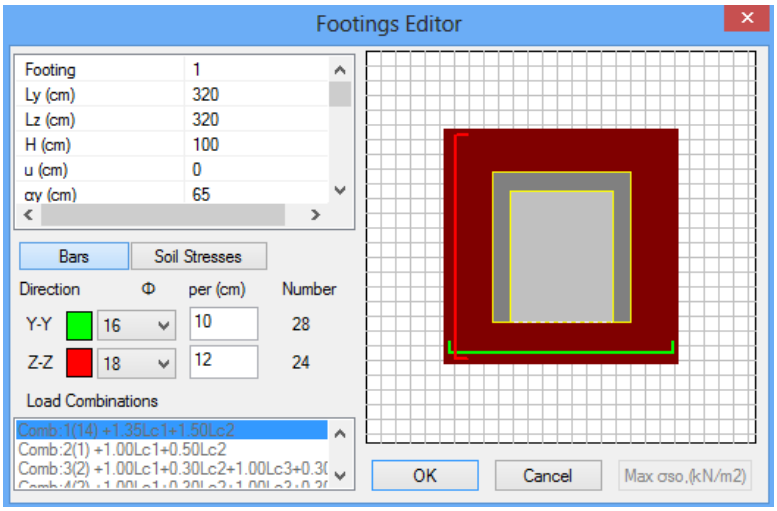
5.2 Results



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:

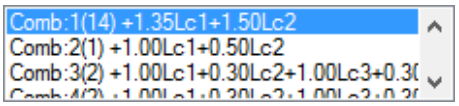
Direction	Φ	per (cm)	Number
Y-Y	16	10	28
Z-Z	18	12	24

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.

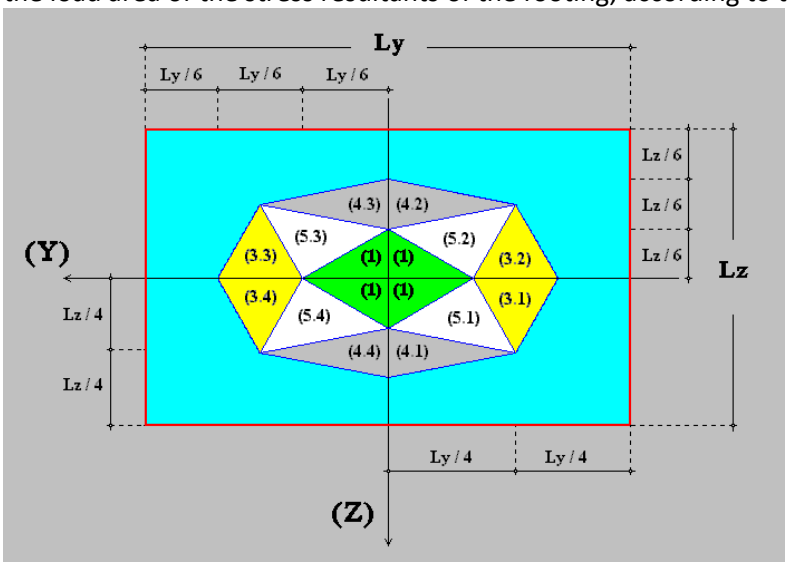
In the upper left section, a table is displayed with the title and the values of the geometric data of the footing, and other data related to soil.

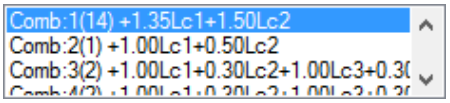
Footing	1	dz (cm)	205
Ly (cm)	320	hsoil (cm)	0
Lz (cm)	320	Footing Weight (KN)	256.00
H (cm)	100	Soil Weight(KN)	0.00
u (cm)	0	σal.(kN/m2) (102)	121.712
αy (cm)	65	σfr.(kN/m2) (36)	385.601

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

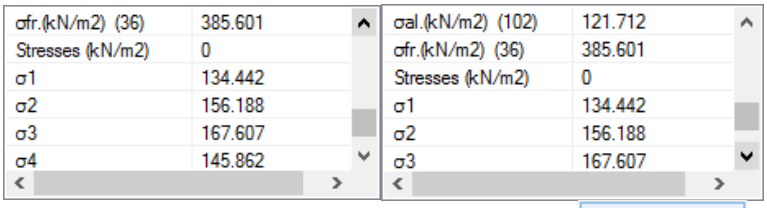


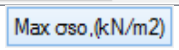
In the figure on the right, the soil stresses on the 4 vertices of the footing are displayed, indicating the load area of the stress resultants of the footing, according to the sketch below.

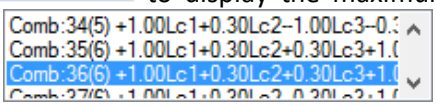




Choose a combination and scroll the list to read the respective stresses σ1, σ2, σ3, σ4 and the critical stresses σ_{al} and σ_{fr}:



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

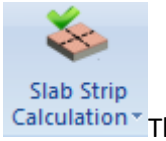

Select this command to display the TXT file that contains all design checks for footings and the corresponding results. Select the command and left click on the beam’s centroid axis to open the TXT file and read the results.

---

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.

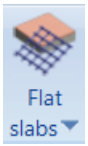
6.1.3 Single (Load Pattern)


Perform analysis of the selected strips in accordance with the load pattern as well.

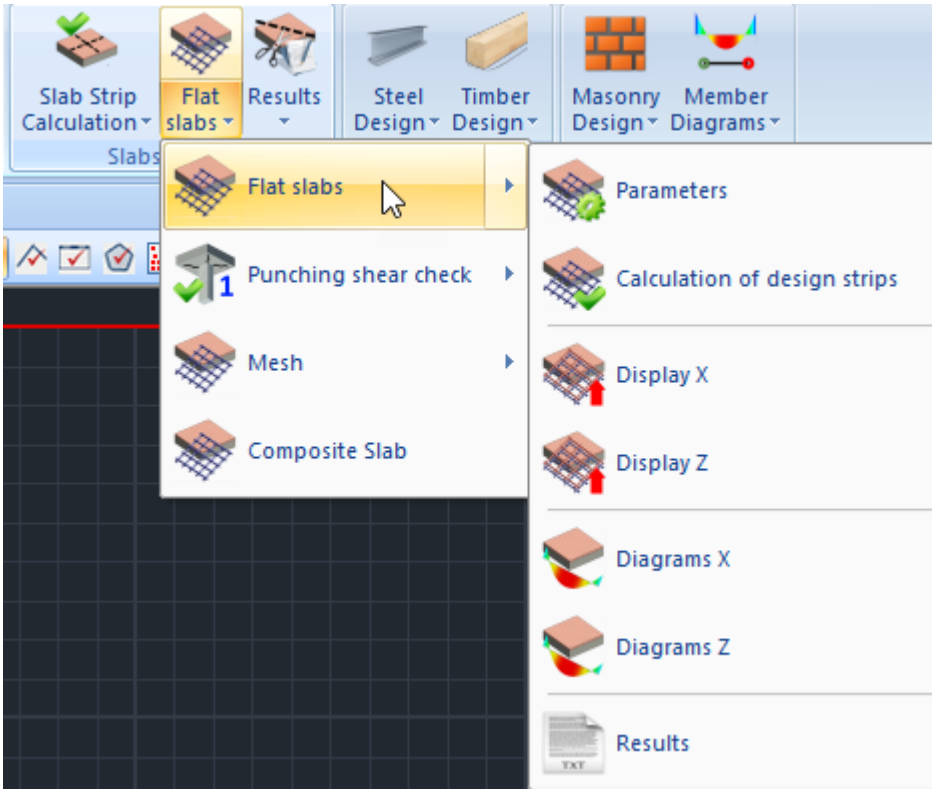
6.1.4 Overall (Load Pattern)


Perform analysis of all the current level strips in accordance with the load pattern as well.

6.2 Flat Slabs



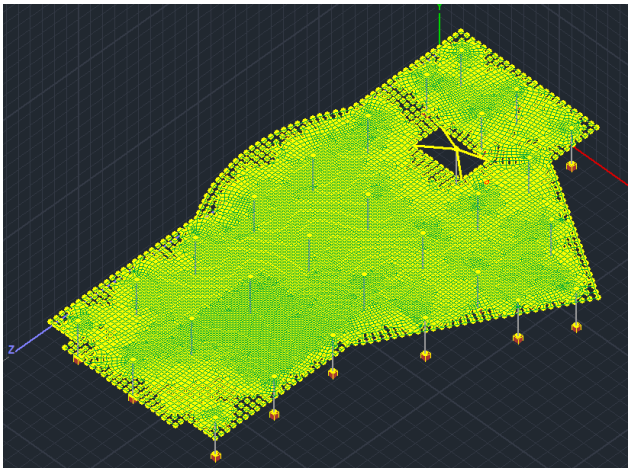
6.2.1 Flat Slabs



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

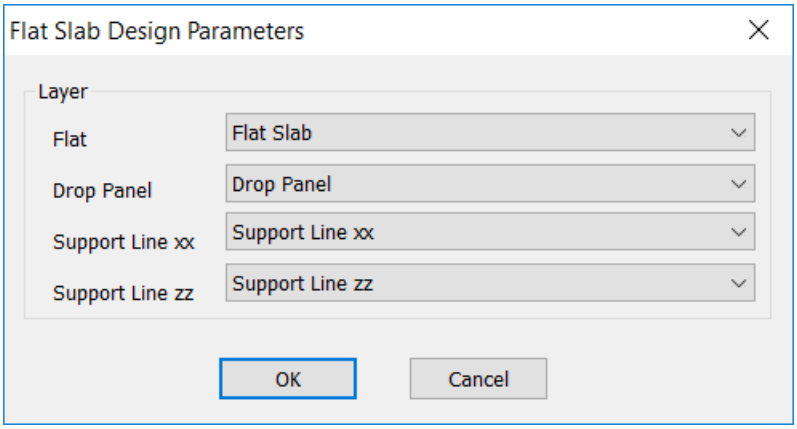
The procedure for the modeling of flat slabs requires:

- 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.

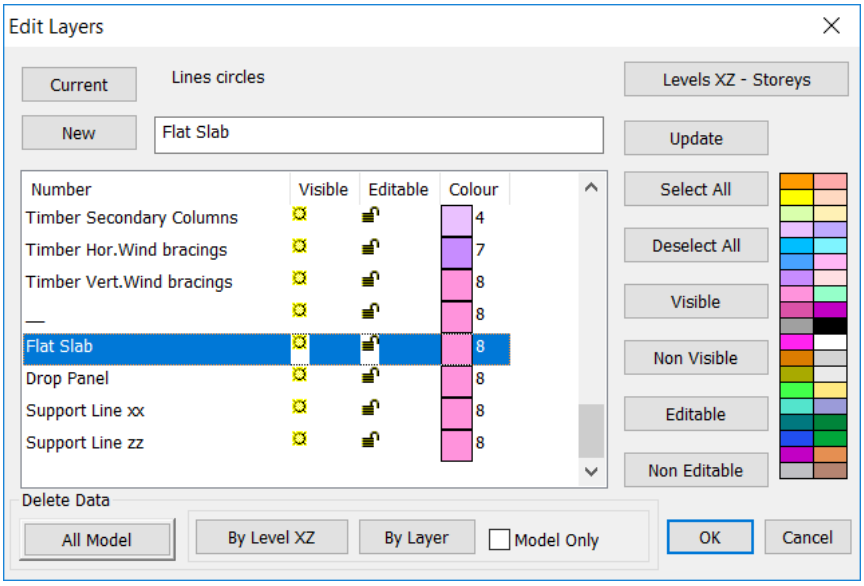


The command "Flat slabs" includes the commands:

6.2.1.1 Parameters

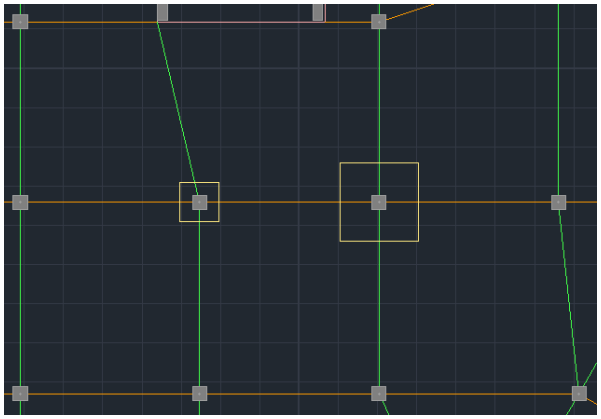


In the dialog box, you define the correlation between Layers.



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.



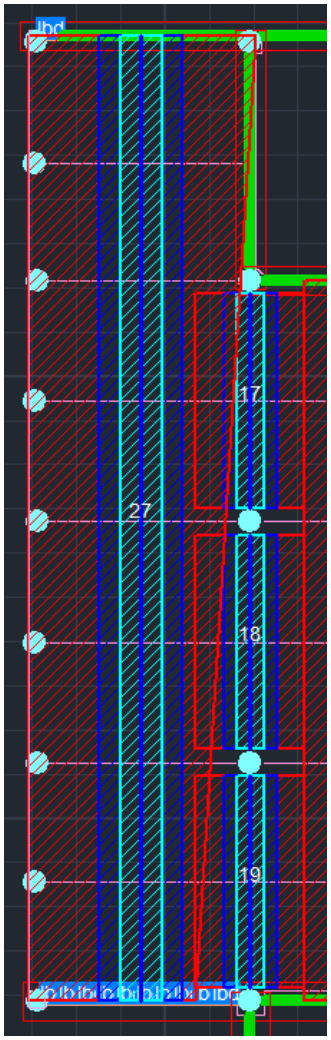
Drop Panels and Support Lines

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

§ Instructions for inserting the support lines in flat slabs

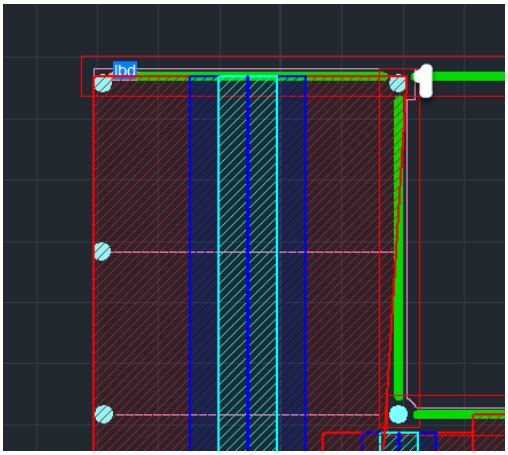
1. The support lines should start from a column and end to a column (or free end). In any case, they must contain at least one column.
2. The support line must reach the outline of the slab as long as it is a free end. Otherwise, it may stop on the outline or on the node of the column.
3. If the boundary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

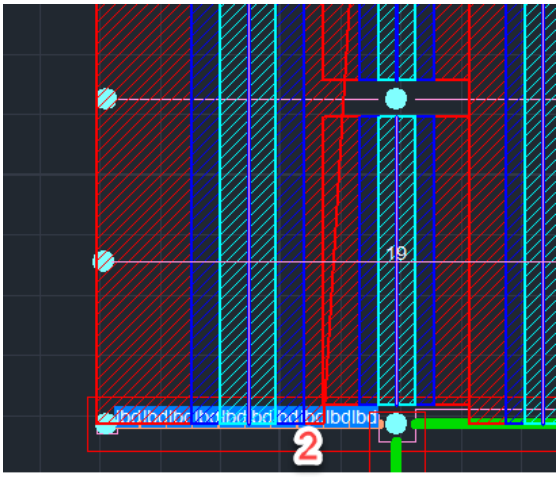


On the supported Line 27, the red right line has not formed correctly (it is the red inclined line). This happened because a support line was inserted which:

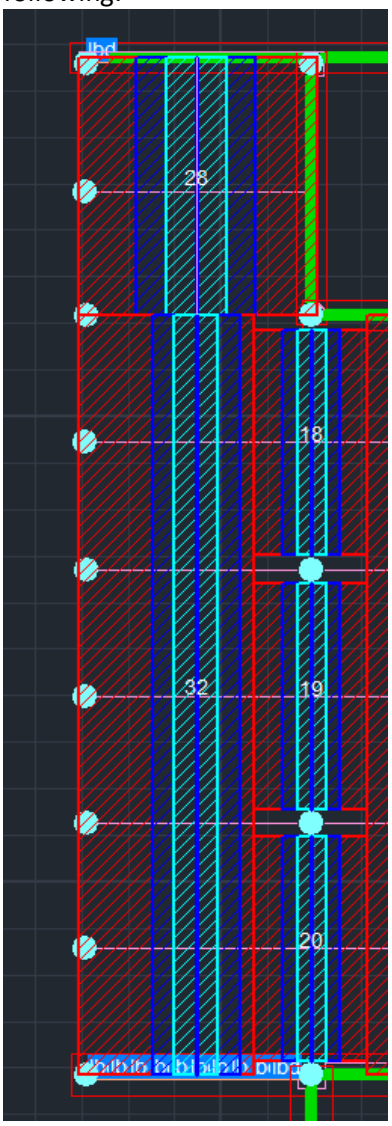
At the right top, it is bordered by beams, that is, by the slab boundary, and so the program has delineated the red out area at this slab boundary (point 1)



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



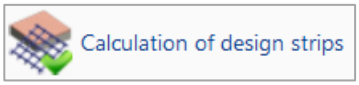
For that reason, the inclined boundary of the red area emerged.
But, in case we had inserted two support lines, one for the upper part(with the slab boundary) and one for the bottom part (boundary with the support lines 17,18,19) the correct result is the following:



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 line is now formed, that is, the 28 and 32.

6.2.1.2 Calculation of design strips

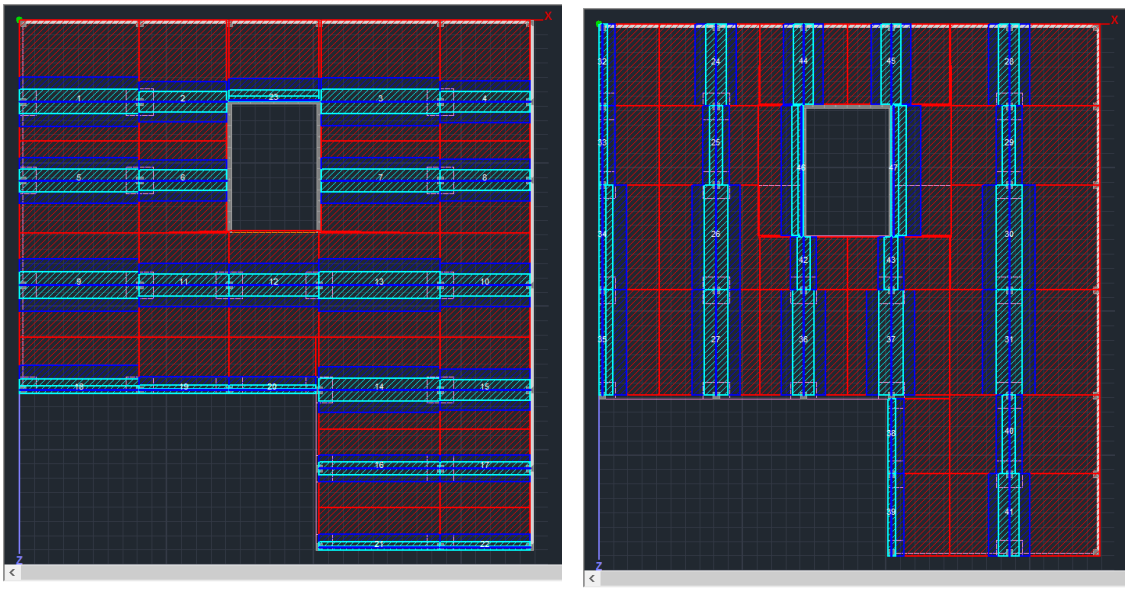


According to Annex I of EC2 flat slab is divided into design strips. These are the areas that are automatically created by the program on both sides of the Support Line, according to Figure I.1 of EC2.
Select the command Calculation of design strips and the program automatically creates them.

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

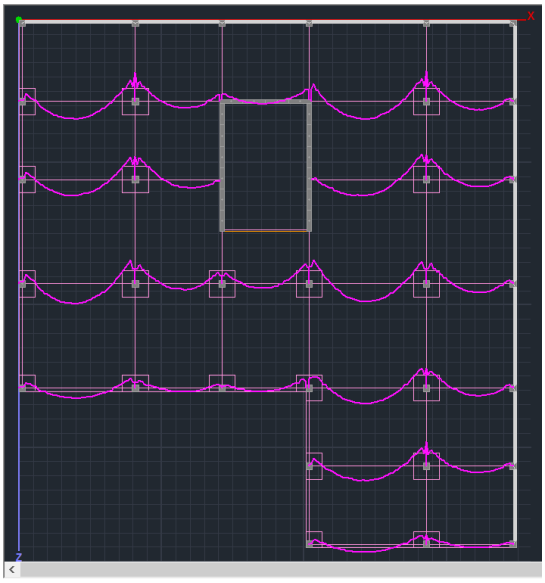
Select to display the Design Strips in both directions.

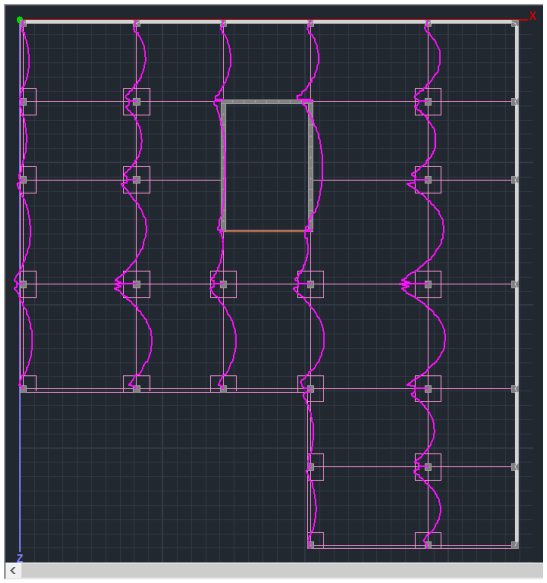


Design Strips along the X and Z axes

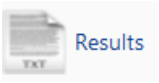
6.2.1.4 Diagrams X, Z

Select Diagrams in both directions to see the corresponding diagrams.





6.2.1.5 Results



This command opens the Results file through the Report.

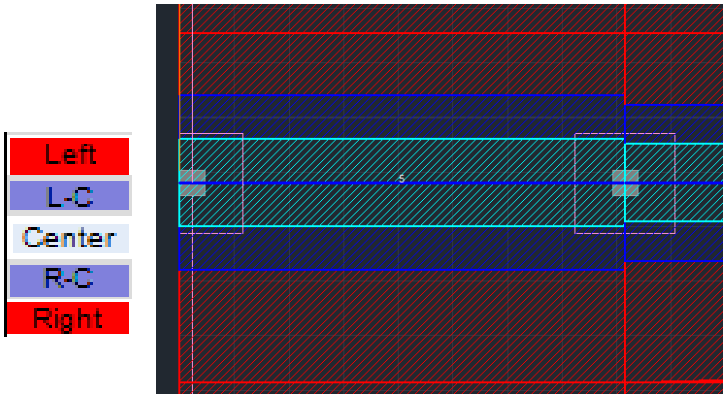
Each page concerns a Strip Line.

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



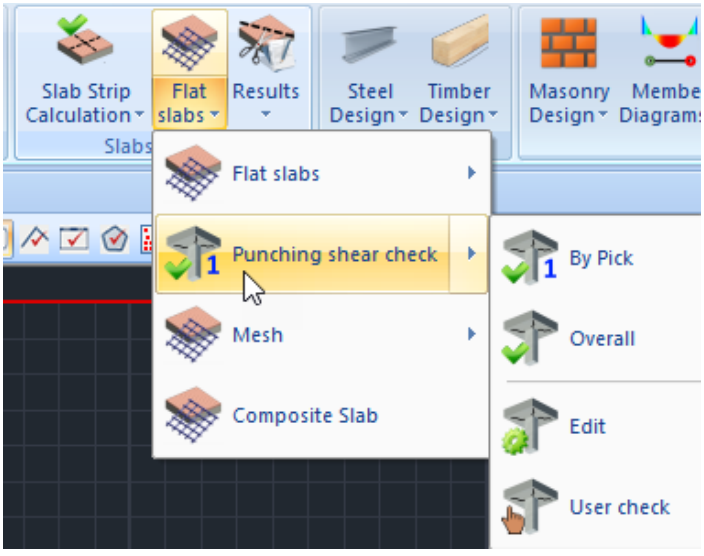
Analysis Results and Reinforcement										Top
	203.87 cm (L_{start})					407.74 cm (L_{centre})				
Zone	M (kNm)	Width (cm)	$A_{s,rqd}$ (cm^2/m)	$A_{s,prvd}$ (cm^2/m)	Φ/s	M (kNm)	Width (cm)	$A_{s,rqd}$ (cm^2/m)	$A_{s,prvd}$ (cm^2/m)	Φ/s
Left		400.0		3.246	8/15		401.1		0.812	8/20
L-C		400.0		3.246	8/15		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

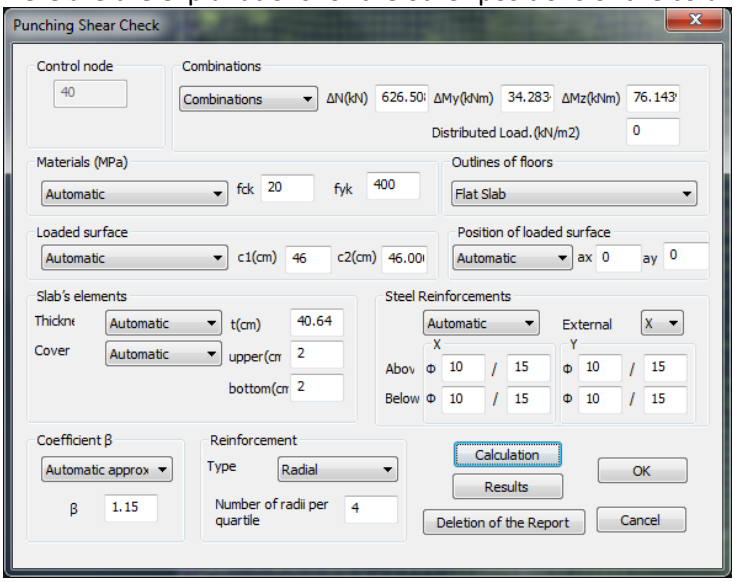
6.2.2 Punching shear checks

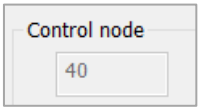


The punching checks are also added in the new version in accordance with EC2. The check is part of the check and design process of the flat slabs which runs automatically, but can also run as an individual check for any column. All data can be set automatically or even manually.

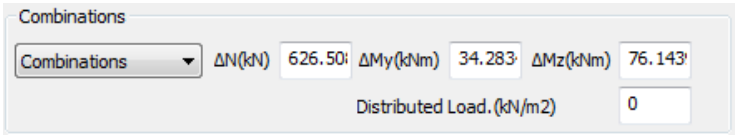
6.2.2.1 Pick

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.

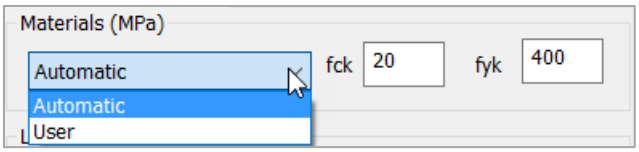


In Combinations field:

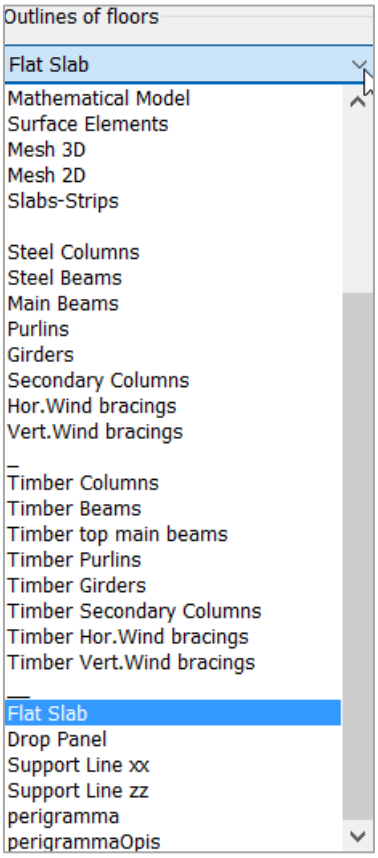
- By choosing Combinations, the program finds automatically the combination that gives the worst Axial ΔN resulting and displays its value along with the corresponding moments.
- Choosing User, allows user values to be set for axial force and moments, in their respective fields, as well as the definition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

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

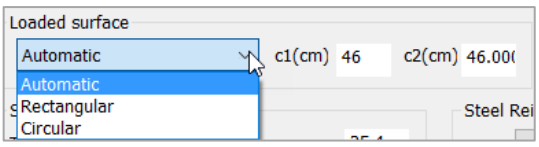
Shear force ($V_{Ed,init.}$)	626.5	(kN)
Distributed load (p)	50.0	(kN/m^2)
Reduced shear force ($V_{Ed,fin.}$)	478.8	(kN)



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.



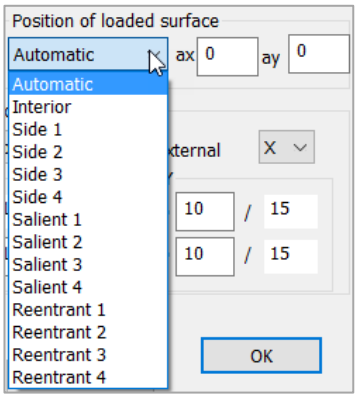
In Outlines of Floors, choose the Layer containing the contour of the slab including the columns located in the outline.
Then select the layer Flat Slab that includes exactly the lines that define the overall contour of the slab.



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 own dimensions c1 and c2 for calculating the loaded rectangular surface
- Circular, the user defines his own diameter c1 for calculating the loaded circular surface

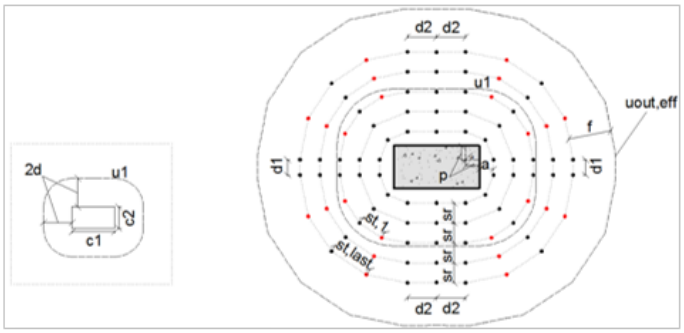


The Position of the loaded surface may be determined either automatically or selectively. It depends on the contour of the slab and the position of the selected column in it.

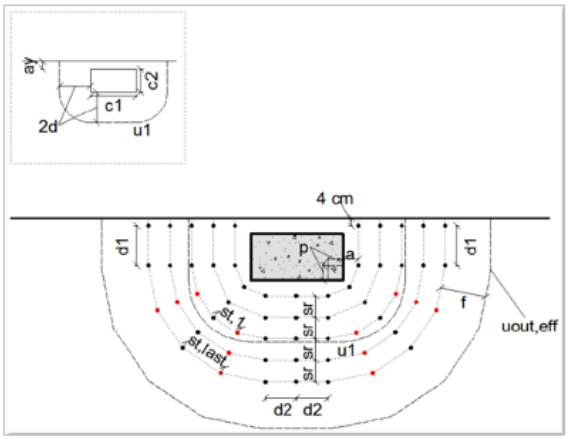
The proposed positions are:

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

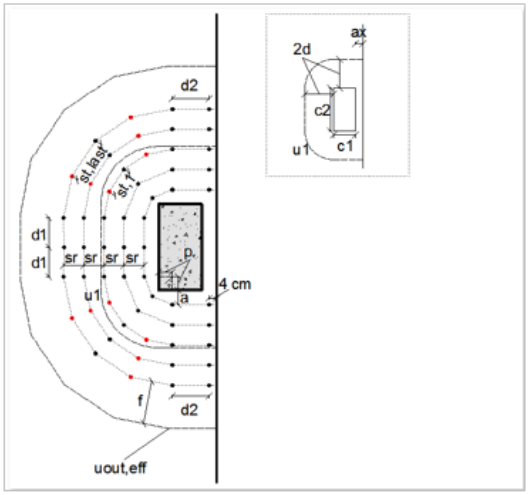
Select the position of the selected column and set the distances from the perimeter, ax, and ay, (except indoor) according to the following figures:



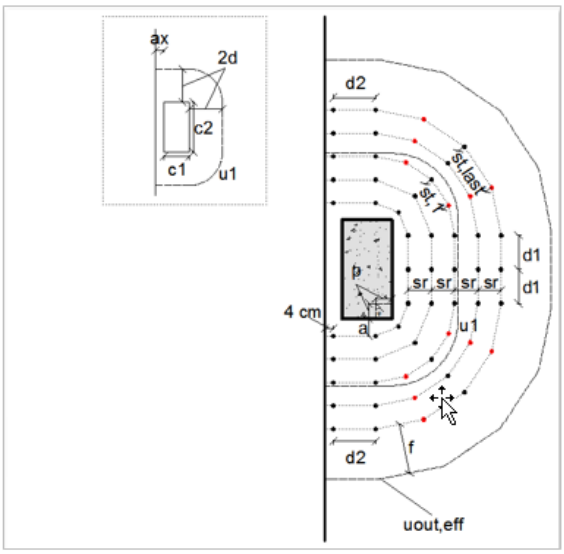
INTERIOR



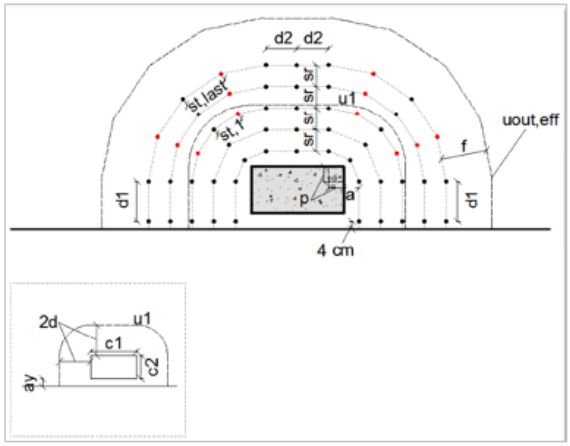
SIDE 1



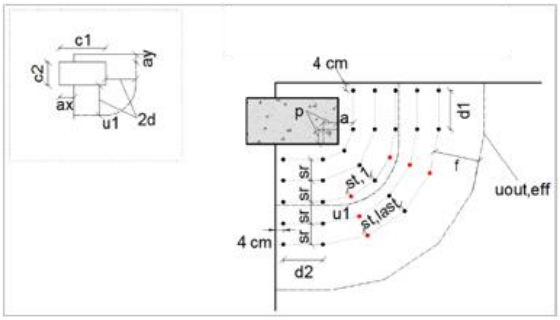
SIDE 2



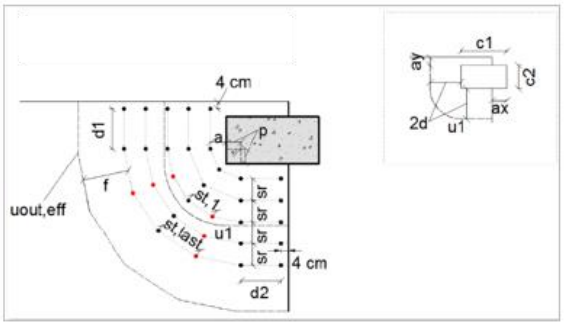
SIDE 4



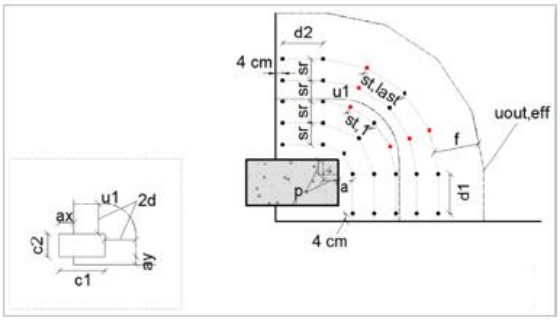
SIDE 3



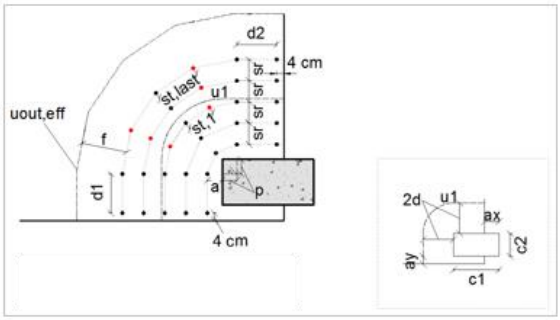
SALIENT 1



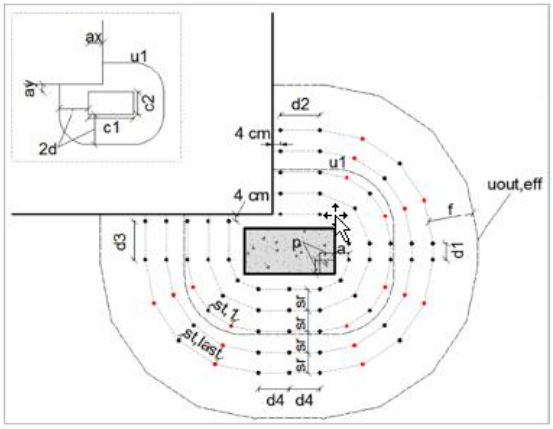
SALIENT 2



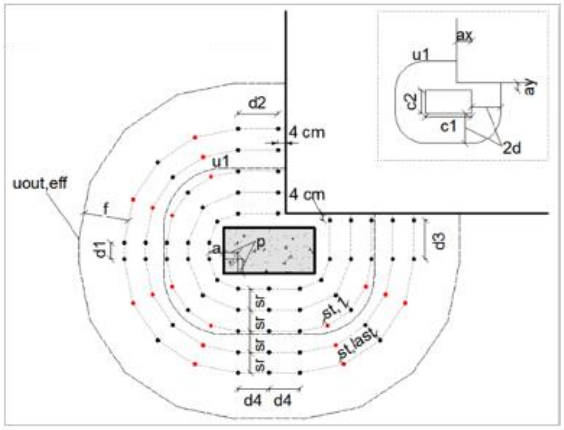
SALIENT 4



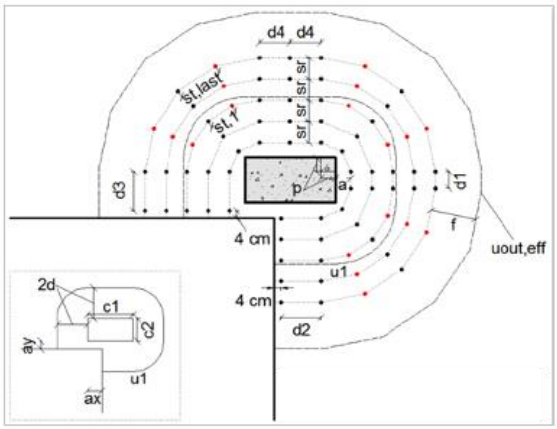
SALIENT 3



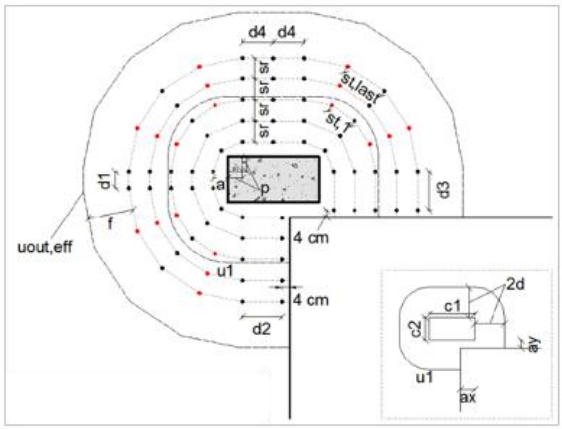
REENTRANT 1



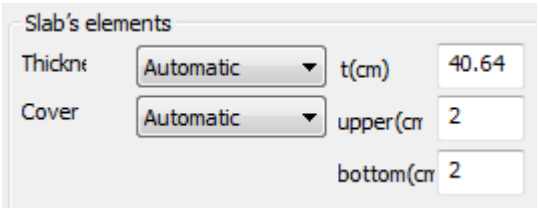
REENTRANT 2



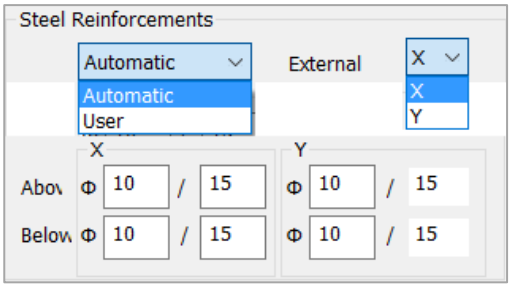
REENTRANT 4



REENTRANT 3



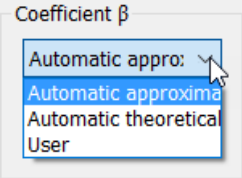
The Thickness and Cover of the Slab either considered Automatically or modified by the User, by typing the corresponding values for Thickness and Cover, upper and bottom, of the slab.



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.



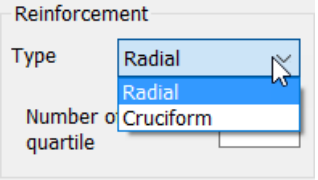
The Coefficient β for punching calculation can be calculated automatically in two ways:

- Automatic approximation or
- Automatic theoretical.

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

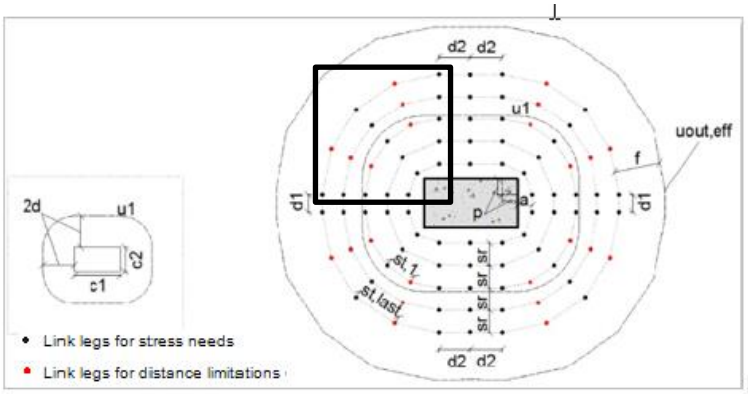
⚠ The Theoretical way is a function of the Moments My, Mz.

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

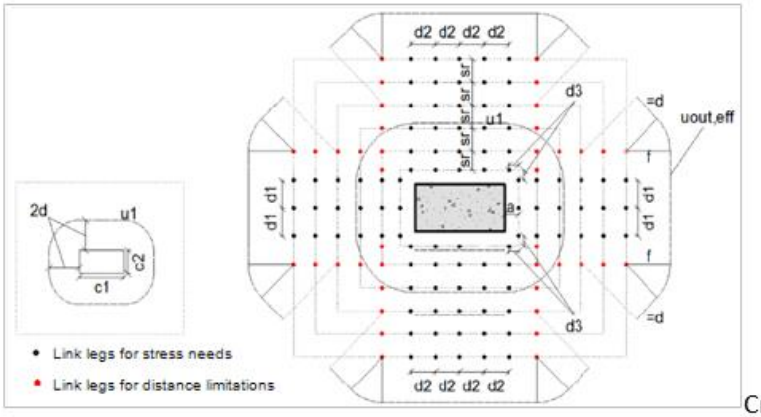


Finally, in Reinforcement set the layout type of punching reinforcement choosing between Radial and Cruciform.
For the Radial layout set the number of radii per quartile.

In the figure below the black box contains one quartile of the radial layout. The first perimeter has 3 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

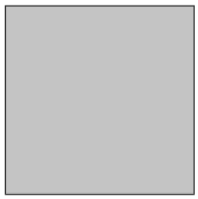
Calculation The Calculation command performs all the necessary checks for punching, taking into account all the above parameters.

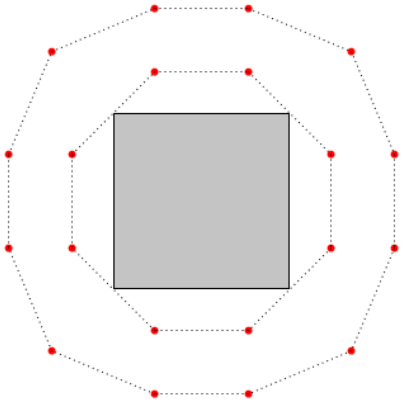
Results This command displays the results file:

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

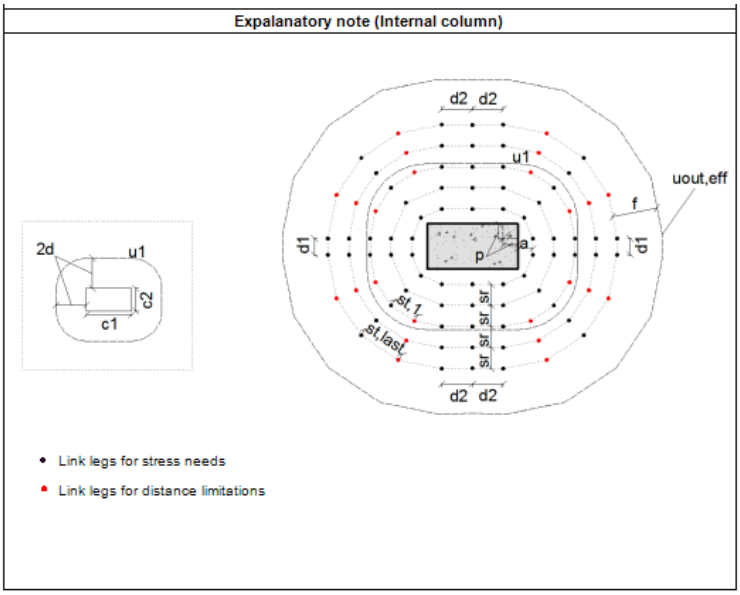
Description	Value	Units	EC2	Περιγραφή	Τιμή	Μονάδες	EC2
Perimeter $u_{out,ef}$	483.7	(cm)	(eq6.54)	Distance (d_3)		(cm)	
(a) - Distance of 1st perimeter of reinforcement from the loaded area	11.2	(cm)		Distance (d_4)		(cm)	
Limit: $0.3 \cdot d \le a \le 0.5 \cdot d$	6.7<=a<=11.2		(9.4.3)	Angle (φ)	90.0	°	
(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 in accordance with their respective formulas and chapters of EC2 are listed in detail.

Grouping of punching shear reinforcement

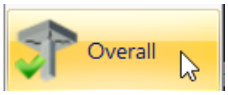
Group	Number of lines	Φ (mm)	Number of link legs on line	Height of link leg (cm)	Perimeter where the 1st link leg of the line stands	Distance of the 1st link leg from the loaded area
1	8	10	2	21.4	1	11.20
2	4	10	1	21.4	2	27.70

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 in accordance with the position of the Loaded Surface.

6.2.2.2 Overall

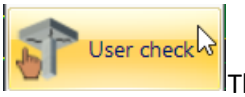


This is the command for making the punching checks in all columns included in the outline of the flat slab, automatically, using the parameters automatically from the program. The same parameters are used for all columns.

6.2.2.3 Edit

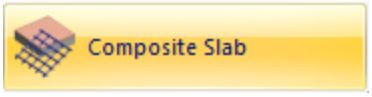
The Edit command allows you to change the parameters set by the “By Pick” check or “Overall” check.
Select the command and left-click on the node of the column to edit. Automatically the previous parameters window opens with the original settings for the selected column. You can make any changes you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

6.2.2.4 User check



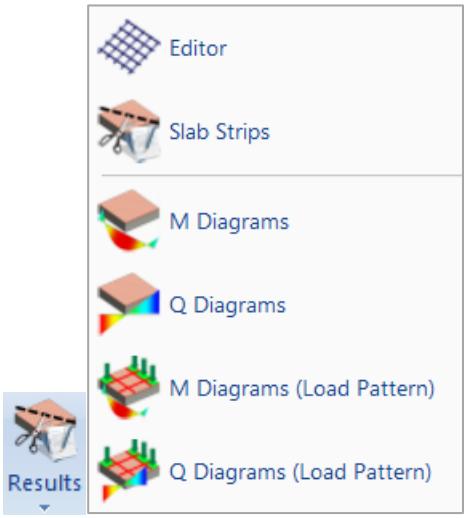
The user has the opportunity to test different data on different nodes for a supervisory result image. This is a "draft" that is not saved in the printout but allows the user to make tests in order to reach the desired solution.

6.2.3 Composite Slabs



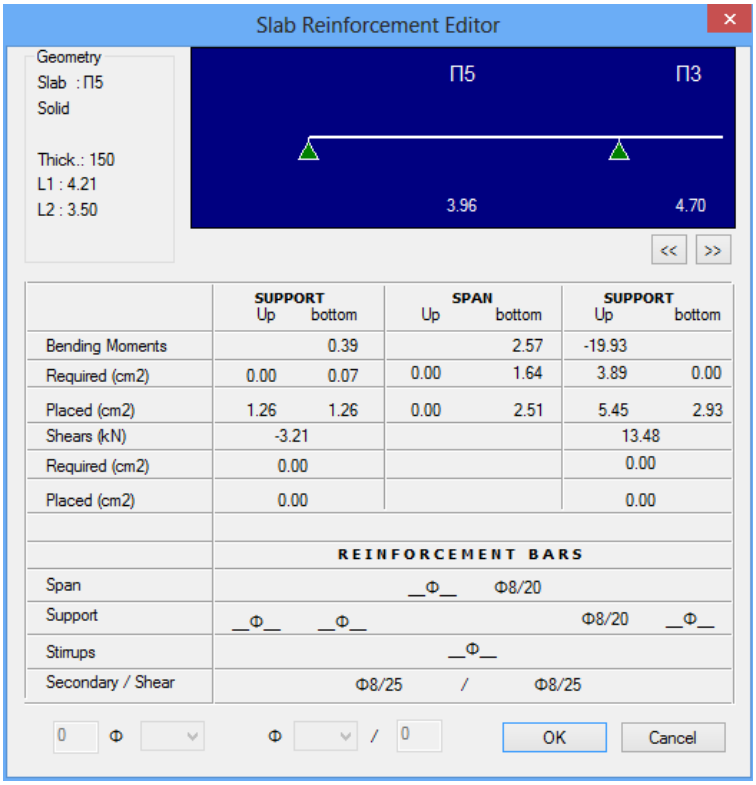
6.3 Results

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 in both directions).

In order 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” have 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)

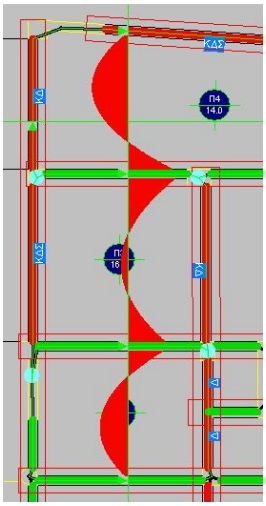
l/d	l/d perm.	Suf.	Suggested.min thick. hs(mm)	Max. M (kNm)	dul (mm)	a	l/a (perm.) (mm)	Suf.
34.59	80.10	YES	77	-7.64	0.42	250	18.40	YES

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 is marked on slab symbol when the thickness is less than permitted by regulation*

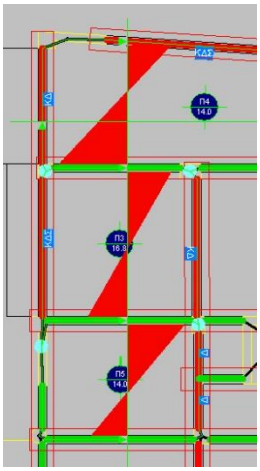
6.3.3 M Diagrams

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.

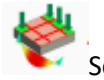


6.3.4 Q Diagrams


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.



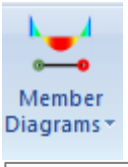
6.3.5 M Diagrams (Load Pattern)

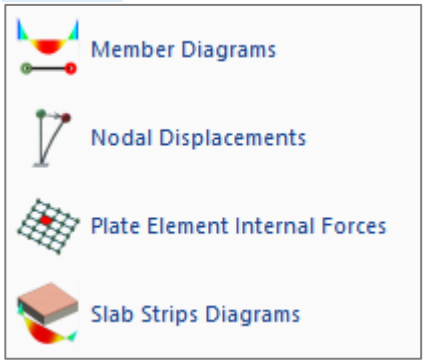

Select the command to display a qualitative representation of the moment diagrams over the slab strip, selected with left click, resulting from a load pattern.

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. Member Diagrams

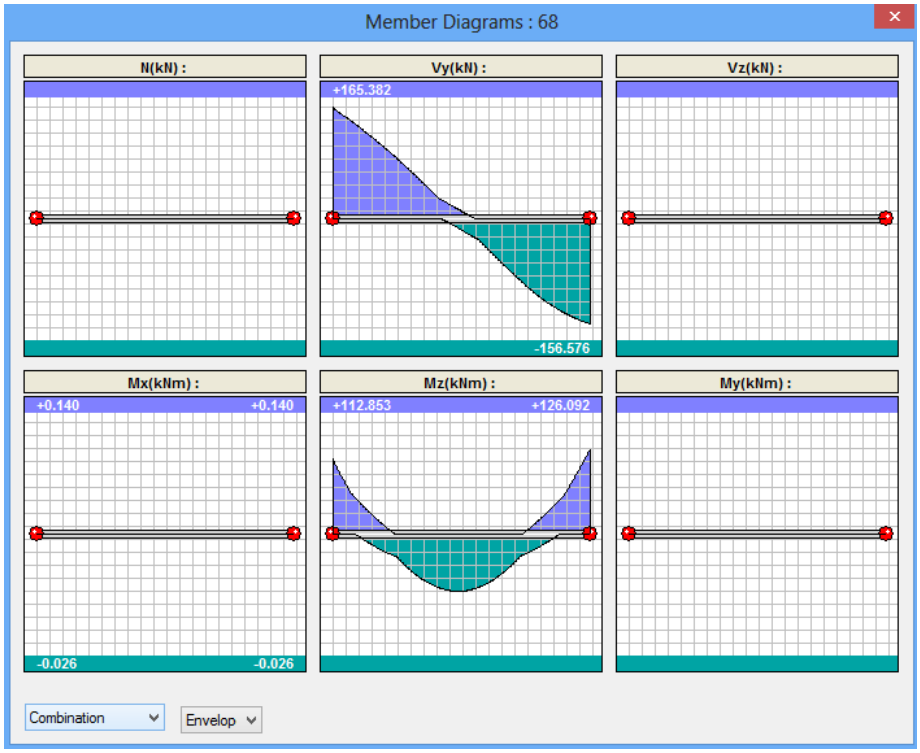




7.2 Member Diagrams

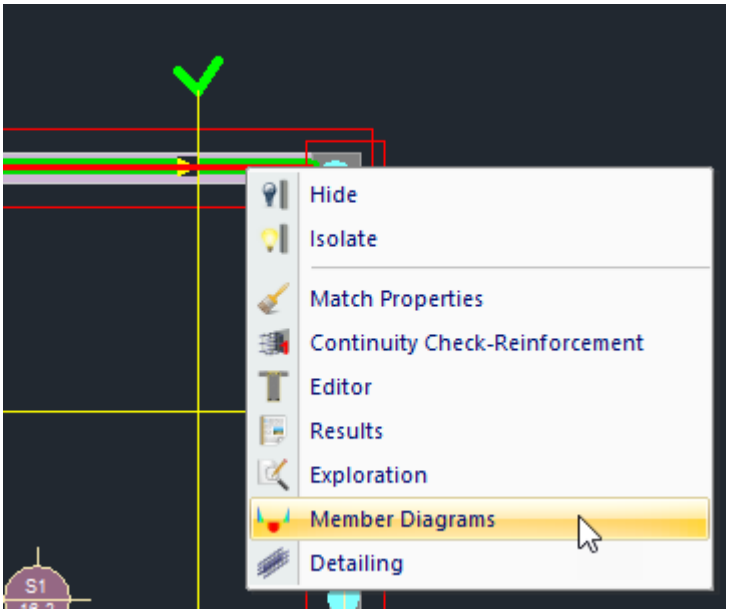
This command is used for the display of the structural members’ stress resultants diagrams in 2D.

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.



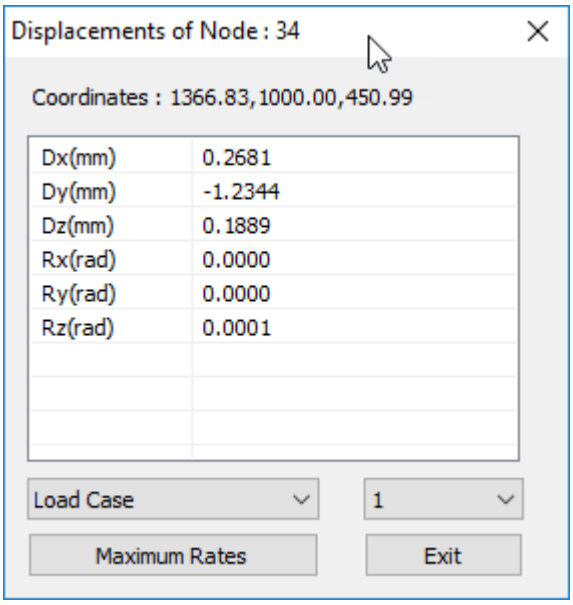
NOTE

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

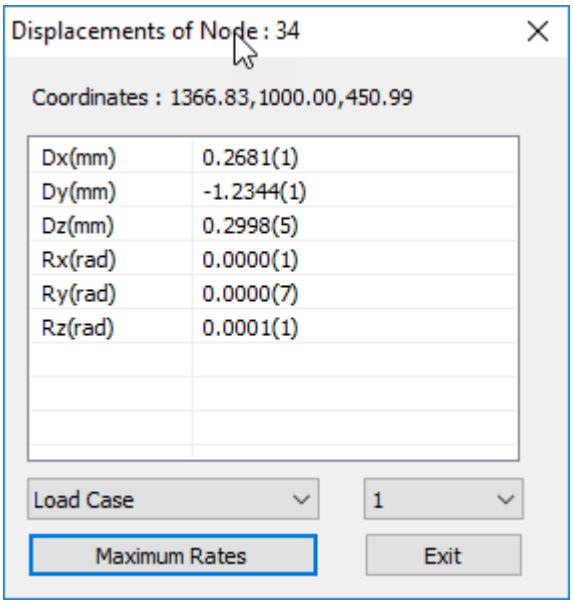


7.3 Nodes Displacements

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.

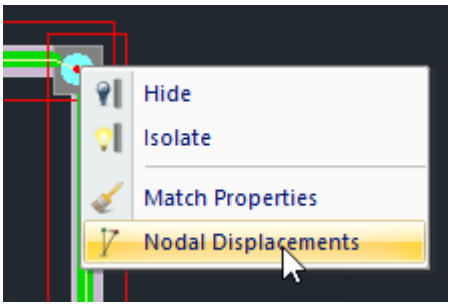


Press Maximum Rates to see the maximum value for each movement and rotation, as well as the combination from which it originates. The number of this combination is written in brackets next to the value.



NOTE

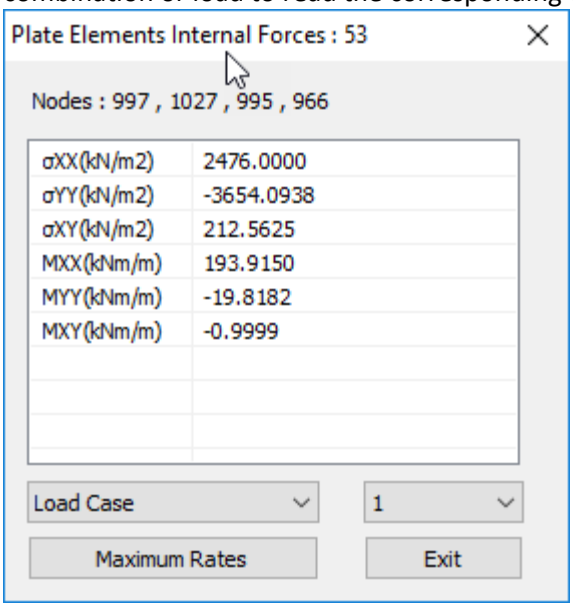
Right mouse click on a member opens a command list containing also Nodal Displacements.



7.4 Plate Elements Internal Forces

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

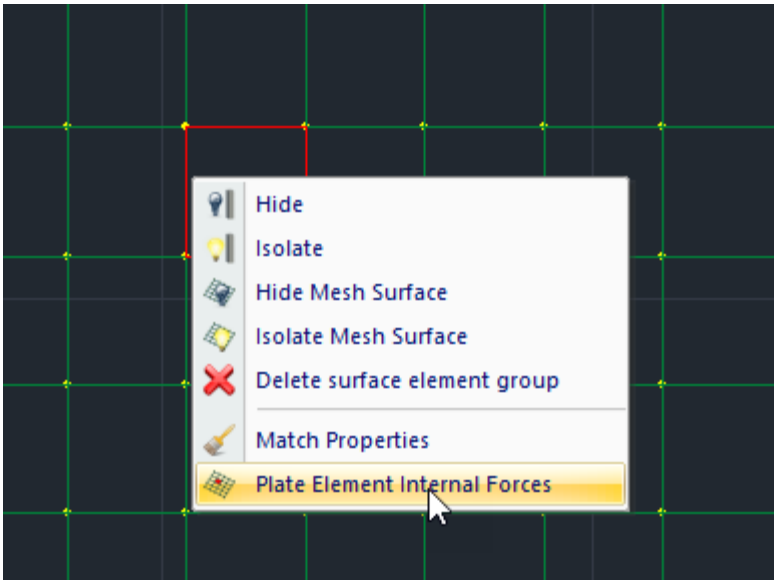
Select the command and point the mouse on a plate element.
The dialog box that opens display all stresses and moments for that plate element. Select a combination or load to read the corresponding values of stresses and moments.



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.

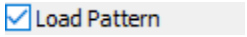
NOTE

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



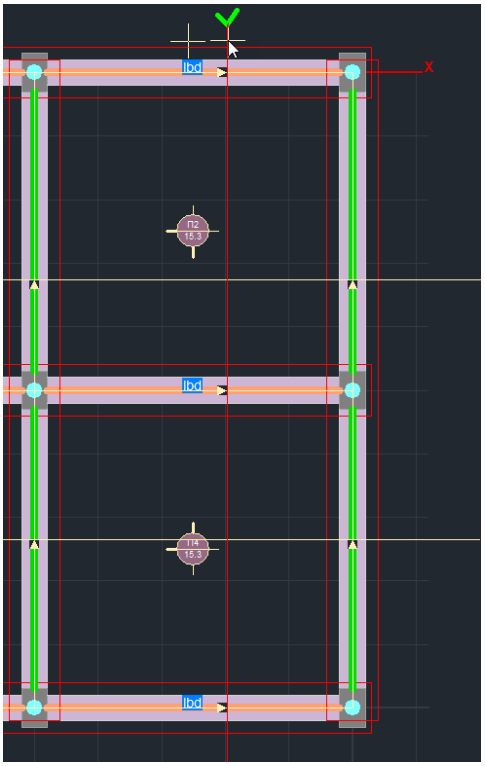
7.5 Slab Strips Diagrams

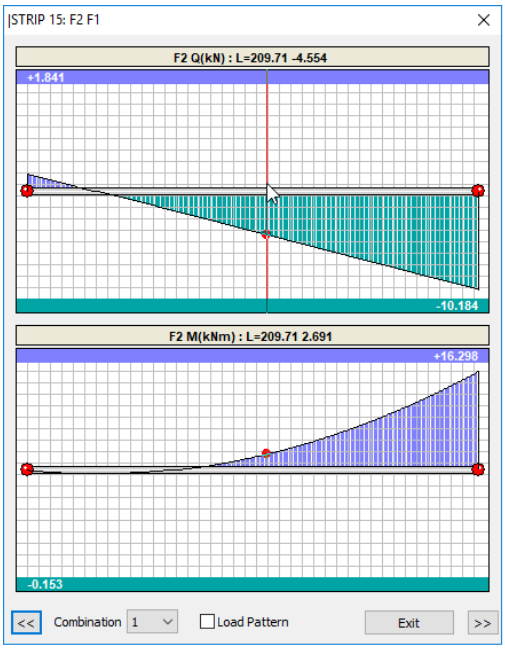
There is now the possibility to see the Slab Strips Diagrams.

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 

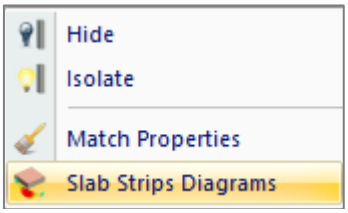
to see the diagrams resulting.

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.