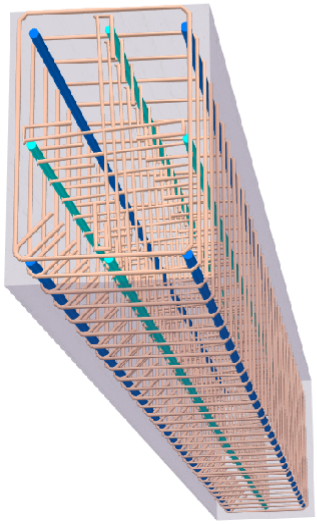
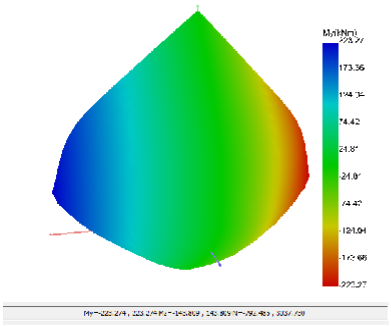
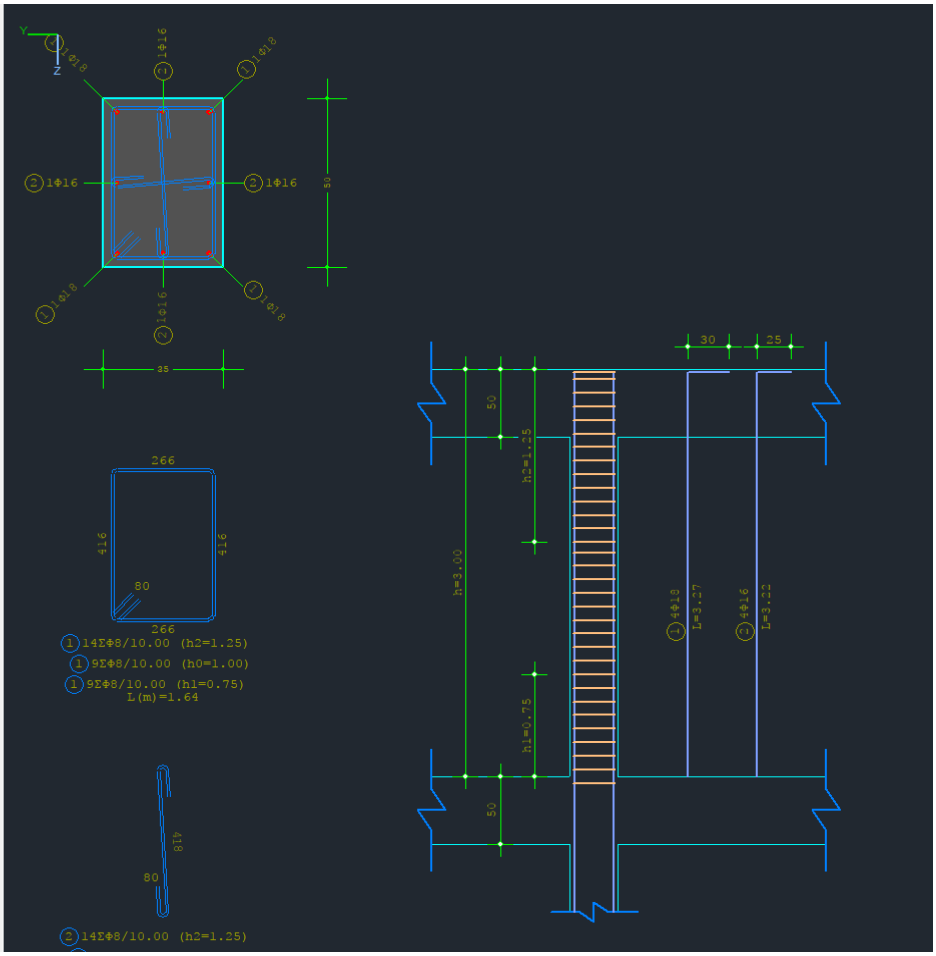
SCADA Pro™
Structural Analysis & Design

User's Manual

B.COLUMNS DETAILING



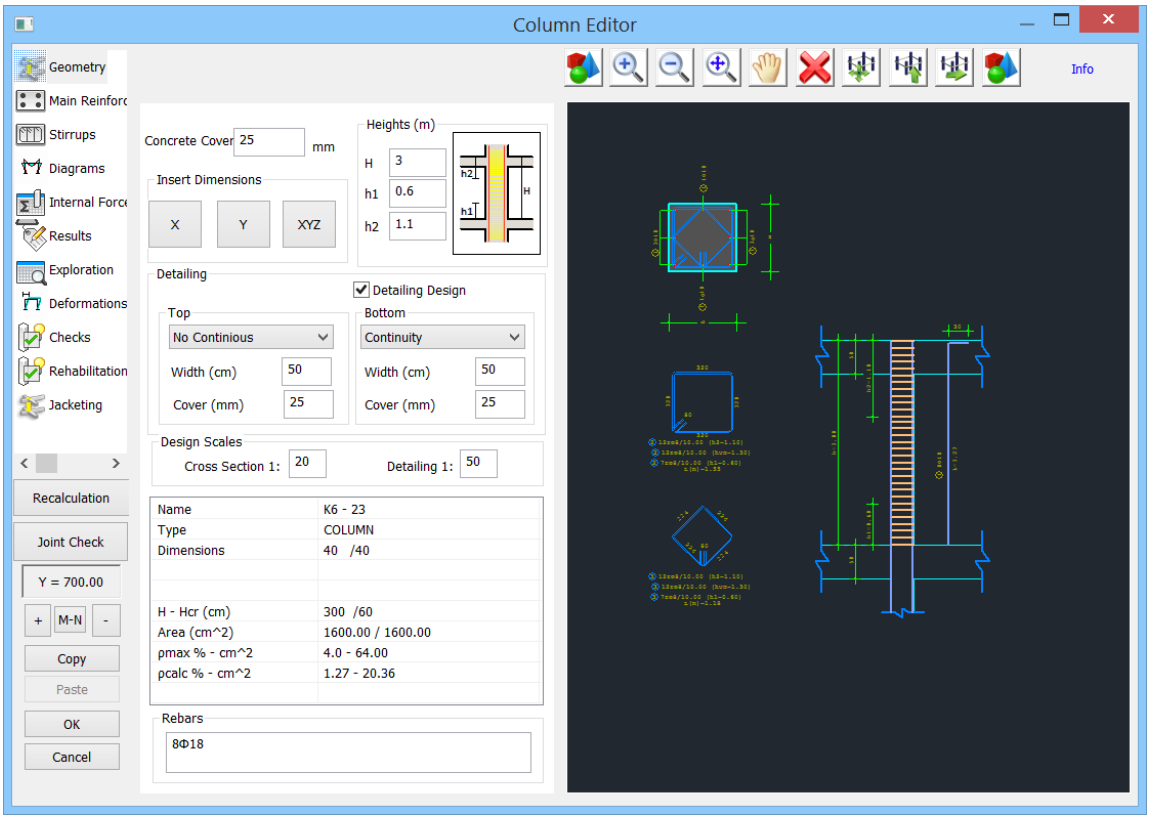




ACE HELLAS
INTEGRATED SOLUTIONS

CONTENTS

A. COLUMN'S DETAILING



The New Column's Editor - "Detailing" of SCADA Pro is part of an innovative new group of tools to manage the design details of the columns.

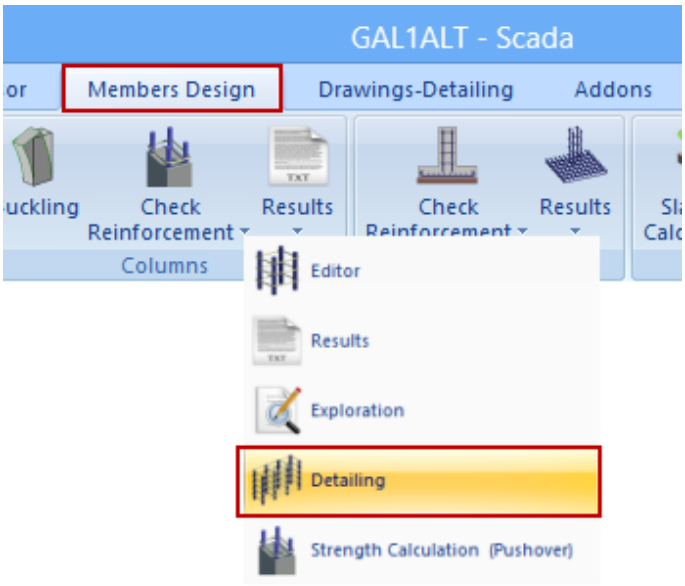
In "Detailing" you can edit, modify and complete section's details and steel reinforcement. You may as well display internal forces, diagrams, results, and deformations, and check all the changes you made or apply retrofitting methods.

An integrated tool, flexible, easy and very useful that saves you a lot of time.

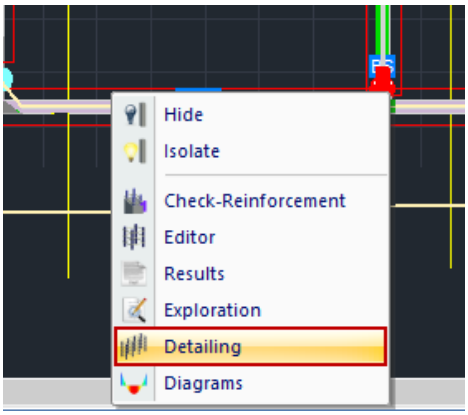
⚠ Prerequisite for accessing the "Detailing" tool is the design of the columns.

NOTE:
There are two ways of accessing the "Detailing" tool:

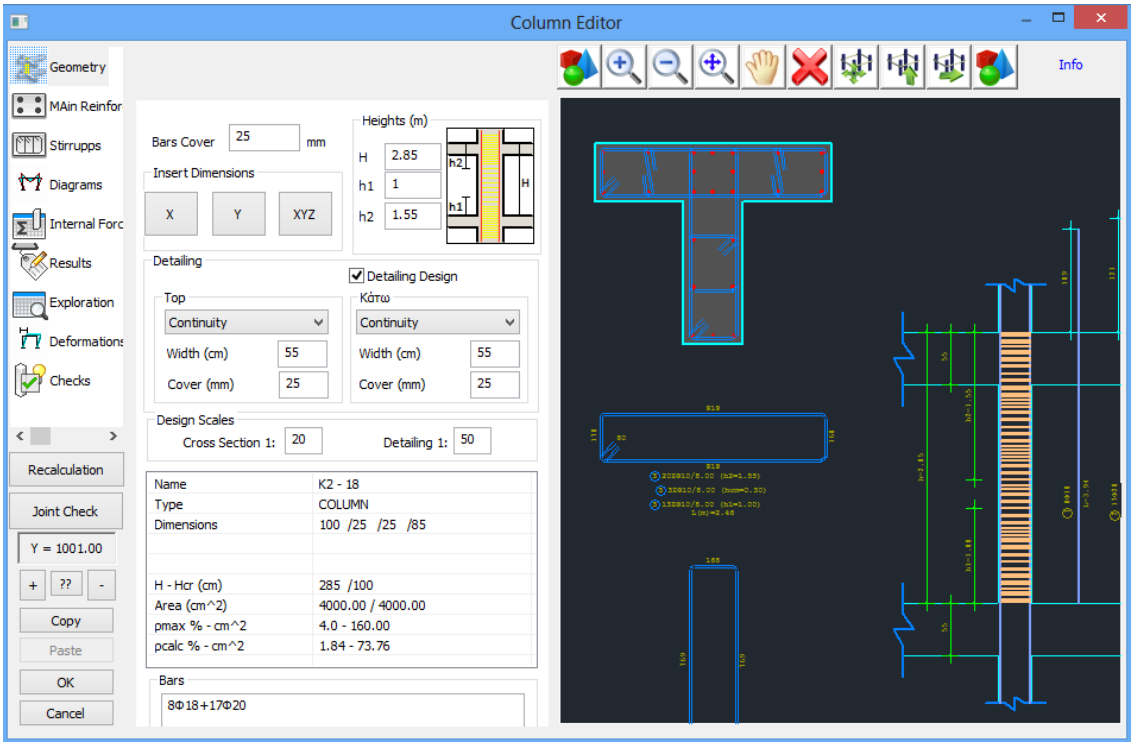
1) Open Members Design>> Columns >> Results >> Detailing.



2) In “Members Design” unit area, press right click on the column to open the command list.



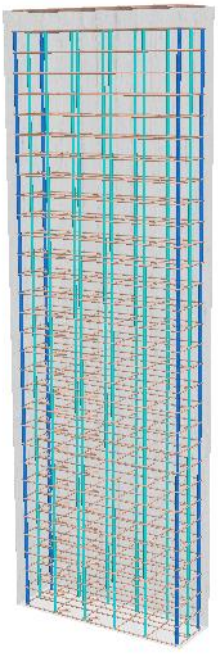
Then, select the command to open the following dialog box:



The horizontal bar above the cad interface helps to manage the drawing.

In detail:

 : This button is used for the display of the column and the steel reinforcement in the 3D view.



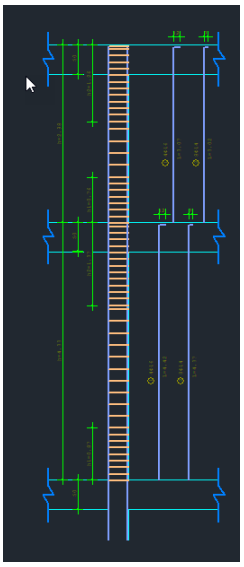
 : These buttons are used for zoom in, zoom out and zoom all respectively.

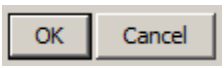
 : This button is used for moving the drawing inside the cad area (Pan).

 : This button is used for deleting objects (Delete). Select the icon and click the rebar, the stirrup or the dimension line that you want to delete.

 These buttons are arrows to move in the drawing in the corresponding directions.

 : This button is used to display column in height.



 : Press the button "OK" to save the changes you made in the columns editor and turn back in SCADA Pro's interface. Press the button "Cancel" to turn back in SCADA Pro's interface without saving the changes you made in the editor.

Next, to the horizontal bar described above, the current status is displayed:

 It's the status in which you receive information. While you move the mouse in the cad area on the right, the program updates the corresponding parameters on the left.

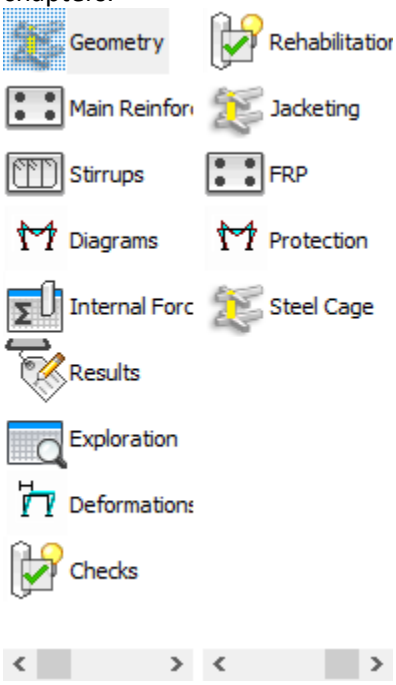
 It's the status in which you can edit/modify the data. Select the command to activate edit utility, enter the parameter and execute by clicking on the drawing.

 It's the status to which you can add a rebar.

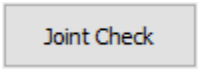
 It's the status in which you can make a deletion.

Pass from "Info", "Add", "Delete" to "Info" with right click.

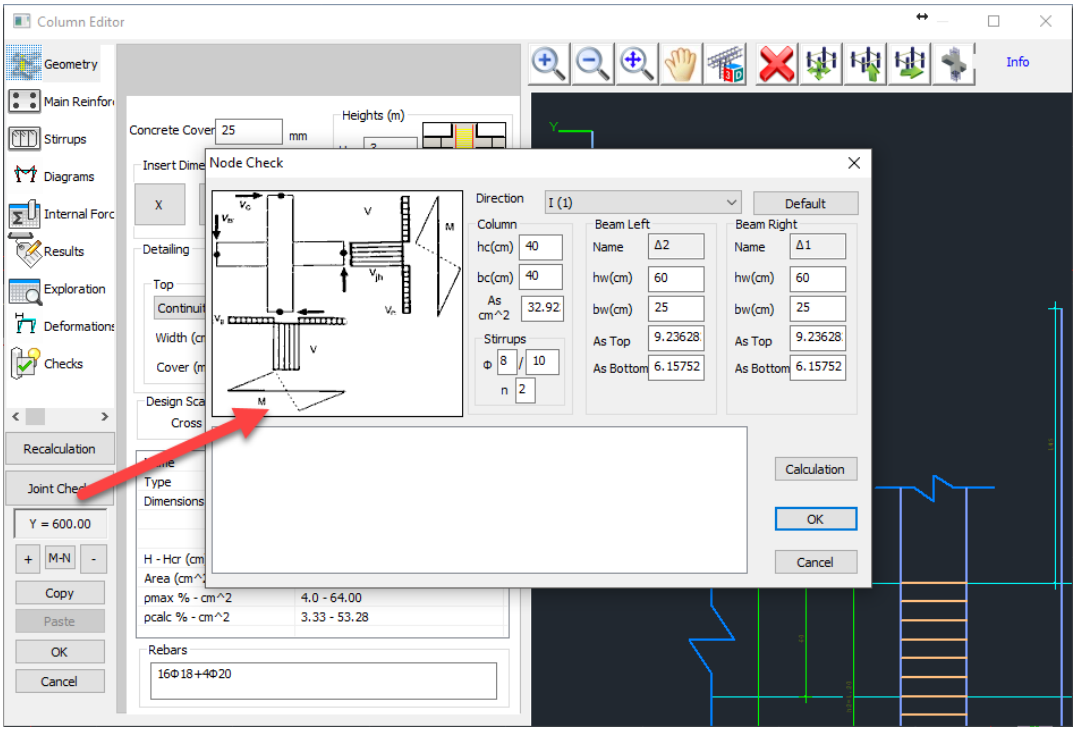
The "Column Editor" dialog box contains 14 sections, explained in detail in the subsequent chapters.

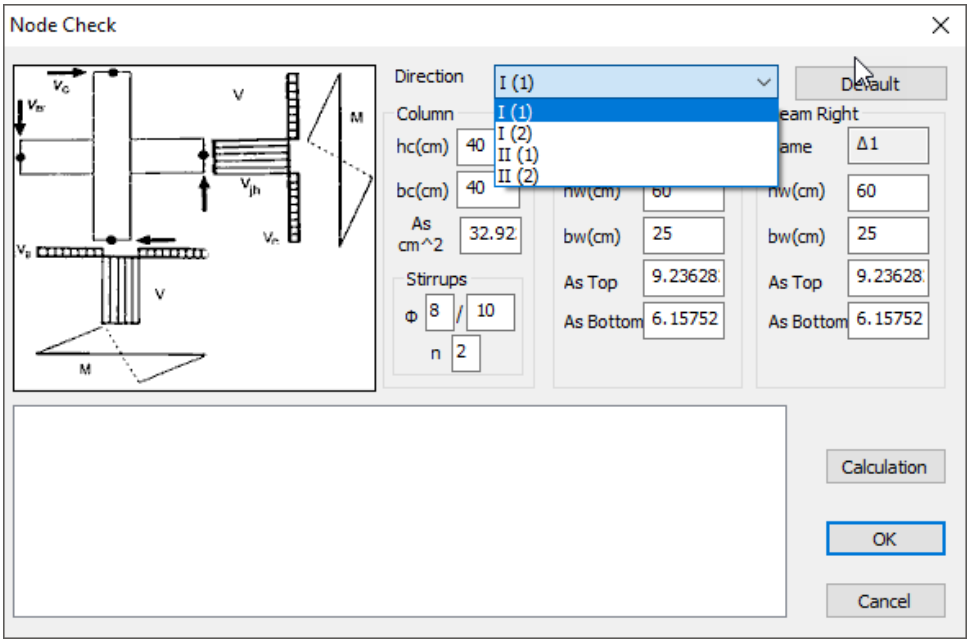


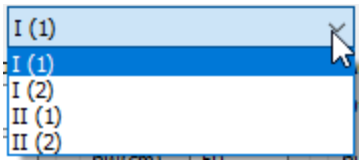
(a) Joint Check

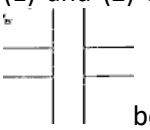


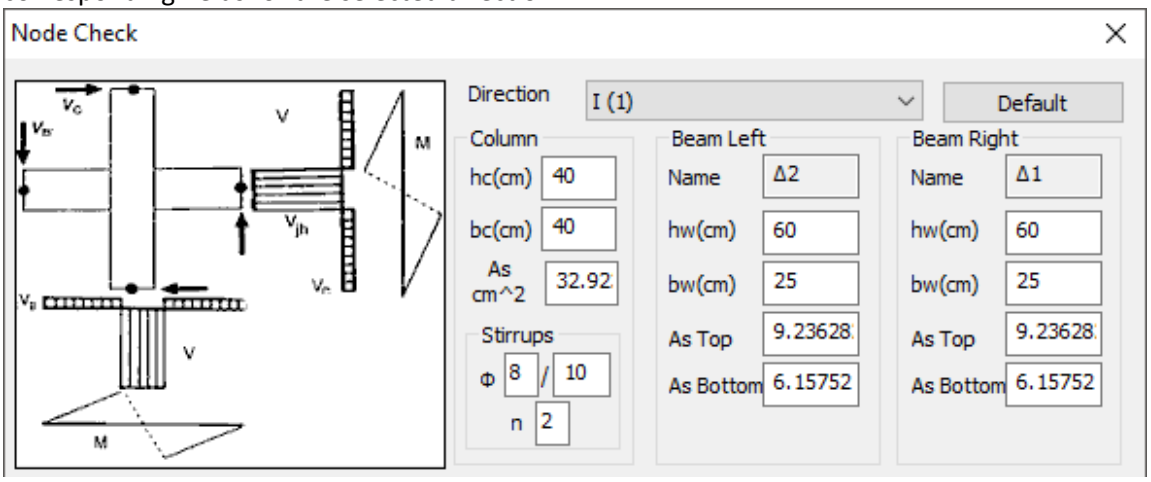
By selecting this command, you perform the check that EC8 provides for DCM ductility categories in chapters §5.5.2.3 & §5.5.3.3. Prerequisite for making the check is to activate the corresponding checkbox “Node Check” in Column Design Parameters.





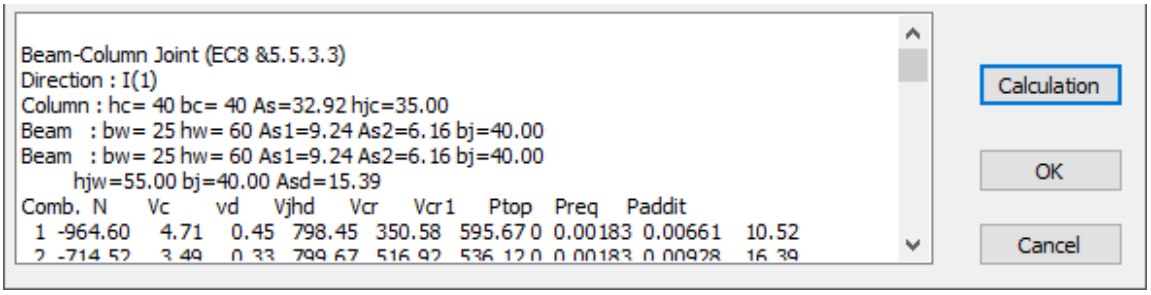
In the field Direction 

I and II are the two vertical and horizontal directions according to the local axes of the column.
(1) and (2) concern the beam members that reach the column's node. Simple cases such as  belong to (1), while in case that more members reach the node along the same direction  then we will have the (2) case.
Choose Direction and press the Default button. The program fills in automatically the corresponding fields for the selected direction.



Thus, the dimensions of the elements reaching the node are filled in. Concerning the column, the area of the vertical reinforcement of the column As for the selected direction and the corresponding horizontal stirrups are written. Concerning the beams, As the top is the top beam reinforcement, As bottom is the bottom beam reinforcement.

Press the Calculation button

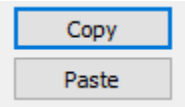


The empty field is filled in with the Node Check results for the selected direction and each combination. The summary results are recorded at the end of the Results file. In this file, the requirement to change or not the cross section of the column, as well as the possible requirement

for additional horizontal (Ash) and vertical (Asv) reinforcement and their unfavorable combination are shown.

⚠ The program characterizes automatically the node as End and Middle.

(b) COPY-PASTE Reinforcement



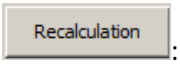
The Copy and Paste commands allow the copy of column's/wall's reinforcement (Copy) to another one (Paste), in order every change you make in a column's/wall's cross-section to be copied to the same or different level without repeating the process.

USE:
- All the changes made in the column on a defined level can be copied to other levels, avoiding in this way repeating the same process many times. Just select Copy, change the level by using the following buttons 

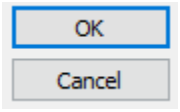
and 

, and then select Paste.
- To copy column's/wall's reinforcement to another column/wall, click on COPY, close the Details window and then open the details of the other element in the same or different level and then click on PASTE so as the reinforcement of the selected element to be copied.

(c) Recalculation



: The "Recalculation" command is used to restore the steel reinforcement of the section as it was before any modifications.



Select OK to save the changes and close the Editor window, or Cancel to get out of the window without saving it.

(d) Interaction Surface M-N Calculation

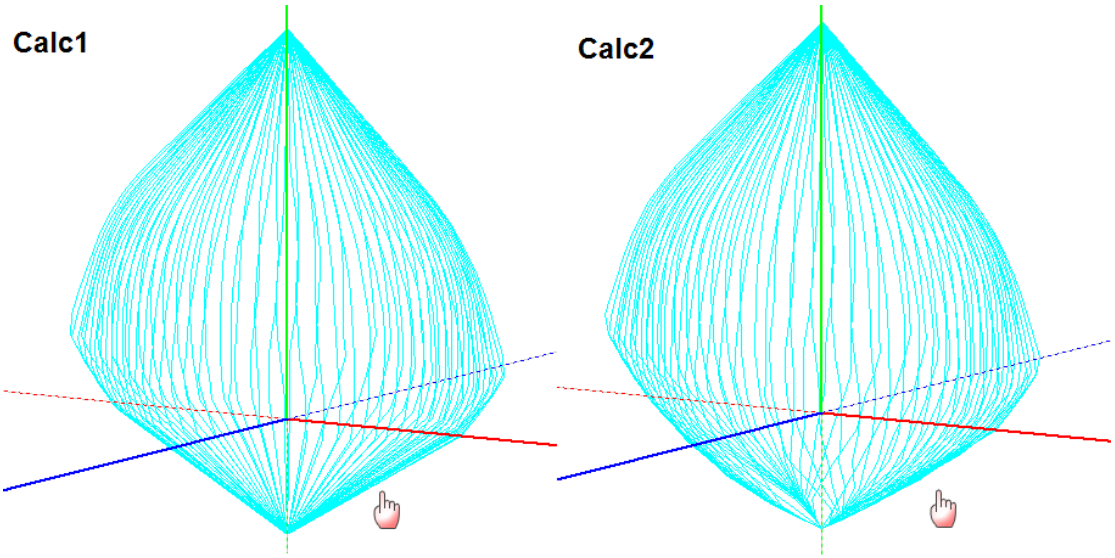
It's about the calculation and the display of the interaction moment-axial diagrams, based on the geometry of the cross section, the quality of the materials and its reinforcement. Produced the three-dimensional diagram of the envelope of strength (My, Mz, N). Furthermore, produced

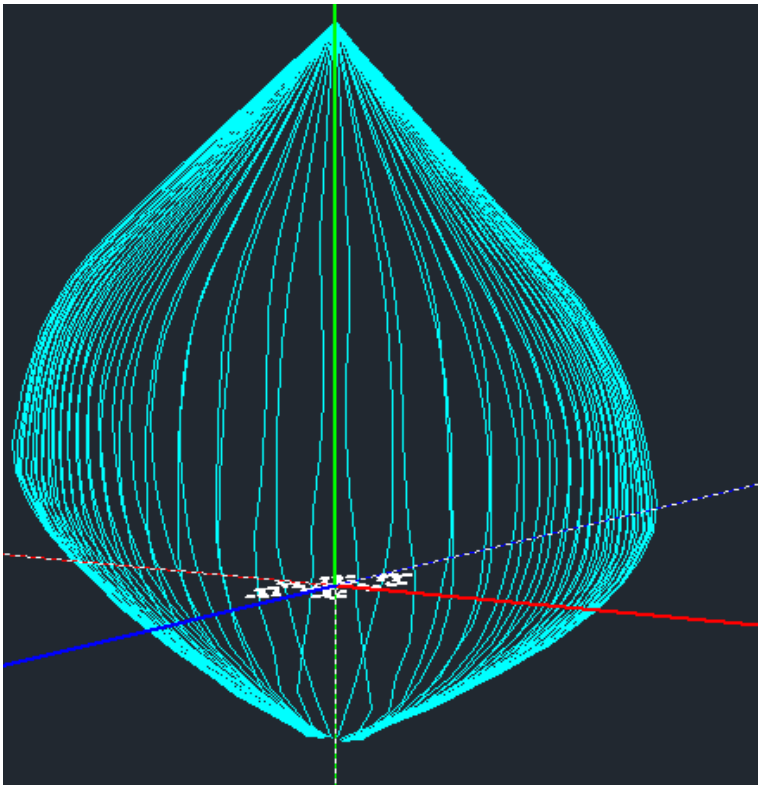
stress-strain schematic diagrams for steel and concrete, and detailed the Moment-Curvature diagram. Follows the process for the production of the diagrams and the presentation of all the necessary information you can see in this dialog.

- **DIAGRAM CALCULATION**

To generate the interaction diagram of the selected section, select either «Calc1» or "Calc2" key. The difference between the two diagrams concerns the part of the diagram with a negative axial force (N) representing the strength.

-**Calc1:** produces linear tension diagram, which means →lower tensile strengths, so → unfavorable conditions.
-**Calc2**: calculates also the intermediate tension values, resulting → the diagram acquire curved form and accurate results in tension.
The upper part of the diagram (Compression) not affected by the above selection. Both ways of calculation ("Calc1" and "Calc2") produce identical Compression diagram.

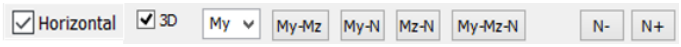




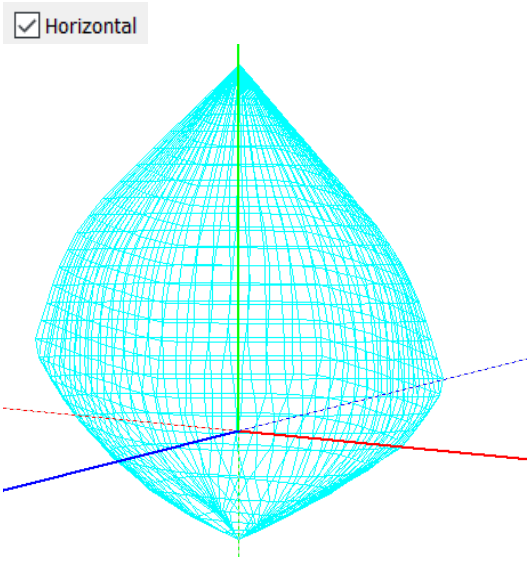
NOTE

The points in the diagram are the N-My-Mz points for each combination.

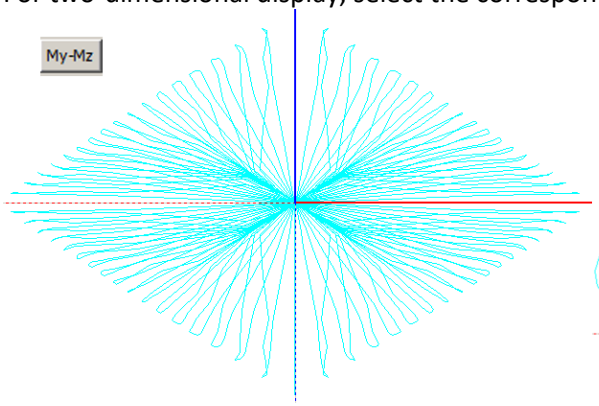
- **ILLUSTRATED**

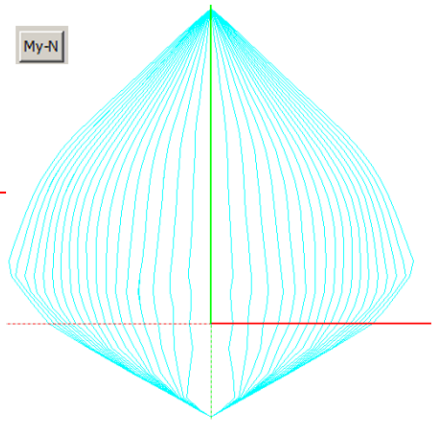


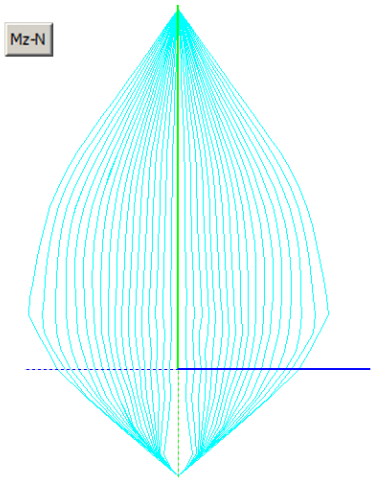
For the schematic illustration of the horizontal curves (N = constant.), activate

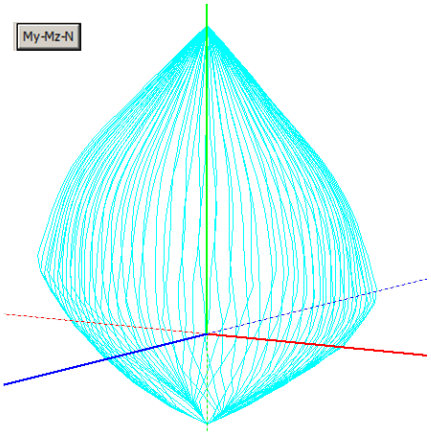


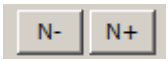
For two-dimensional display, select the corresponding buttons:



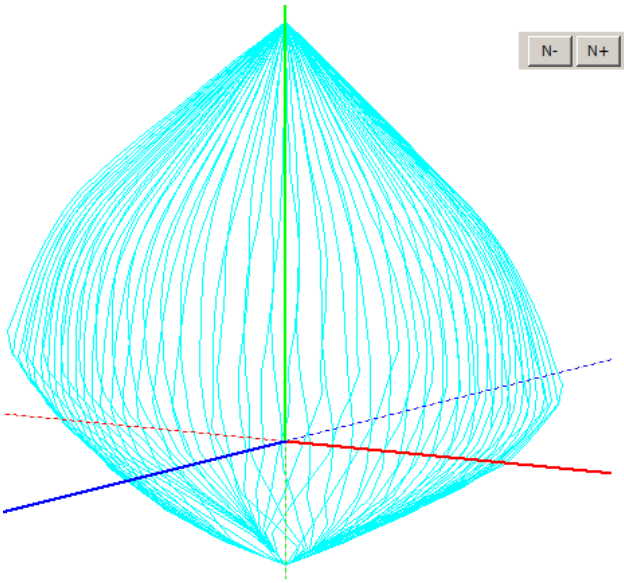


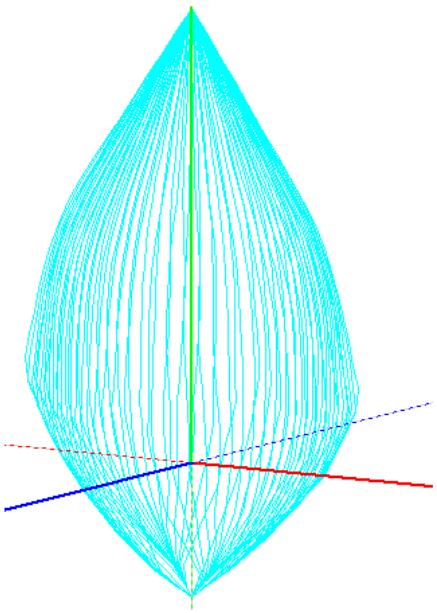




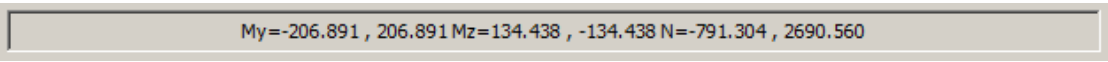


to display the diagram resulting in reducing and increasing the imaging scale of the axial force.

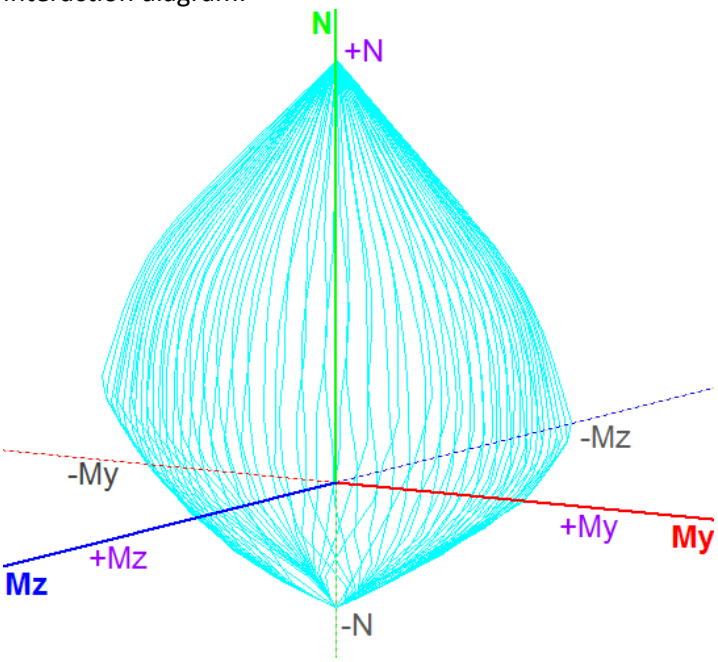




- **HORIZONTAL BAR MAXIMUM VALUES**

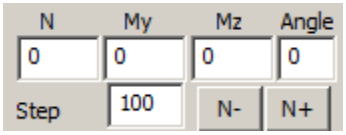


In the horizontal bar showing the six maximum values resulting from the three-dimensional interaction diagram:



These values represent the maximum for each intensive size and they are the extreme curves values.
The coordinate system of the resistance moments coincides with the column's local system, but on condition that you do not change the default, angle beta calculates from the program when the mathematical model is created.
The dashed line represents the axes negative values.

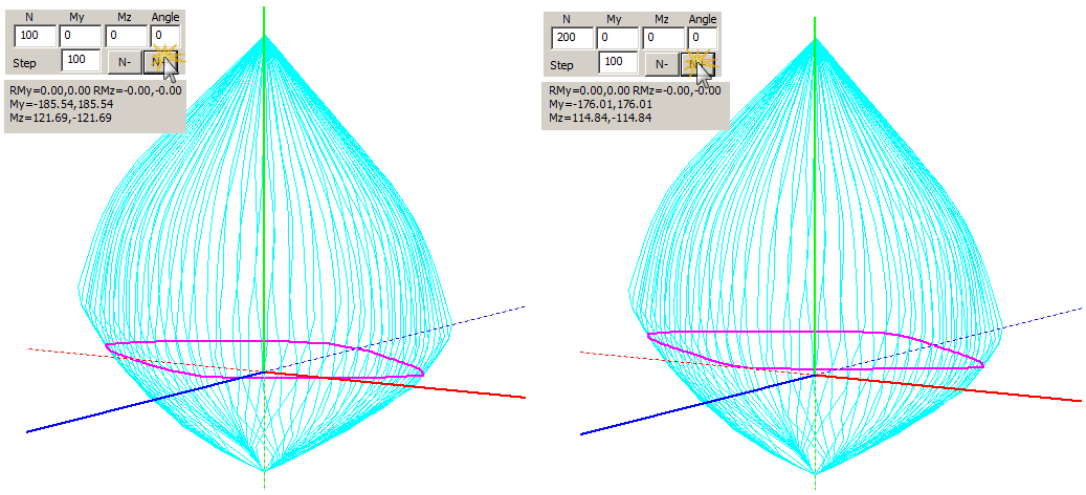
- **SEARCH POINTS ON THE DIAGRAM**



This field can be used in different ways:

1. To display the horizontal curves of the diagram

By typing in the Step field only a value and clicking the 

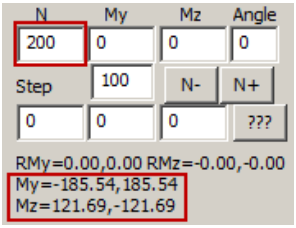


each "click" forms the horizontal curve represents the values of the resistance moments, for a specific value of axial force and different angle values of the neutral axis.

Field "Step" represents the increase or reduction step to form the horizontal curves.

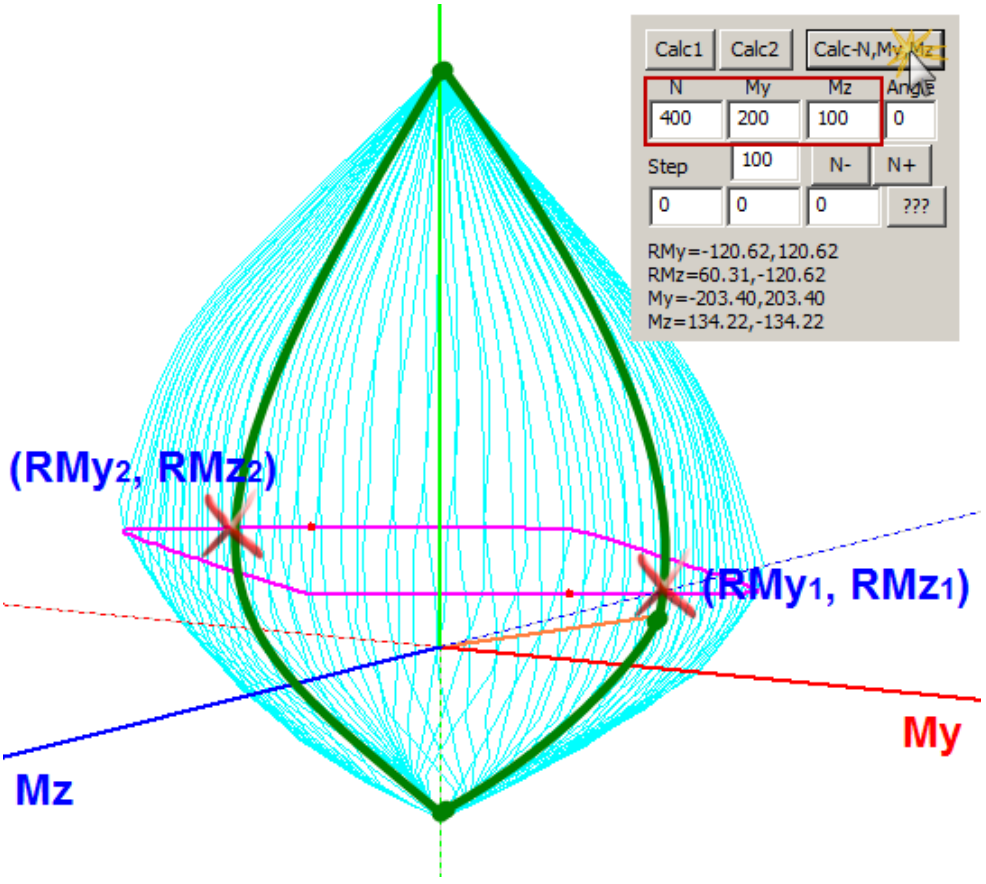
- Choosing N + curves designed upward
- Choosing N + curves designed downward

Furthermore, for each horizontal curve showing the relative maximum positive and negative values My and Mz of the diagram representing the maximum positive and negative moment resistance for the specific N.



1. For the calculation of the resistance moments for specific intensive values N-My-Mz

Typing the values of the internal forces N, My, Mz in the appropriate fields and clicking/, the program:



- finds the point (N, My, Mz) in the diagram
- designs the line segment joining (0,0,0) and (N, My, Mz) (orange line)
- designs the curve N * and calculates the corresponding My, max and Mz, max

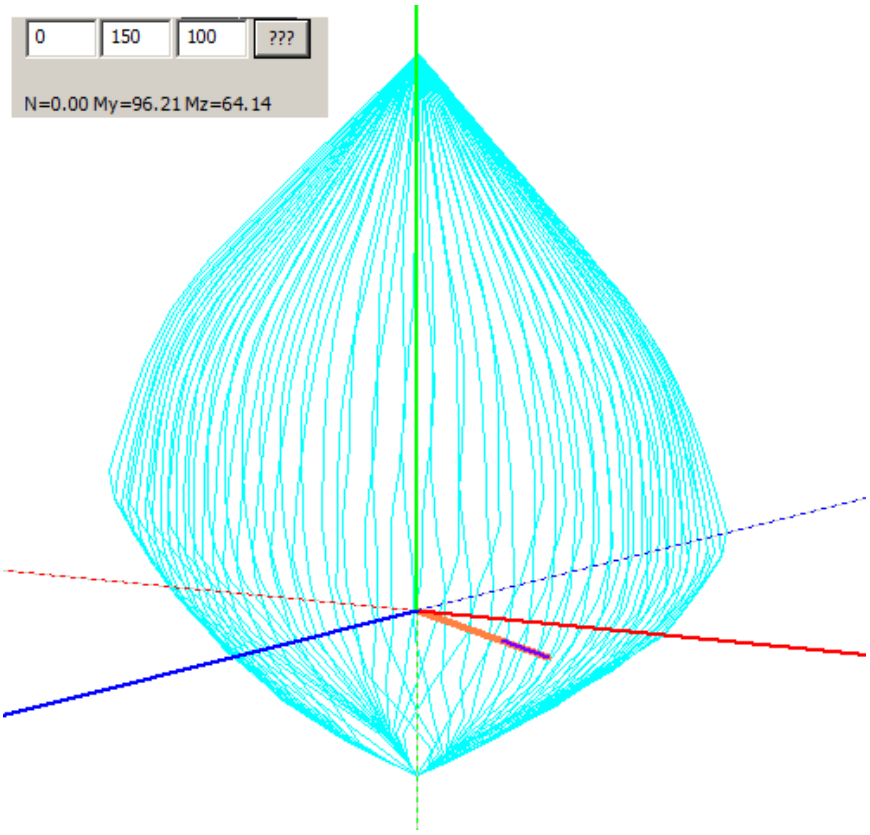
My=-203.40,203.40
Mz=134.22,-134.22

- calculates the flexural strength (RMy, RMz) RMy=-120.62,120.62 RMz=60.31,-120.62 for the specific intensive forces (N, My, Mz). They are the red dots on the horizontal curve.
- forms the "moment-curvature diagram"

⚠ The moment-curvature diagram is defined for a particular angle of the neutral axis.

⚠ Defining in the Angle 30 field an angle value different than 0,90,180,270 degrees, in the diagram will appear also, the curve of the negative moments.

EXTENSION POINTS ON THE DIAGRAM

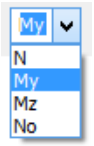


Typing internal forces values in the respective fields and clicking/, the program:

- finds the point with the specific coordinates,
- draws a line segment that connects it with the origin (orange part) and
- extends until it meets the envelop (blue part), indicating the respective values of strength N, My and Mz (values useful for Pushover). N=193.52 My=96.76 Mz=58.05

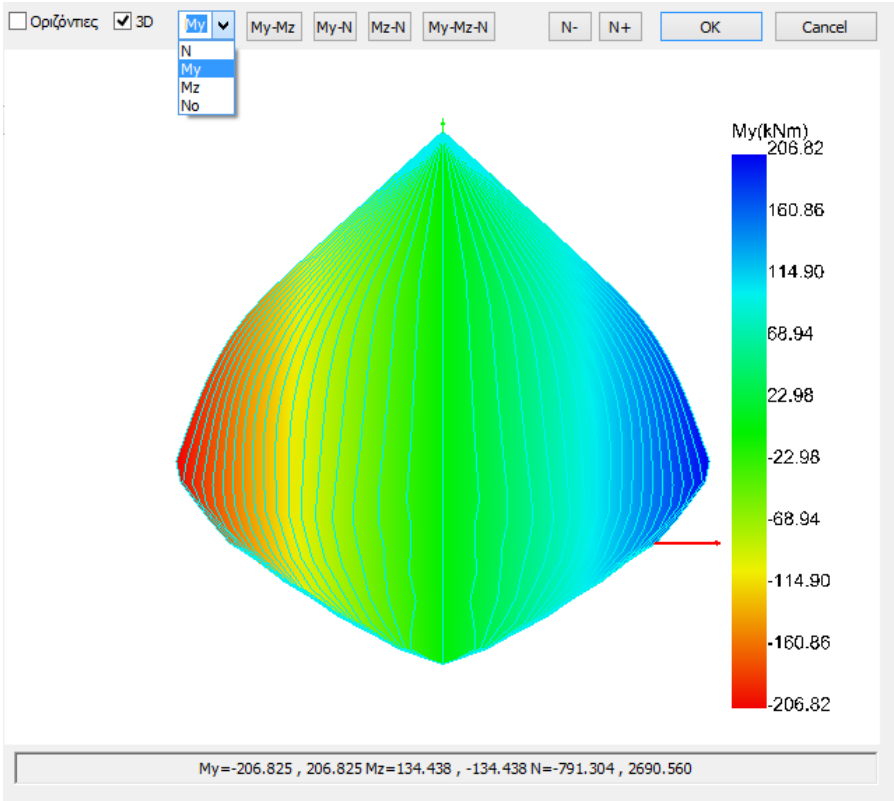
THREE-DIMENSIONAL REPRESENTATION OF THE INTERACTION DIAGRAM

Activate the checkbox 

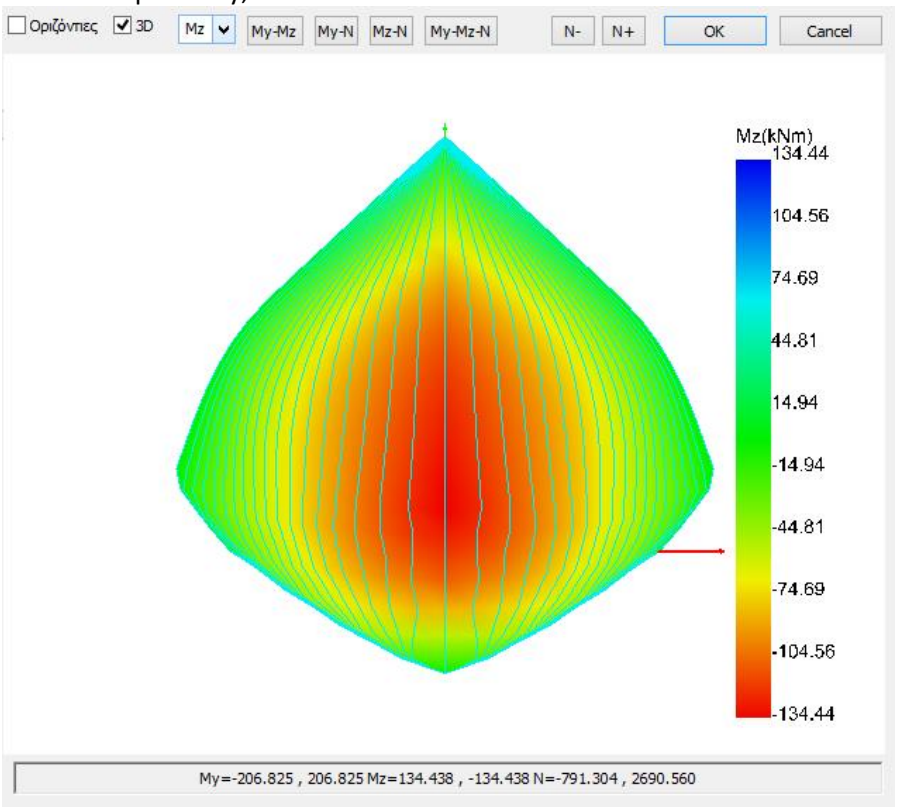
and select an intensive size 

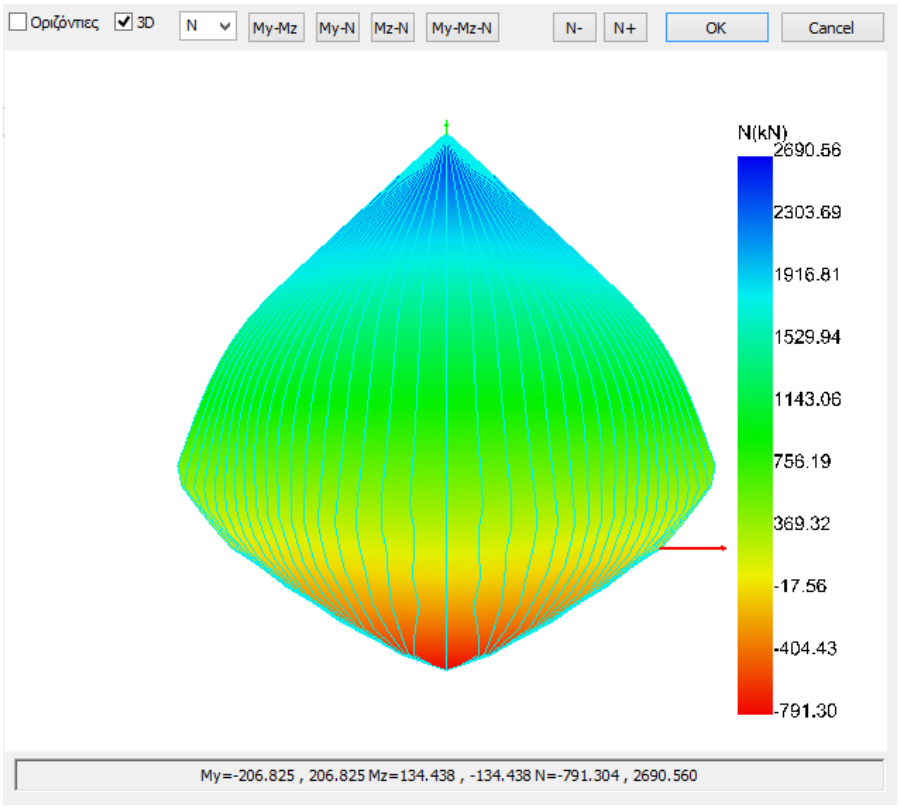
for color representation.

- By selecting the My the diagram is colored in the y axis.
- The color grading provides the width of the sections, according to the bar at the right.
- The horizontal bar at the bottom lists the maximum and minimum values and the three internal forces.

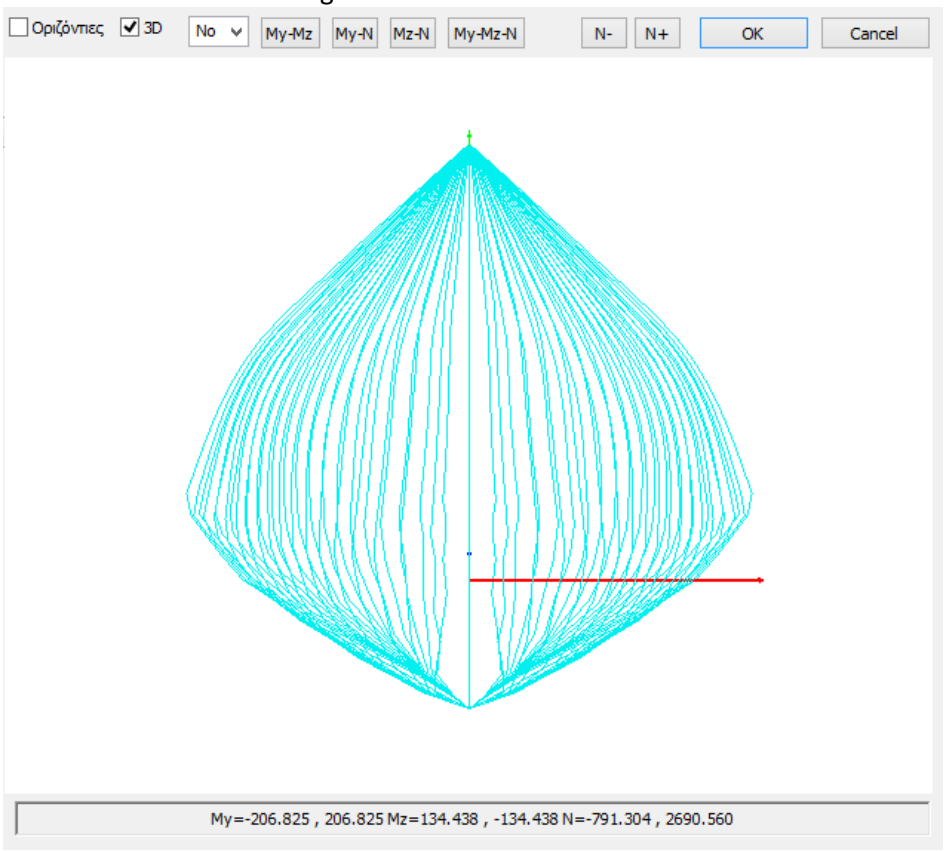


- respectively, for M and N.

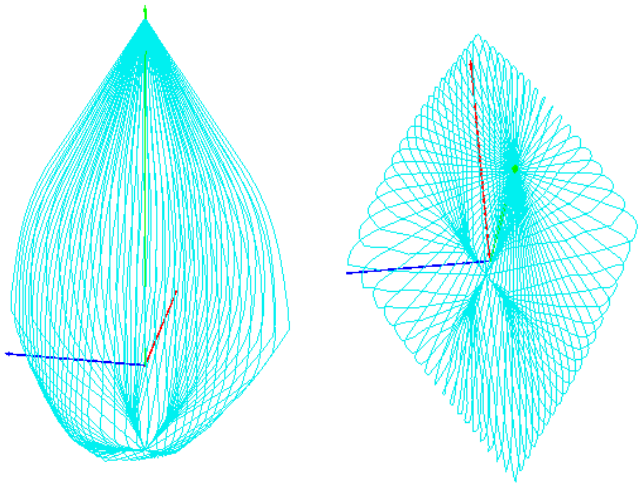




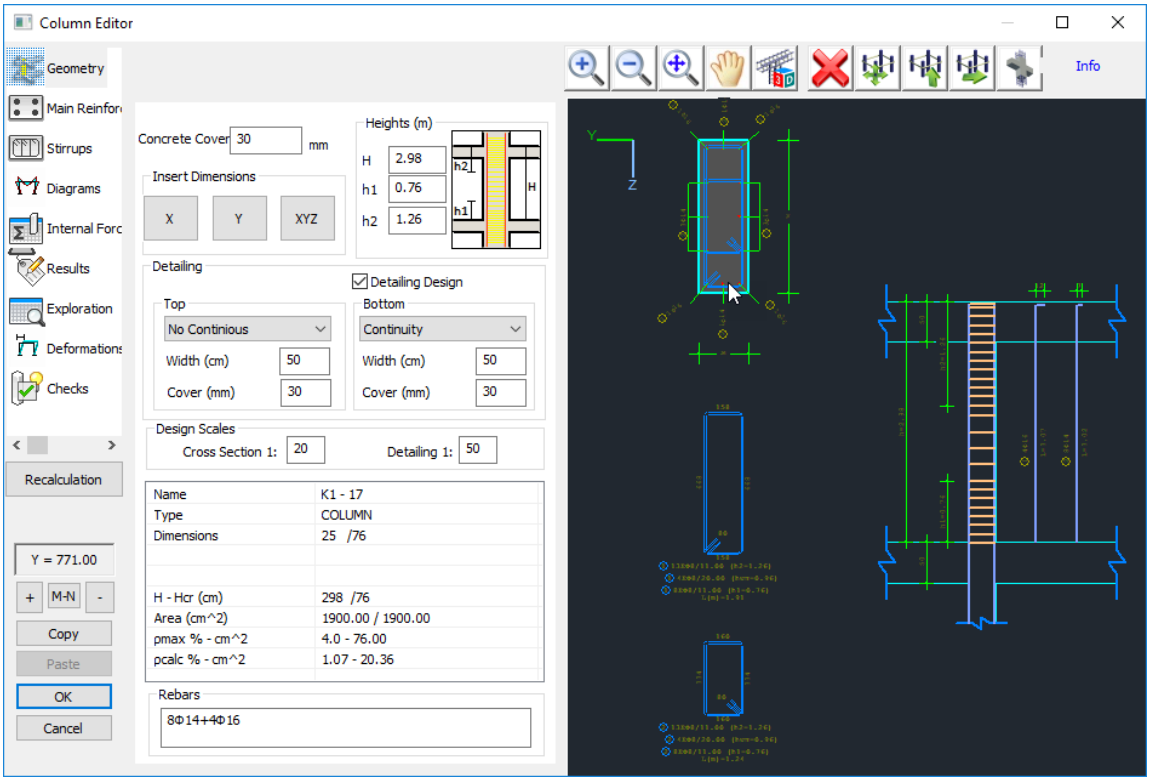
⚠ Activating “No”, the interaction diagram shows a more accurate three-dimensional display without color rendering



- By press and hold the left mouse button and moving it, you can rotate the diagram.

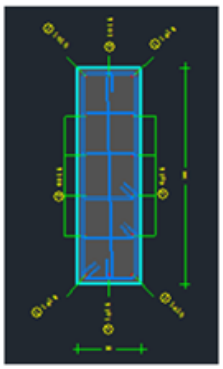


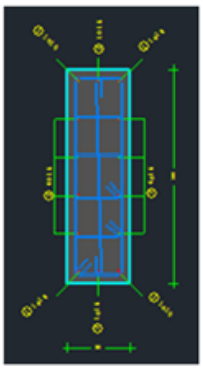
1. Geometry



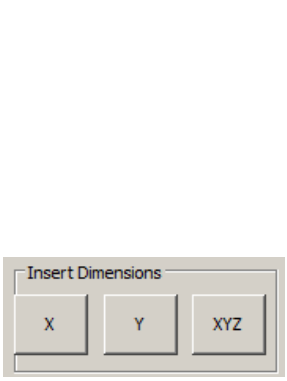
Geometry: This section includes a group of design parameters in the center, and a cad interface on the right, that adapts to the parameters changes.

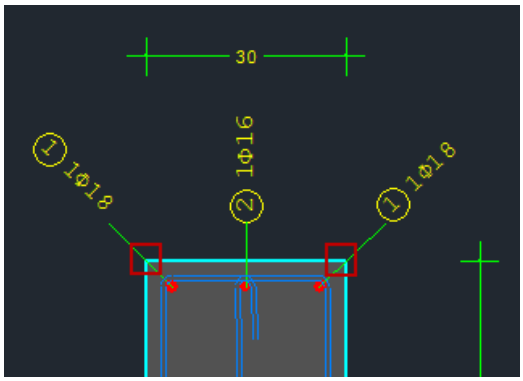
Change the value of the cover in the "Bar Cover" field, and the corresponding drawing will be automatically updated.





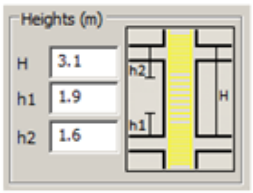
In the "Insert Dimensions" field, select the direction X and show with a left click, in the cad interface the start and end points of the dimension line, and the point to place the dimension.

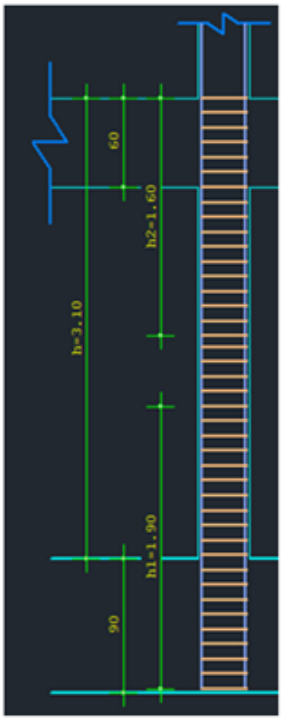


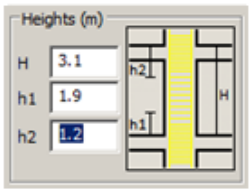


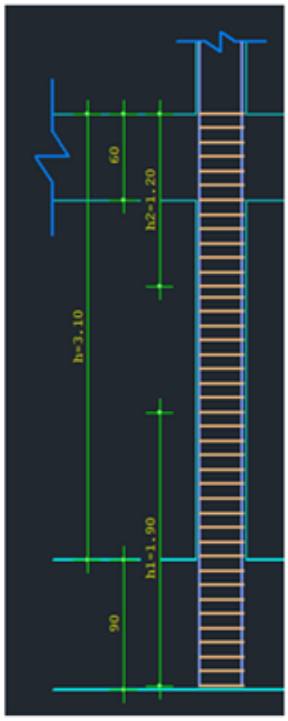
Follow the same procedure for the other two directions.

In the "Heights" field, you can modify the total height of the column and the critical length on the top and the bottom of the column, near the joints. Change the corresponding values and the drawing will be updated automatically.

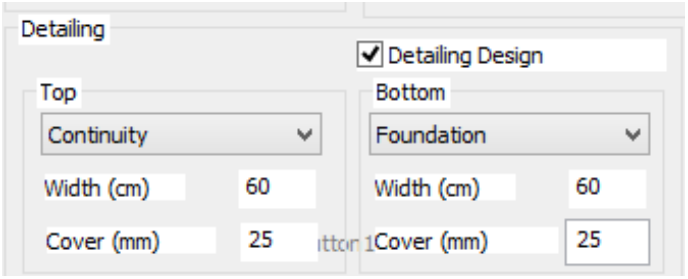




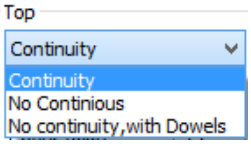




In the "Detailing" field, if you don't want the detailing design to be displayed in the cad interface, you can deactivate the "Detailing Design" checkbox.

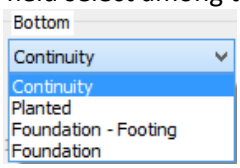


In the "Top" field select among the three options of the following drop-down list:

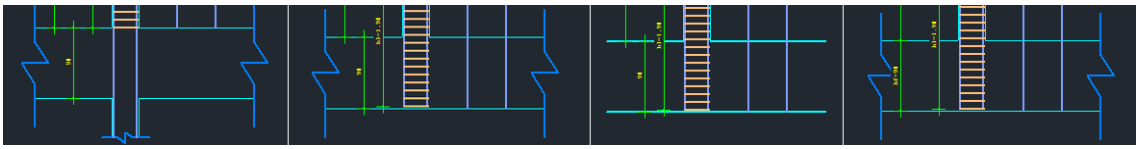


Then, the drawing is updated by the selected option.

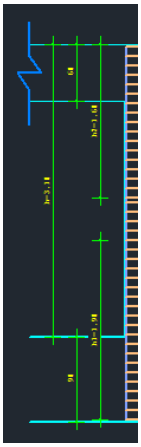
In the "Bottom" field select among the four options of the following drop-down list:



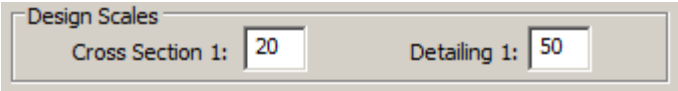
Then, the drawing is updated by the selected option:



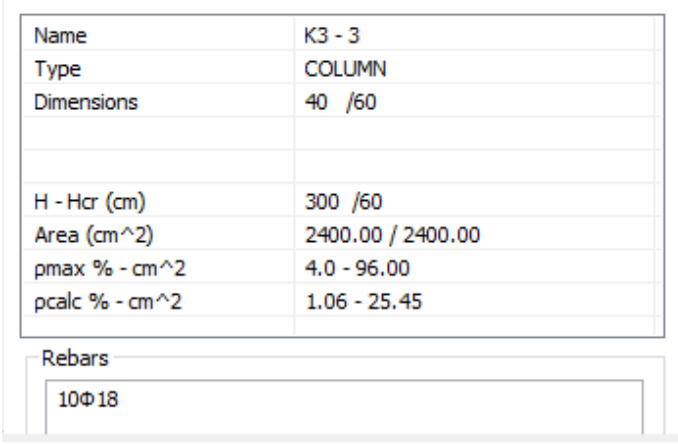
Height and Cover are related to the connected with the column members, up and down. Change the values and the program updates the drawing automatically.



In the "Design Scales" field defined in the corresponding fields the scale of the drawing of the cross-section and the detailing:



On the bottom of the dialog box, there is a table with non-editable general data regarding the column.

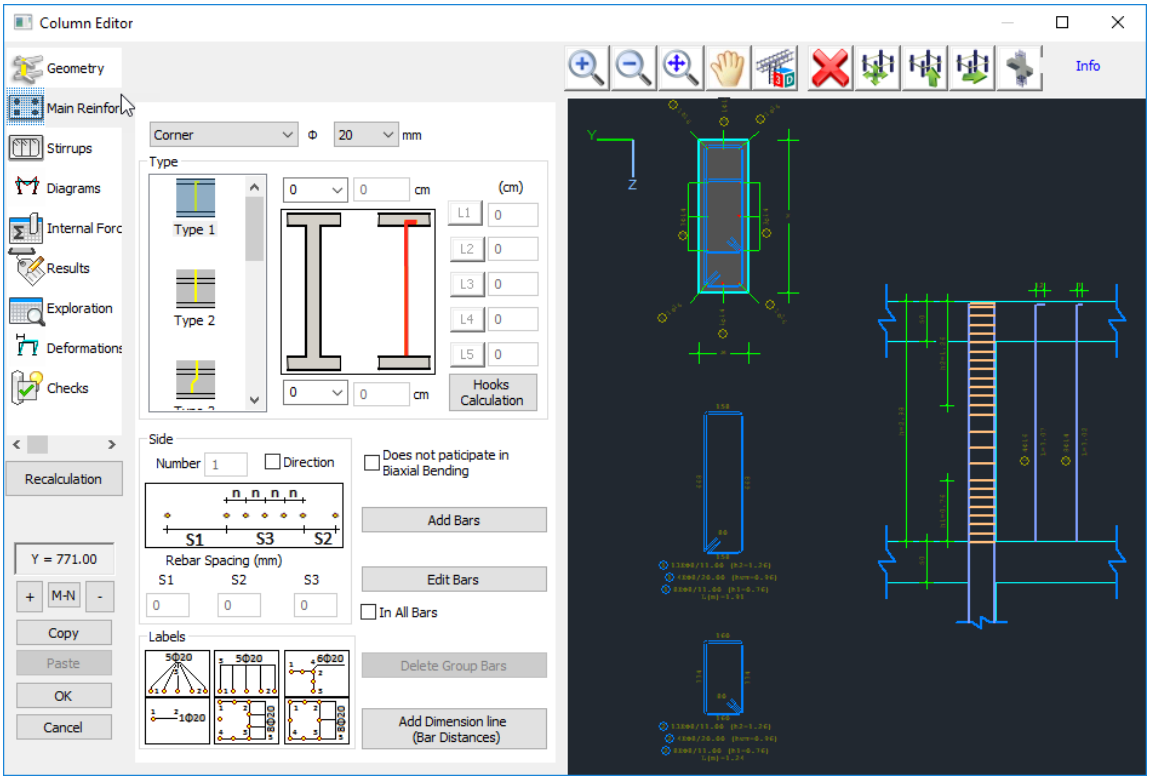


Name	K3 - 3
Type	COLUMN
Dimensions	40 /60
H - Hcr (cm)	300 /60
Area (cm^2)	2400.00 / 2400.00
pmax % - cm^2	4.0 - 96.00
pcalc % - cm^2	1.06 - 25.45

Rebars

10Φ18

2. Main Reinforcement



In the "Main Reinforcement" section you can modify the main steel reinforcement of the column. Main reinforcement consists of two categories of bars concerning their position inside cross-section; corner and side bars. Moving the mouse near the bar in the drawing, the Info state is activated so that you can read the characteristics (category, type).
The procedure to modify is first to select the command, then to show the rebar and then follow the editing process described below.

2.1 How to modify the main reinforcing bars:

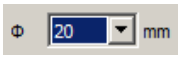
2.1.1 To edit the diameter and the type of corner bars:

. Select the command "Edit Bars" Edit Bars.

. Left click on a corner bar inside the drawing of the column cross-section.

. The status "Edit" is activated Edit.

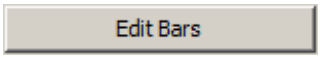
.The following fields are updated automatically 

with the parameters of the selected rebar. Select a new diameter from the drop-down list 

and the new type*.

. Left click on the rebar to apply the modification.

2.1.2 To edit the number, the diameter and the type of side bars:

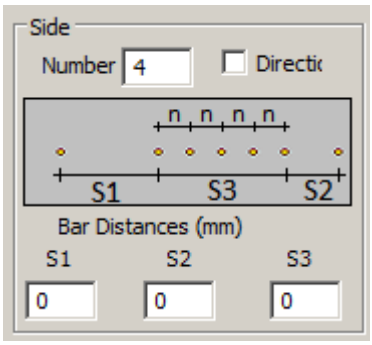
. Select the command “Edit Bars” 

. Left click on a side bar inside the drawing of the column cross-section

. The status “Edit” is activated 

.

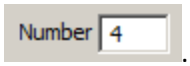
. The following fields are updated automatically 

and the “Side” field is activated 

.

. Select the new diameter 

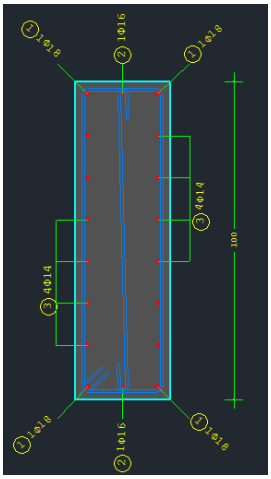
and the new type* (see page 16).

. In the “Side” filed, first type the number of the side bars 

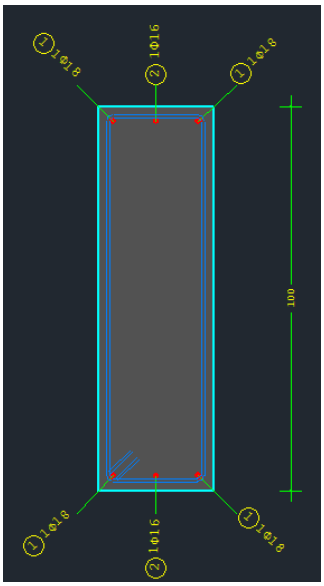
.

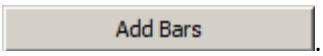
. Define the individual distances according to the shape or leave it as it is, and it will be automatically distributed at equal distances.

. Left click on one side of the detailing drawing. Follow the same steps for the other side.



2.1.3 To add side bars in cross-section detailing with no bars:



. Select the command “Add Bars” 

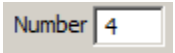
.

. The “Add” status is activated 

.

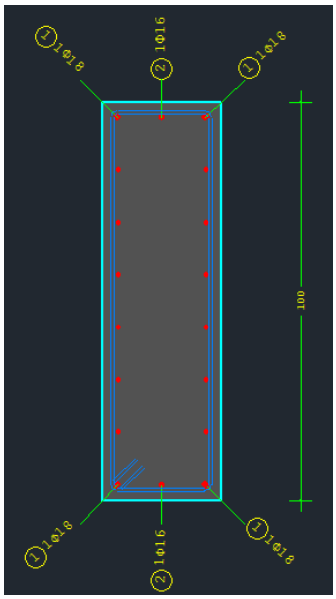
. Select from the corresponding drop-down lists the category “side”, the rebar diameter 

and the type* (see page 16).

. Type the number of the side bars 

and the spacing between them.

. Left click to show a corner bar and then the opposite one on the same side.



. Then repeat the previous step on the other side of the cross-section.

. Right click to activate the "Info" status 

.

In cases where the two opposite corner bars have different diameters and you want to insert side bars between them, active the checkbox 

and follow the procedure of adding side bars (see page 14).

2.1.4 To delete rebars:

. From the horizontal bar, select the following button  .

. Activate the status “Delete” 

.
. Left click on the rebars of the cross-section detailing to delete them.

. Right click to activate the status “Info” 

.

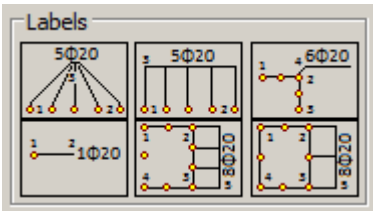
2.1.5 To insert the dimension lines:

. Select the command “Add Dimension Lines” 

.

. The “Add” status 

.

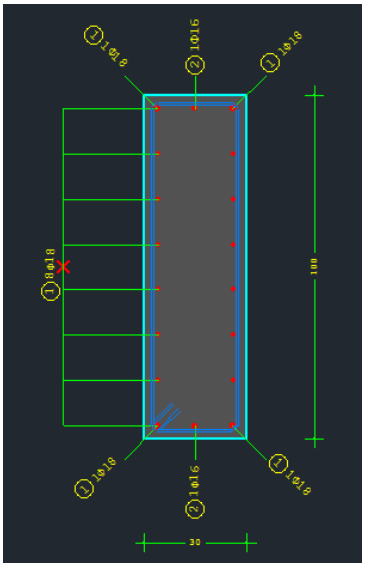


. Select the type of the Label

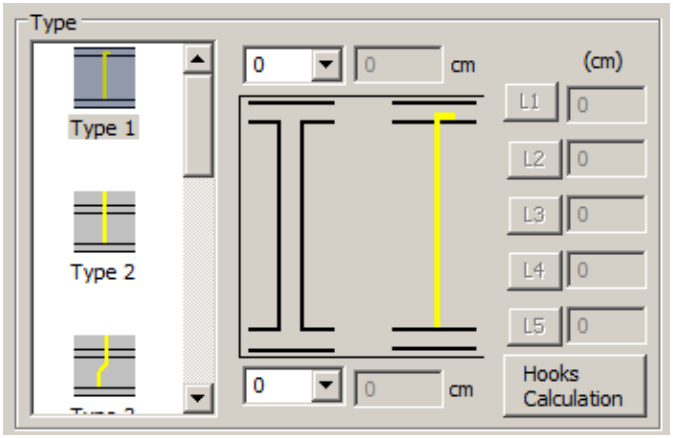
. Left click to show start and end bars

. Right click to activate the "Info" status 

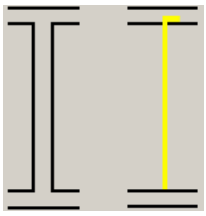
.



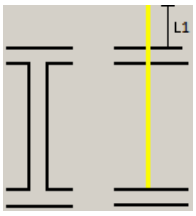
*You may as well modify the type of the rebars.



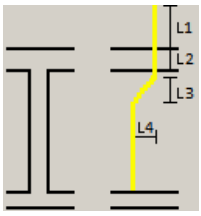
Type of rebars



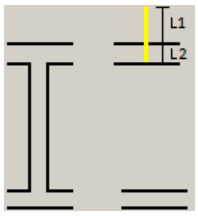
Type 1



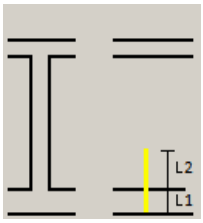
Type 2



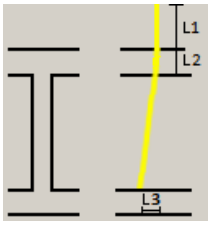
Type 3



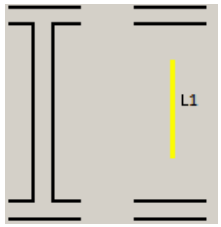
Type 4



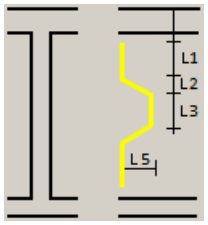
Type 5



Type 6



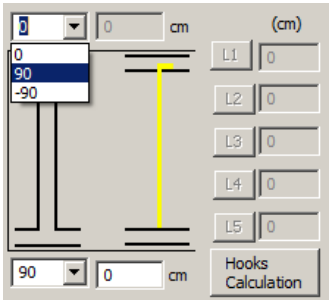
Type 7

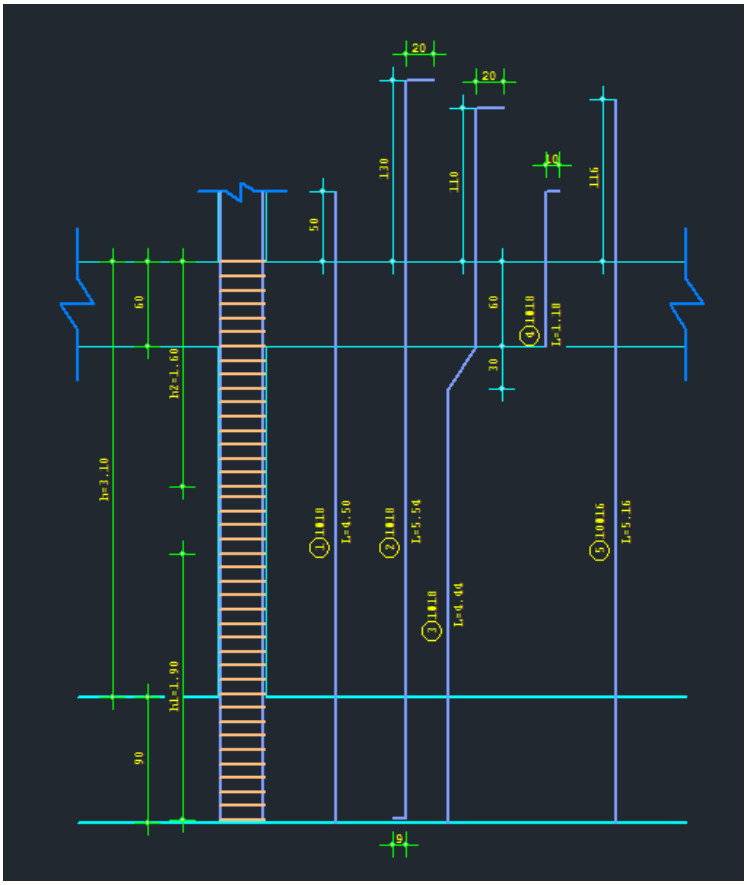


Type 8

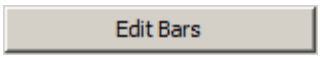
In the “Type” list select one of the types presented above. Each type activates the corresponding fields on the right, where you enter the corresponding values in cm.

In all types, you can insert hooks, up and down, the bar. Select the direction –90, +90 and press the button “Hooks Calculation” to automatically calculate the corresponding values or type your values.



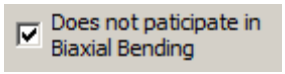


2.1.6 To exclude a rebar from the Biaxial Bending resistance check:

.. select the command “Edit Bars” 

. left click on the rebar inside the detailing of the column cross-section

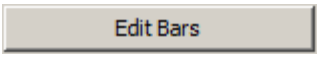
. activate the “Edit” status 

. activate the following checkbox 

. right click to activate the status 

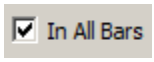
.

2.1.7 To apply the changes, that you make, to all the same rebars:

. Select the command “Edit Rebars” 

. Left click on the rebar inside the detailing of the column cross-section

. Activate the status “Edit” 

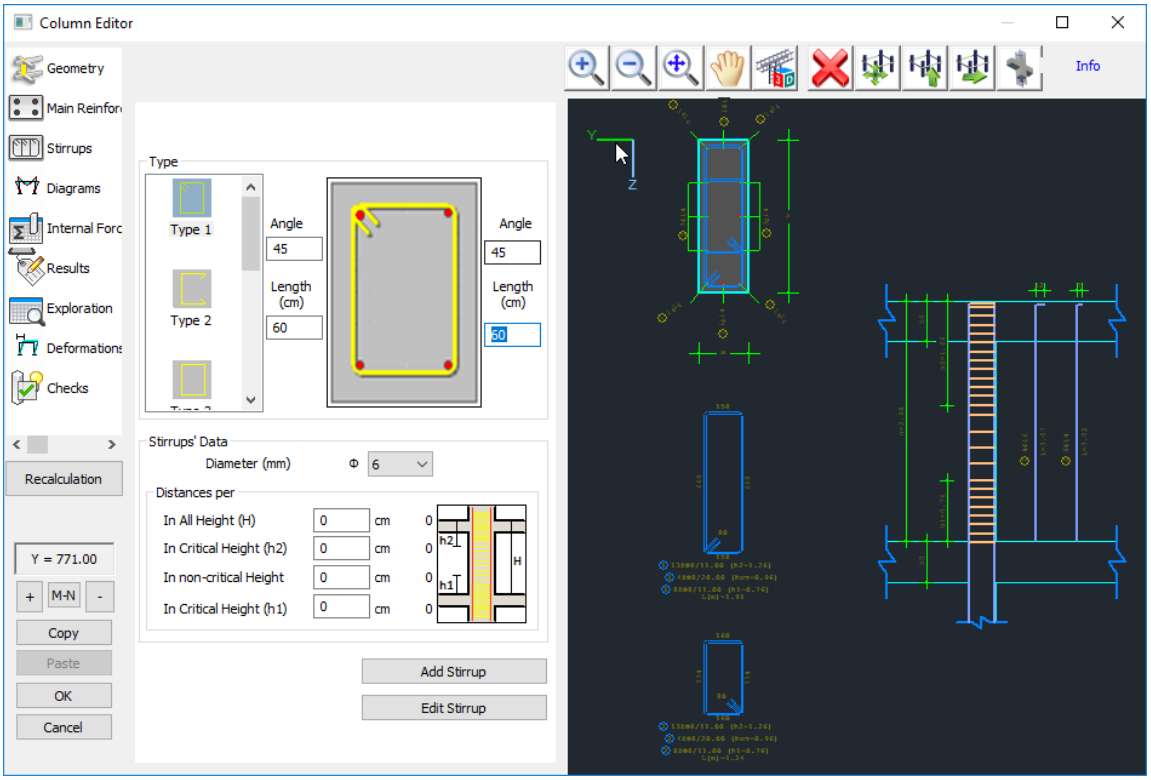
. Activate the following checkbox 

. Make the changes and they will be applied to all rebars of the same diameter.

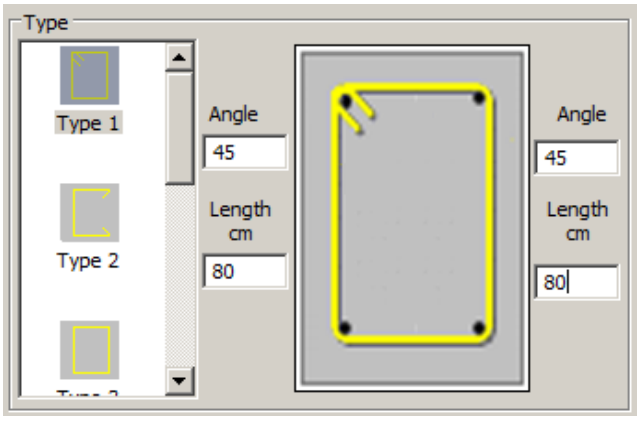
. Right click to activate the status “Info” 

.

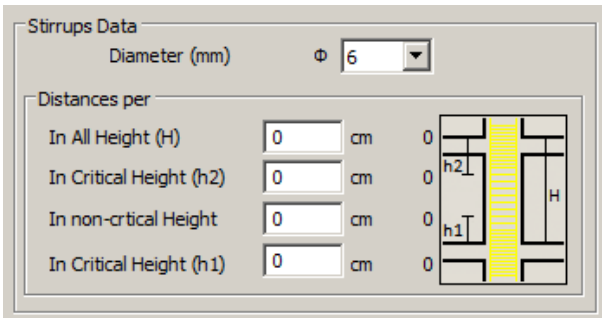
3. Stirrups



In "Stirrups" you can make modifications and interventions on the stirrups of the column. The procedure is similar to the one applied for the main reinforcement. Select the command, show the stirrup and change the type, diameter or individual spacing.



In the "Type" list you can change the type of the stirrups. For stirrups of Type 1 and 2, you may as well define the corresponding length and angle.



"Stirrups Data" field, contains the diameter of the stirrup and the spacing. You can select a diameter from the drop-down list "Φ" and fill in the corresponding field with the spacing value as appropriate.

Moving the mouse near the stirrup in the cad interface, with the status "Info" activated Info, the stirrup is colored red and the values of the diameter and the spacing are updated automatically.

3.1 Stirrups editing:

3.1.1 To modify a stirrup:

. Select the command "Edit Stirrup" Edit Stirrup

. Activate the status "Edit" Edit.

. Select the stirrup
. Select the new diameter, the new spacing, the new type.
. Right click.

3.1.2 To add a new stirrup:

. Select the command "Add Stirrup" Add Stirrup

. Activate the status "Add" Add.
. Select the diameter, the spacing, and the type
. Left click to show the rebars enclosed by the new stirrup
The design is updated automatically and a new stirrup detailing is created with all the related data.

3.1.3 To delete a stirrup:

. From the horizontal bar, select the following button 

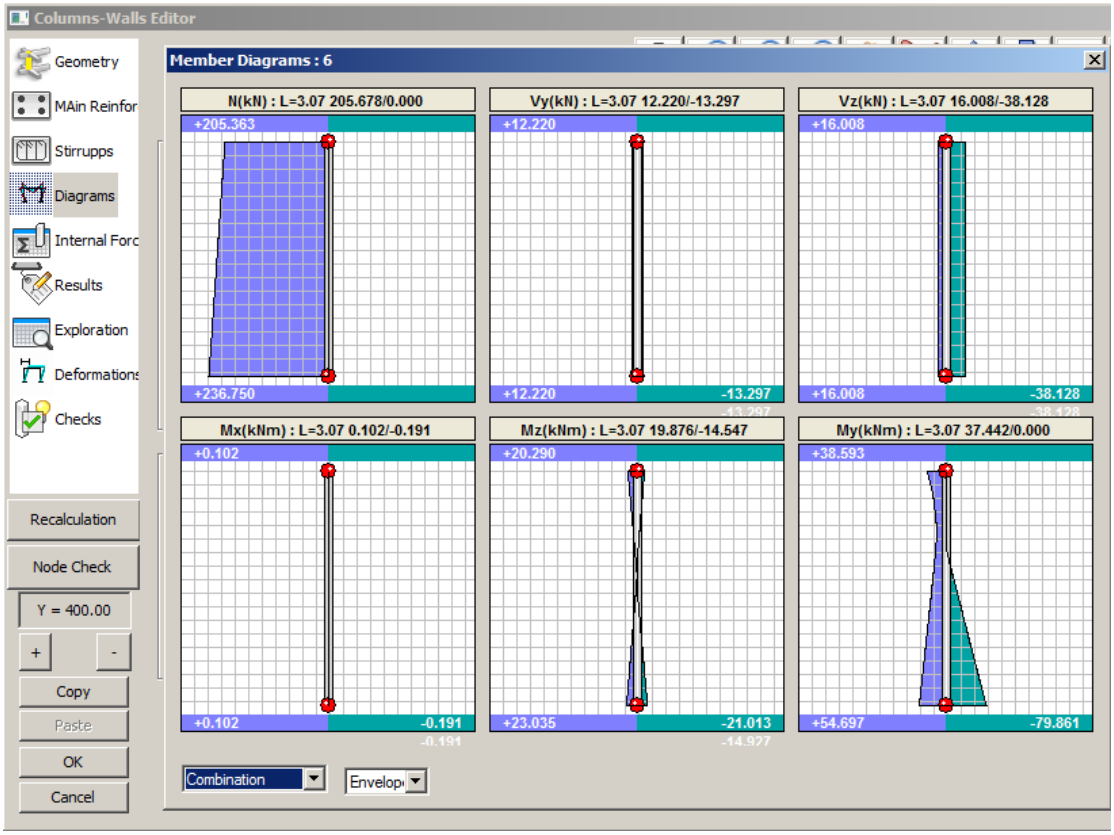
. Activate the "Delete" status Delete.
. Left click on the stirrup of the cross section detailing to delete it

. Right click to activate the status "Info" Info.

NOTES:

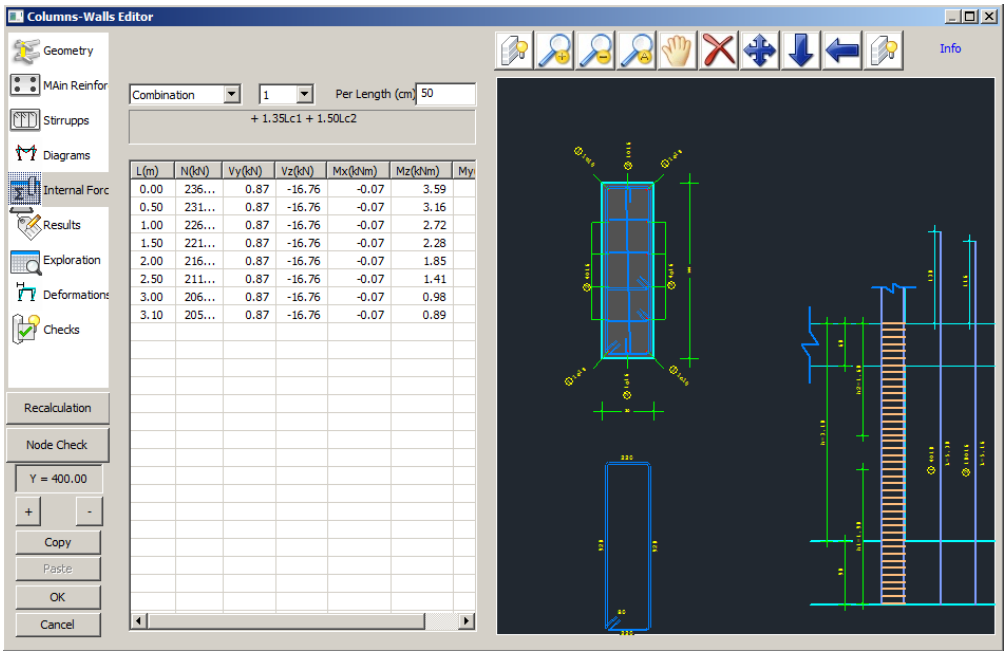
- In the design of the columns, the program takes into consideration that the calculated stirrups have the same diameter and a common spacing. If you modify the stirrups, it is proposed that the modification be uniform. In other cases, the program identifies the less favorable stirrup per area and place the rest stirrups uniformly concerning the less favorable.
- However you can put different stirrups per region, e.g. in a Γ-shaped cross-section, you can change the diameter or the spacing in “Checks” and perform the verification checks (see §2.9). Then go back to the field “Stirrups” and make the changes so that the drawing and the calculations’ print-out will be updated.

4. Diagrams

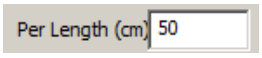


In the "Diagrams" field (when the previous scenario of member design is active) a window opens that contains the diagrams of the internal forces, for each load case and each load combination. Moving the mouse along the column, in the diagrams, you can read the values of internal forces along the height of the column. (You can find this command in the "Results" as well, explained in the corresponding chapter of the user's manual).

5. Internal Forces

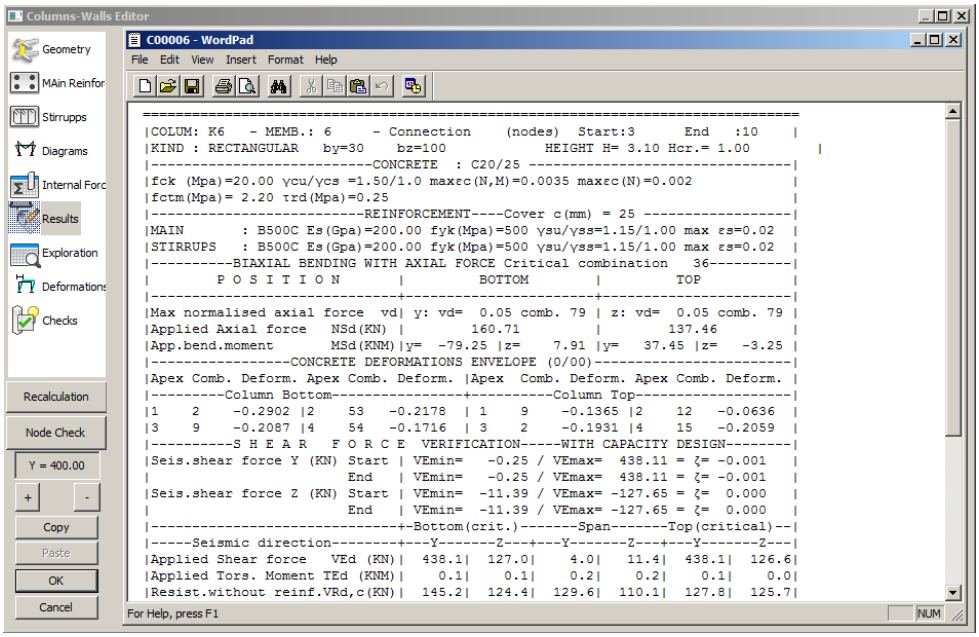


In the "Internal Forces" you can read the values of all the internal forces for each load case and load combination.

In the "Per length (cm)" field 

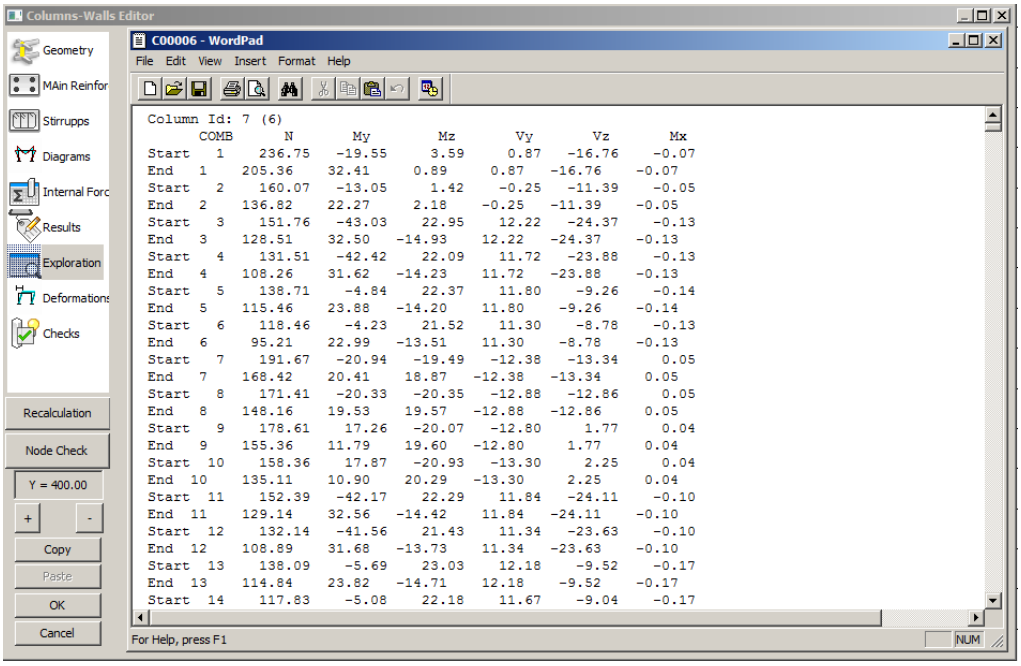
you define the position of the cross-sections of the column, concerning the column's height, where the values of the inertial forces will be calculated.

6. Results



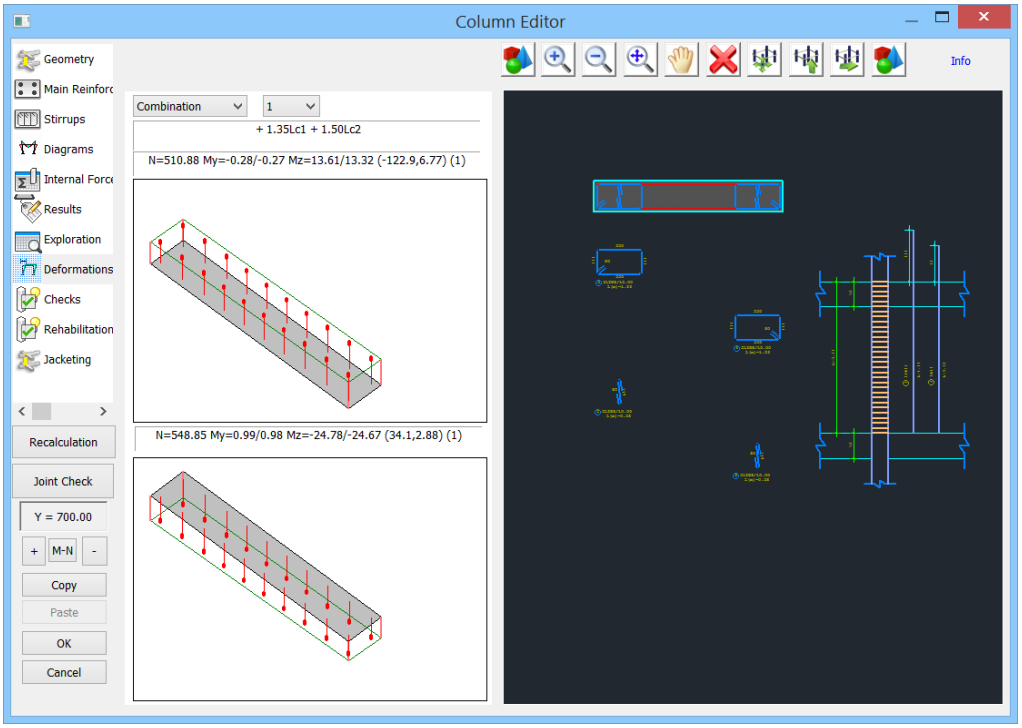
In the "Results" field, a TXT file format opens with the results of the design checks derived from the critical load combination.

7. Exploration



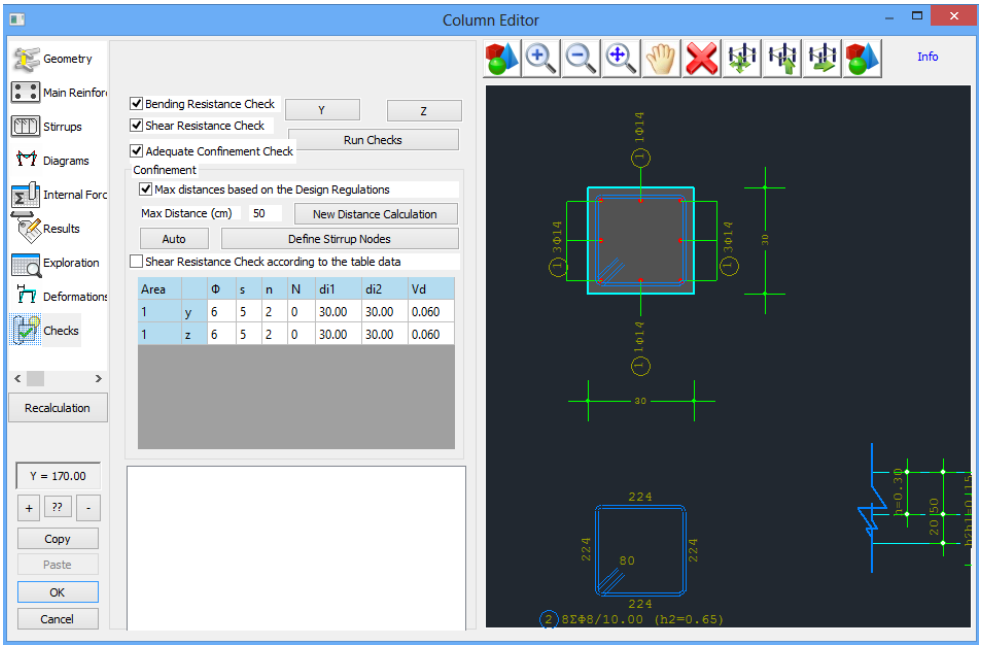
In the field "Exploration", similar to the field "Results", a TXT file format opens with the results of the design checks derived from all load combinations.

8. Deformations



In the "Deformations" field you can see how the cross-sections of the column, on the top and the bottom, are deformed, for each load case and each load combination. The compressive main reinforcement is denoted with blue color and the tensile with red.

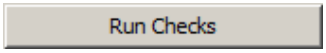
9. Checks



In the "Checks" field you can perform local design checks on the column depending on the modification you've applied to the steel reinforcement, using the tools of the "Columns-Walls Editor" dialog box. So when you make modifications or additions to the main steel reinforcement, you should check the column against biaxial bending failure, and when you make modifications or additions to the stirrups, you should check the column against shear failure and confinement.

9.1 Bending Resistance Check:

. Activate the checkbox "Bending Resistance Check" 

. Select the "Run Checks" command 

The program rechecks the column against biaxial bending considering the modified rebars and displays "sufficient" when the design checks are satisfied. When the cross-section fails, either on the top or the bottom, the numbers of the load combinations causing the failure are displayed in the corresponding cross-section.

The check of the column against biaxial bending failure concerns only the equilibrium of the cross-section (adequacy check) without considering all the other limitations (minimum distance of the rebars, maximum steel reinforcement, etc.).

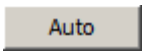
That's because it is possible to have a column that doesn't satisfy the design checks in the tab "Member design" yet it passes the local biaxial bending resistance check. This means that the column fails at the beginning because of the minimum distance of rebars or the maximum steel reinforcement in cross section. The type of failure is displayed in the "Exploration" file at the end of biaxial bending resistance checks.

```
        1          98     76.798     40.508    165.702
        1          99    -24.810     -2.454     88.114
Biaxial Bending Results    1 (1=okey,0=  fails      , 10=max As 11=max number ),
```

9.2 Shear Resistance Check:

. Activate the "Shear Resistance Check" 

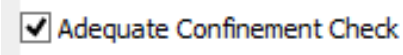
checkbox

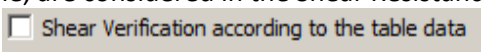
. Select the command "Auto" 

so that the following table will be updated, including the changes you have already done.

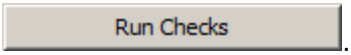
Area		Φ	s	n	N	di1	di2	Vd
1	y	8	10	6	0	100.00	30.00	0.028
1	z	8	10	2	0	100.00	30.00	0.022
2	z	8	10	5	0	80.00	30.00	0.022
2	y	8	10	2	0	80.00	30.00	0.028

NOTES

1. You can modify this table and change the diameter Φ of the rebar, the distance s or the number of the critical cross-sections n.
2. Activate the following command 

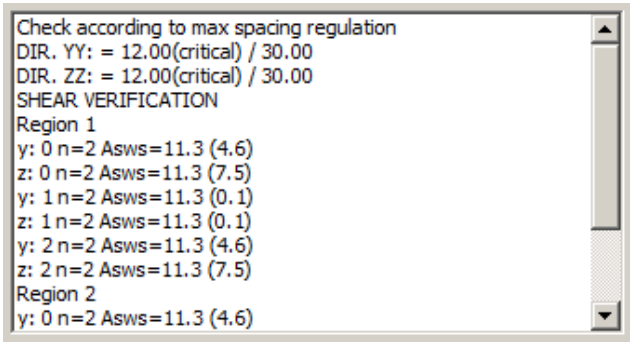
and the changes you make directly into the table, are considered in the Shear Resistance Check. If you do not activate the command 

then the initial values will be considered in the Shear Resistance Check, coming from the editor, automatically, by selecting the button "Auto".

. Select the command "Run Checks" 

.

On Shear Resistance Check, the program calculates the new spacing of the stirrups, according to the new diameter and the new number of the critical cross-sections.
The procedure is the following:
From the initial design, checks result in a required ratio A_{sw} / s of stirrups in Y and Z direction, for the critical and non-critical length (6 values). So, according to the new type of stirrups and the new diameter, the program starting from the maximum spacing, according to the design regulations, follows an iterative process searching that A_{sw} / s that is greater than the required one. The required is indicated in the parentheses.

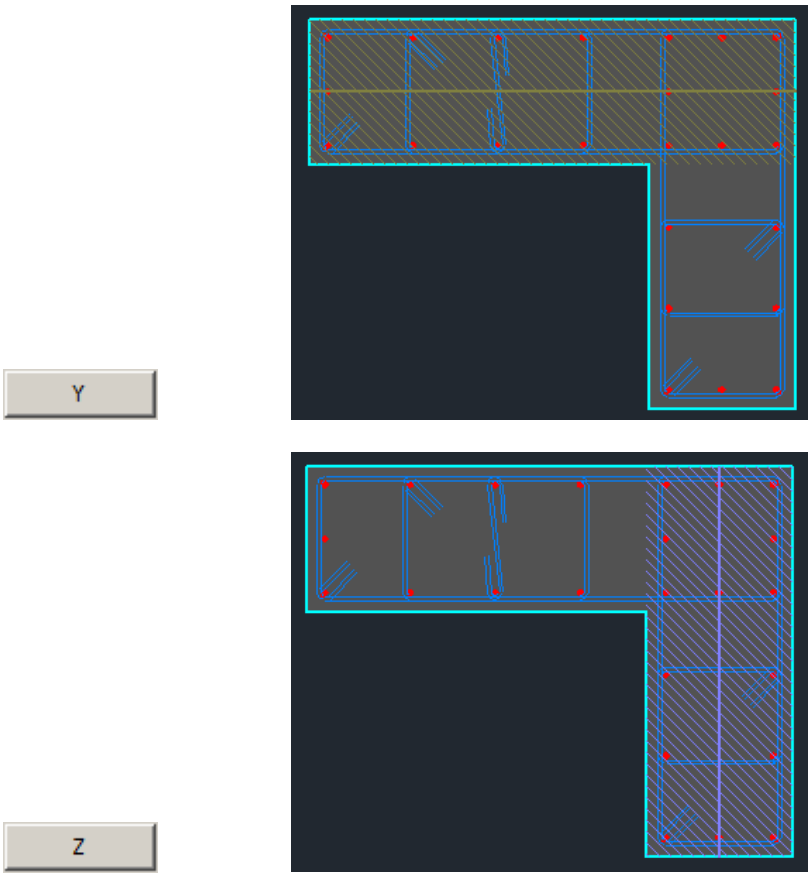


The results appear with respect to:

- Region (for cross-section with more than one critical region, e.g. Γ, Π)
- Direction (y, z)
- Height (0: critical zone top, 1: no critical zone, 2: critical zone bottom)

Press 

:



In the cross-section's detailing a marked area in Y or Z and a horizontal line, that indicates the direction Y or Z, respectively, appear, to distinguish easily the areas and directions and read the checks' results without difficulty.
You can repeat the design checks more than once, by modifying the table's data and changing the diameter Φ, distance s and number of sections n.

Area		Φ	s	n	N	di1	di2	Vd
1	y	8	10	6	0	100.00	30.00	0.028
1	z	8	10	2	0	100.00	30.00	0.022
2	z	8	10	5	0	80.00	30.00	0.022
2	y	8	10	2	0	80.00	30.00	0.028

Just remember to activate the following checkbox 

and when you define the stirrups, go back to "Stirrups" and make the changes to update the drawing and the calculations' print-out as well.

9.3 Confinement Verification:

. Activate the following checkbox ☑ Confinemnt Verification and

. Select the command "Auto" Auto to update the following table, including the changes you have already made.

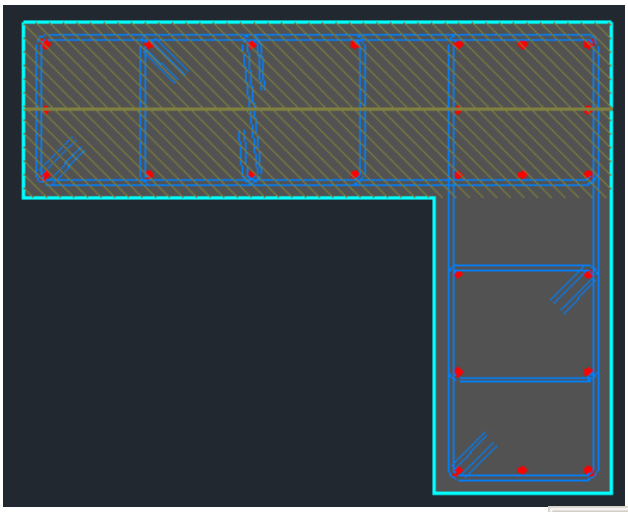
Area		Φ	s	n	N	di1	di2	Vd
1	y	8	10	6	0	100.00	30.00	0.028
1	z	8	10	2	0	100.00	30.00	0.022
2	z	8	10	5	0	80.00	30.00	0.022
2	y	8	10	2	0	80.00	30.00	0.028

The diameter Φ, the distance s, the number of sections n, the cross-section's dimensions d_1, d_2 and the value of the relative axial force v_d by region and direction are automatically updated.
The values in "N" column are the numbers of the vertices of the stirrups (stirrups – nodes) that represent the number of the confined main rebar. To fill in the column "N" follow the procedure:

. Select one by one the regions in each direction

Area		Φ	s	n	N	di1	di2	Vd
1	y	8	10	6	0	100.00	30.00	0.028
1	z	8	10	2	0	100.00	30.00	0.022
2	z	8	10	5	0	80.00	30.00	0.022
2	y	8	10	2	0	80.00	30.00	0.028

In the cross-section's detailing a marked area in Y or Z direction appears to be distinguished easily.



. Select the command "Define Stirrup Nodes" Define Stirrup Nodes

. Left click to show the rebar of the region confined by stirrups, regardless of the direction, starting from a rebar and ending in the same rebar.

Area		Φ	s	n	N	di1	di2	Vd
1	y	8	10	6	12	100.00	30.00	0.028
1	z	8	10	2	0	100.00	30.00	0.022
2	z	8	10	5	0	80.00	30.00	0.022
2	y	8	10	2	0	80.00	30.00	0.028

Repeat the process on the other direction, showing the same rebar. Alternatively, if you select both directions with the "Shift" button and define the vertices once, then both values of N will be filled in automatically.

Follow the same process for the second region, to complete the "N" column in total. The column "n" is already completed.

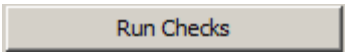
Area		Φ	s	n	N	di1	di2	Vd
1	y	8	10	6	12	100.00	30.00	0.028
1	z	8	10	2	12	100.00	30.00	0.022
2	z	8	10	5	10	80.00	30.00	0.022
2	y	8	10	2	10	80.00	30.00	0.028

Now you have two options:

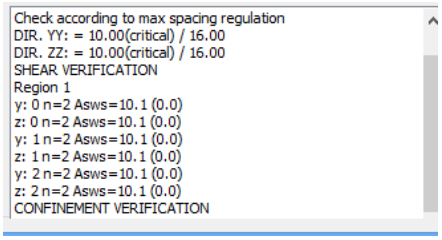
1. Perform the design check "Adequate Confinement check" for each region and direction.
2. Calculate the distance between the stirrups with the diameter Φ fixed, in order the confinement adequacy check to be satisfied.

Both cases in detail:

1. To check the adequacy of the confinement:

. Select the command "Run Checks" 

. Check the results



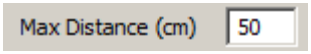
You can repeat the design checks more than once, by modifying the table's data and changing the diameter Φ, distance s and number of sections n.
When you define the stirrups, go back to "Stirrups" and make the changes to update the drawing and the calculations' print-out.

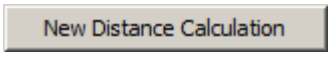
2. To calculate the distance between the stirrups with the diameter Φ fixed, in order, the confinement adequacy check to be satisfied.

. Type the new diameter Φ

Area		Φ	s	n	N	di1	di2	Vd
1	y	12	10	6	12	100.00	30.00	0.028
1	z	8	10	2	12	100.00	30.00	0.022
2	z	8	10	5	10	80.00	30.00	0.022
2	y	8	10	2	10	80.00	30.00	0.028

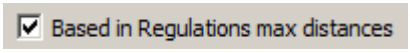
. Define the maximum distance as an upper limit to start the calculation process



. Select the command "New Distance Calculation" 

and the program calculates the distances and fills in the column "s".

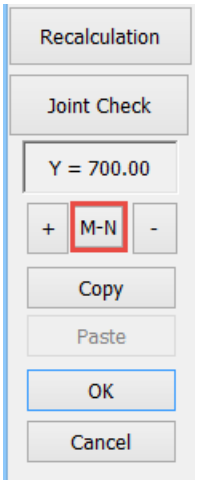
Area		Φ	s	n	N	di1	di2	Vd
1	y	12	44	6	12	100.00	30.00	0.028
1	z	8	21	2	12	100.00	30.00	0.022
2	z	8	20	5	10	80.00	30.00	0.022
2	y	8	21	2	10	80.00	30.00	0.028

If you activate the following checkbox 

then during the calculation of the distances that satisfy the design checks, the program will take into consideration the maximum distances proposed in the Design Regulations, too.

Always remember, when you define the stirrups, to go back to "Stirrups" and make the changes to update the drawing and the calculations' print-out.

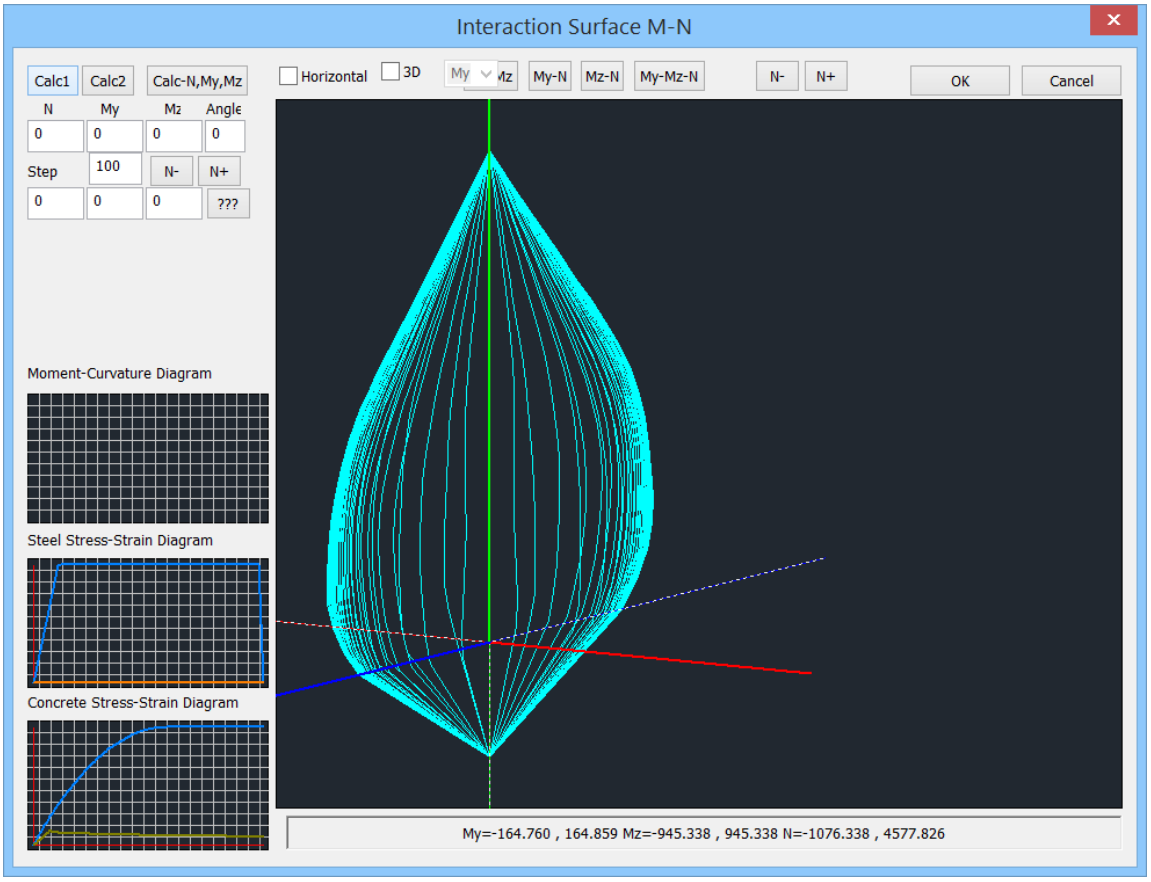
3. M-N Calculation



The "Column's Editor" tool enables the calculation of the interaction curves N-M and the interaction surfaces N-Mx-Mz as well.

For the interaction curves/surfaces click the following button 

.

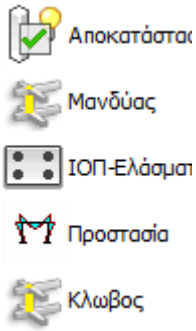


- Calculation of the interaction curves/surfaces M-N

It’s about the calculation and the display of the interaction surface of the axial load and the ultimate bending moment. It depends on the geometry of the cross-section, the material, and the steel reinforcement. It is a 3D surface and represents the envelope of the ultimate biaxial bending resistance and the axial load (My, Mz, N). Also, the strain-stress relationship diagrams for the steel and concrete as well as the moment – curvature diagram of the cross-section are displayed as well.

B. COLUMNS-WALLS REINFORCEMENT

SCADA Pro has integrated the tools for the needs of rehabilitation and reinforcement of the columns and walls, as predicted by the regulation of interventions (KANEPE)



As well as all the checks and procedures necessary for them

Regarding Confinement

Regarding **Confinement,** it is noted that it has been integrated for all existing types of column reinforcement as well as the steel cage in the program. Increase in concrete strength and fracture deformation occurs for all types of reinforcement (stirrups, plates, and FRPs).

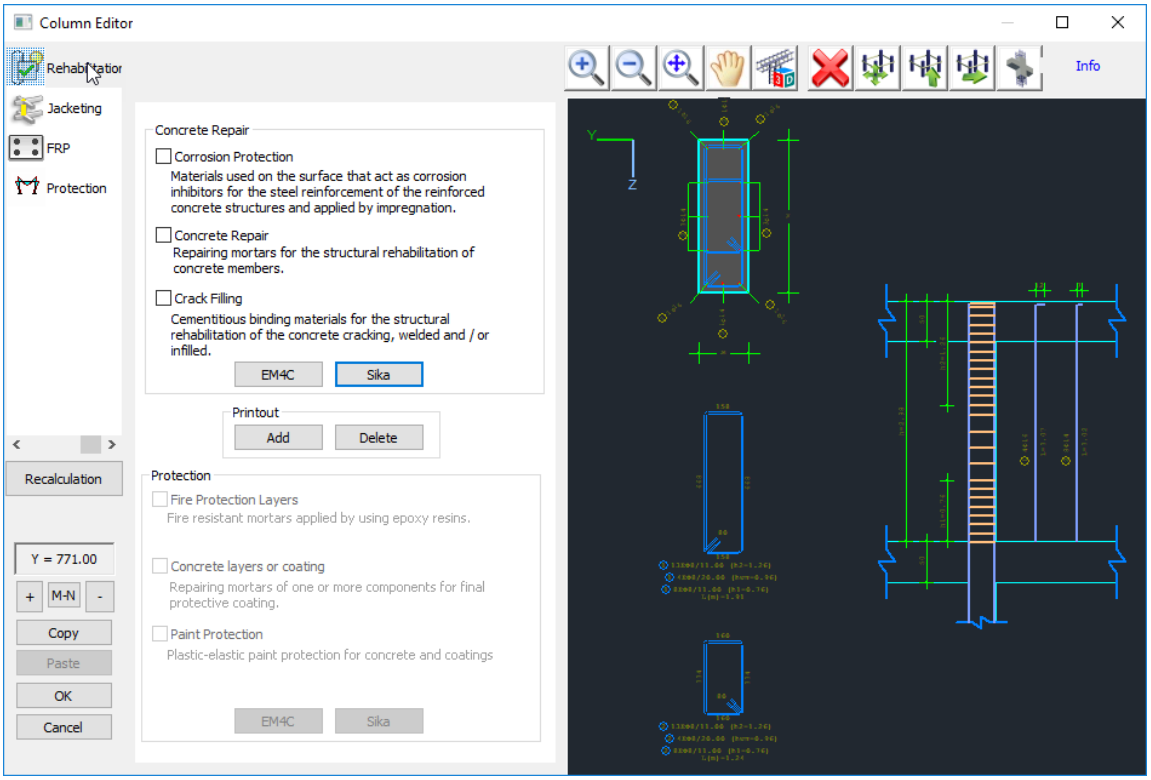
The new values are displayed on the page with the data of the existing section, in the printout of reinforcement.

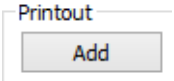
Confinement works only in columns (not walls) and requires enclosed reinforcement on all sides. Regarding walls'confinement, it is taken into account only in case of reinforcement with FRPs or plates.

⚠ The increased values of strength and deformation are displayed on the first page of the reinforcement printout.

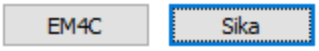
10. Rehabilitation of columns-walls

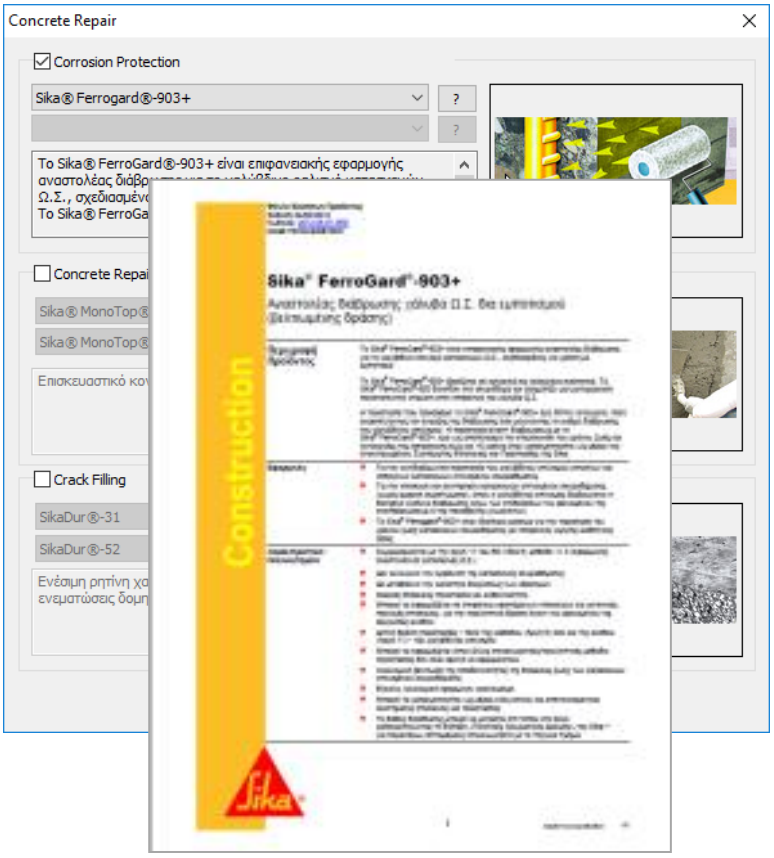
The "Rehabilitation" section contains tools for the rehabilitation of the columns according to the Code of Structural Interventions.

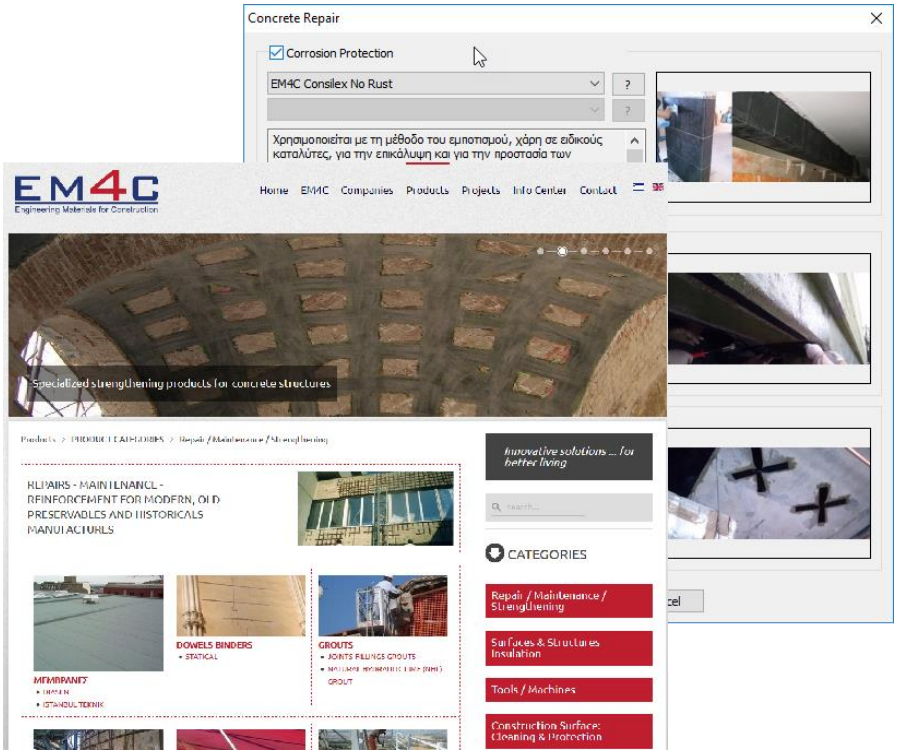


The user can select one of the three rehabilitation methods by activating the corresponding checkbox. Then, select the command "Add"  and the rehabilitation methods will be included in the final report.
Select the command "Delete", to exclude the rehabilitation methods from the report.

⚠ *Furthermore, in SCADA Pro, the techniques and the material considered in each rehabilitation method are enriched with the corresponding material and techniques of the companies' EM4C and Sika. The user has direct access to the library of EM4C and Sika materials by pressing the corresponding button, which appears in the dialog boxes related to column reinforcement.*

Select one command  , and then select the appropriate material for each rehabilitation method. Also, select the following button  and a PDF file, with an analytical description of the material properties as well as information related to its use, is automatically downloaded.

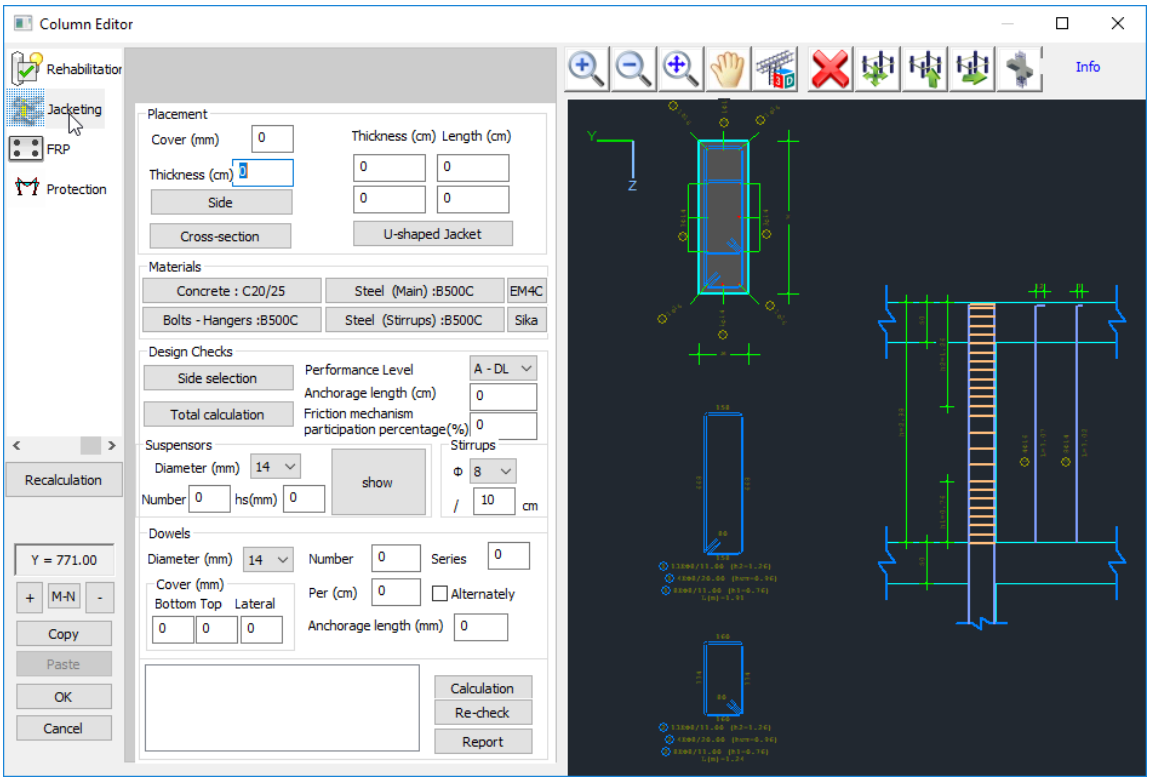

Concrete Repair
Corrosion Protection
Sika® Ferrogard®-903+
Concrete Repai
Crack Filling
SikaDur®-31
SikaDur®-52
Sika® FerroGard®-903+
Construction


Concrete Repair
Corrosion Protection
EM4C Consilex No Rust
Χρησιμοποιείται με τη μέθοδο του εμποτισμού, χάρη σε ειδικούς καταλύτες, για την επικάλυψη και για την προστασία των
EM4C
Home
EM4C
Companies
Products
Projects
Info Center
Contact
Specialized strengthening products for concrete structures
innovative solutions ... for better living
CATEGORIES
Repair / Maintenance / Strengthening
Surfaces & Structures Insulation
Tools / Machines
Construction Surface: Cleaning & Protection

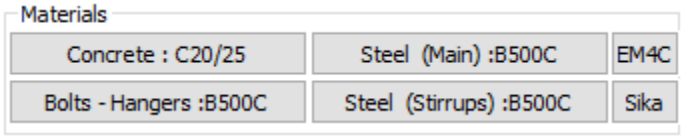
11. Concrete jacket for columns-walls

The section "Jacket" contains tools for the reinforcement of the columns according to the Code of Structural Interventions.

According to the Code of Structural Interventions, the concrete jacket is a uniform concrete layer that surrounds the column cross-section in a closed form. Otherwise, when the concrete layer is applied to some of the edges of the cross-section, then the reinforcing method is considered as additional concrete layers.



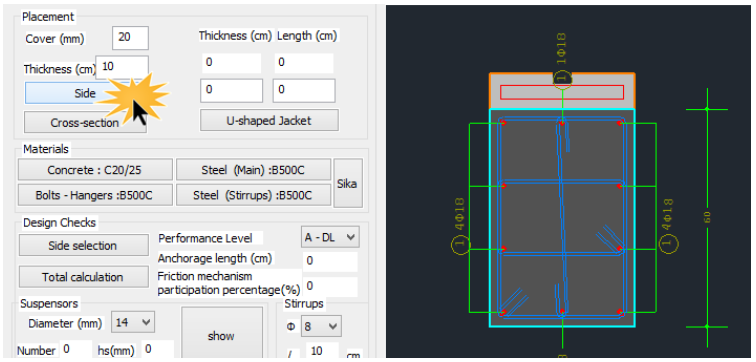
Define all "Materials" (concrete jacket, main steel reinforcement, stirrups)



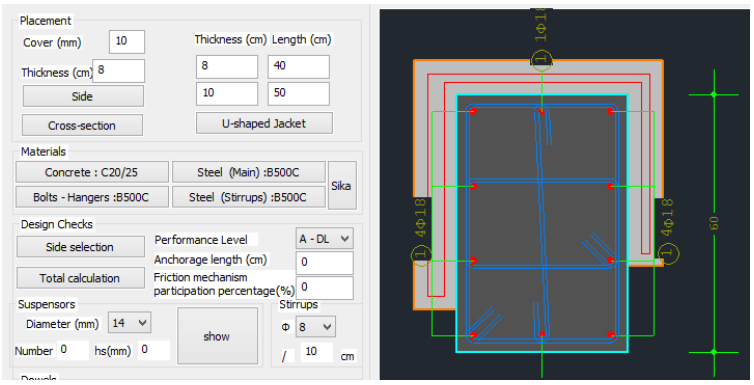
⚠ *Furthermore, in SCADA Pro, the techniques and the material considered in each rehabilitation method are enriched with the corresponding material and techniques of the companies' EM4C and Sika. The user has direct access to the library of EM4C and Sika materials by pressing the corresponding button, which appears in the dialog boxes related to column reinforcement.*

1. Define the "Cover" and "Thickness" of the concrete layer and apply either on the total of the cross-section as a jacket or a side by clicking the button "Side" and then selecting with the mouse the corresponding side. In this way, you can define different thickness per side. The cover is applied to all sides of the cross-sections.

The minimum "Thickness" of the jacket is modified concerning the type of the concrete (standard, gunite, special concrete).

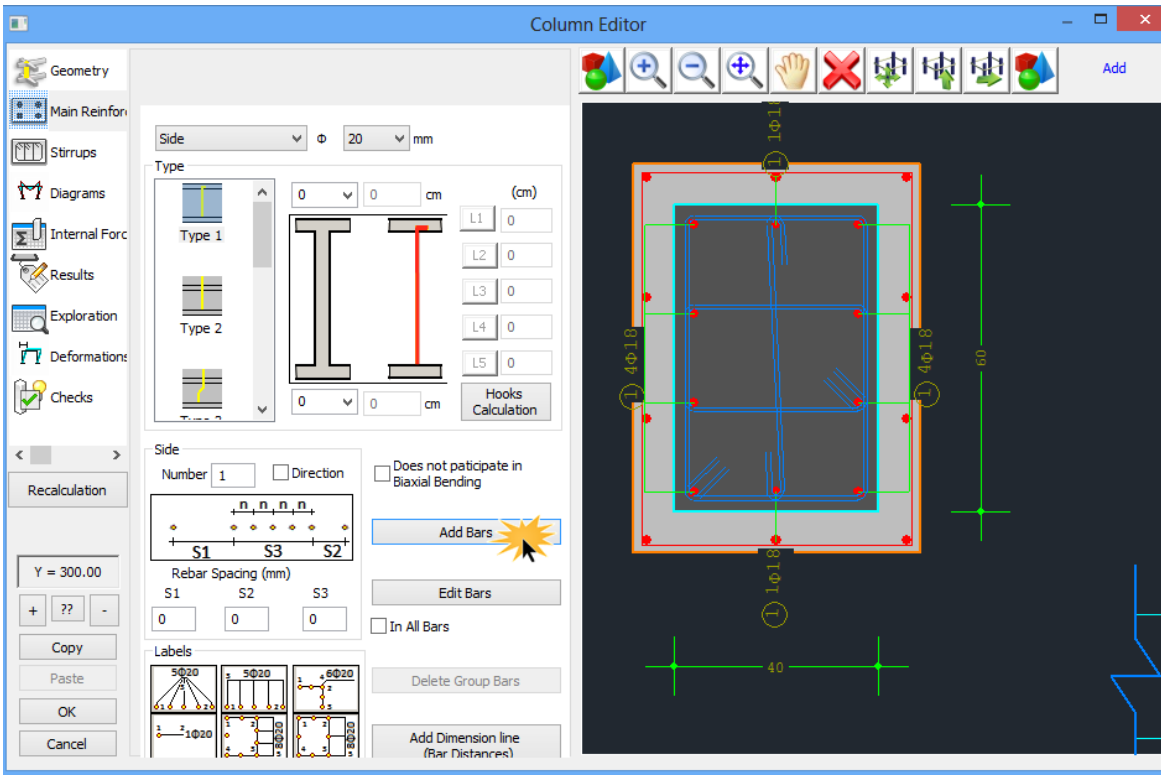


When the thickness per side defers, then you select the command "Side" and pick with the mouse the corresponding side. If the thickness is the same in the total cross-section, you select the button "Total cross-section".

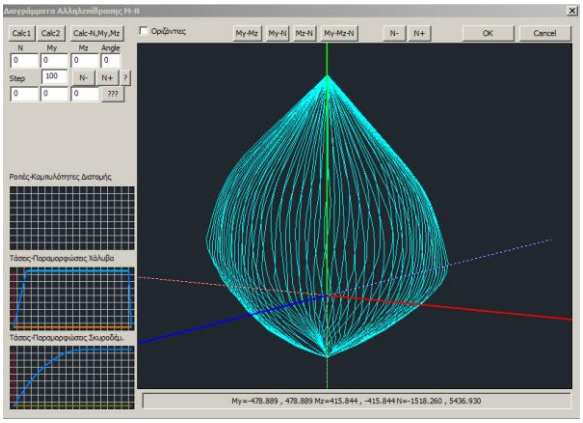


Furthermore, there is the option to insert U-shaped Jacket, typing the respective Thickness and Length.

2. Insert the steel reinforcement of the jacket with the commands "Main Reinforcement" and "Stirrups" from the list (Chapter B "Column's Detailing").

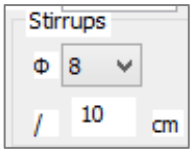


3. Calculate the new interaction surface of the reinforced cross-section by selecting the following button ??.

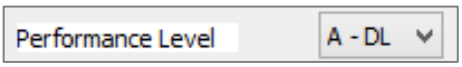


4. Return to the "Jacket" command for the calculation of the dowels.

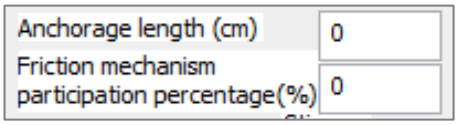
5. In the "Stirrups" field give the dimeter and the spacing of the stirrups of the jacket.



6. Select the appropriate "Performance level"; Damage Limitation-DL (Immediate Occupancy), Significant Damage-SD (Life Safety), Near Collapse-NC (Collapse Prevention).



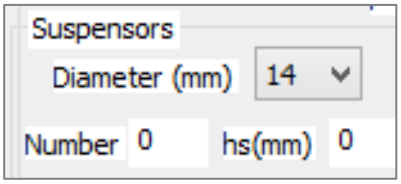
7. The compressive force F_{cm} of the jacket is safely transferred as a shear force along the interface through the three following mechanisms:
   - friction
   - welded suspensors
   - dowels



and are activated within an available assemblage length "u_o". The shear resistance along the interface is calculated considering the friction, welded suspensors, and dowels mechanisms.

In SCADA Pro the critical mechanism for the transfer of the compressive force is the dowels. The friction and the welded suspensors are optional and the user decides if they will be taken into consideration in the calculation of the shear resistance along the interface.

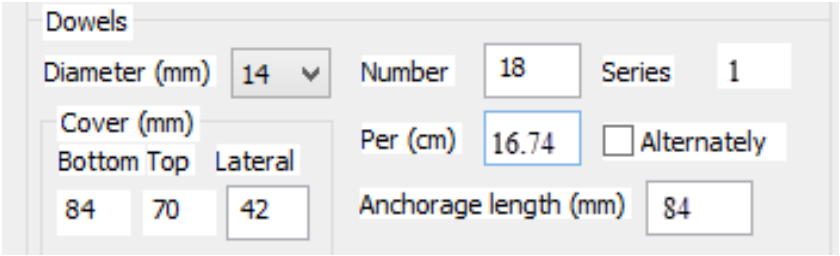
For the welded suspensors define the diameter, the number and the spacing h_s between the new and the existing main steel reinforcement.



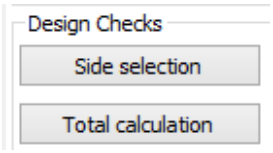
For the friction mechanism you have to define one of the following parameters:
- The assemblage length and then the program calculates the resistance considering the friction coefficient μ=1.0.
- The percentage (%) of the compressive force that will be transferred through the friction mechanism.

In case that the friction and the welded suspensors mechanisms are not taken into consideration, the total compressive force is transferred through the dowels.

8. In the field "Dowels" define the diameter and then the program calculates the number and the spacing of the dowels, as well as the cover in the top, bottom and in both sides:



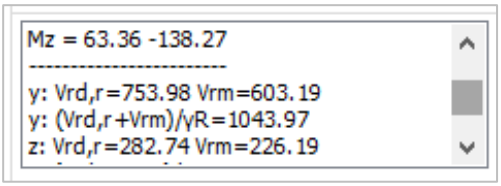
9. In the "Design checks" field, select the following:



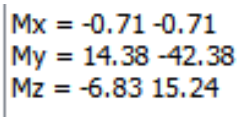
- The calculation in total: Select this command and the checks will be performed in all sides of the cross-section (according to Code of Structural Interventions) and the corresponding results will be presented per side.
- Side selection: Select the side for the checks to be performed per side. Show the corresponding side with the mouse, define the diameter of the dowels and click the button "Calculation". The program calculates the dowels' parameters automatically for the corresponding side.

The command "Recheck" will be activated in a future version of the software.

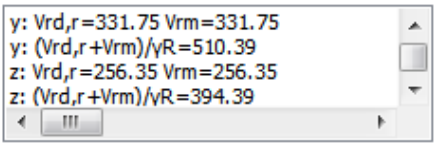
The results of the design checks are presented at the bottom of the dialog box:



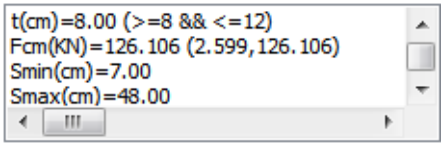
At the beginning of the design checks, the inertial forces in the top and the bottom of the column, appear.



Also, the shear resistance per direction, according to the Code of Structural Interventions, is presented.



In the end, the thickness of the concrete layer for the corresponding side, as well as all the parameters of the dowels, are presented.



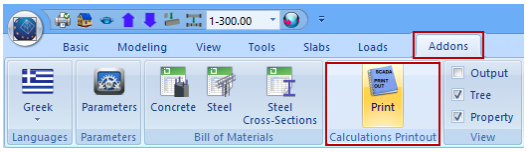
The program calculates the appropriate number of dowels by comparing the number of dowels based on the value of the compression force and the minimum number of dowels based on the jacket's area and keeps the greater.

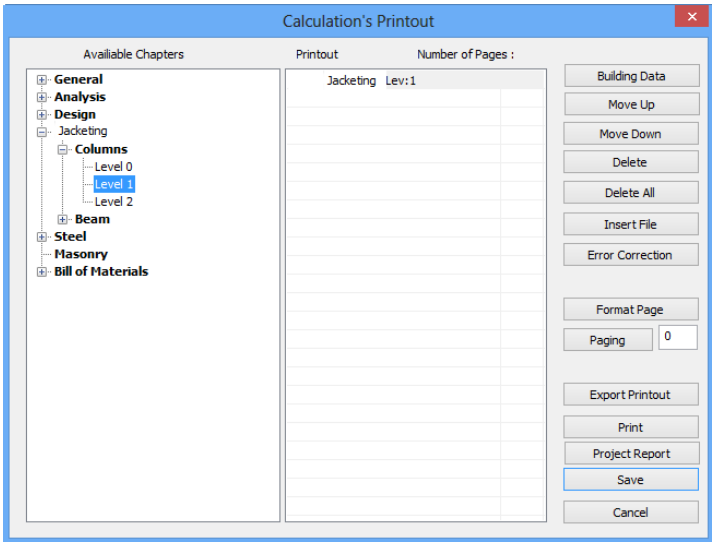
⚠ In the previous example, the minimum number of dowels is 13, while the calculated one is 18, which is the final number of dowels.

Finally, select the "Report" command to add the design checks' results in the corresponding chapter of the report.

The command "Recheck" will be activated in the future version for the software.

The analytical printout of the results is located in the ribbon "Add-ons" in the "Calculations' Printout" command.



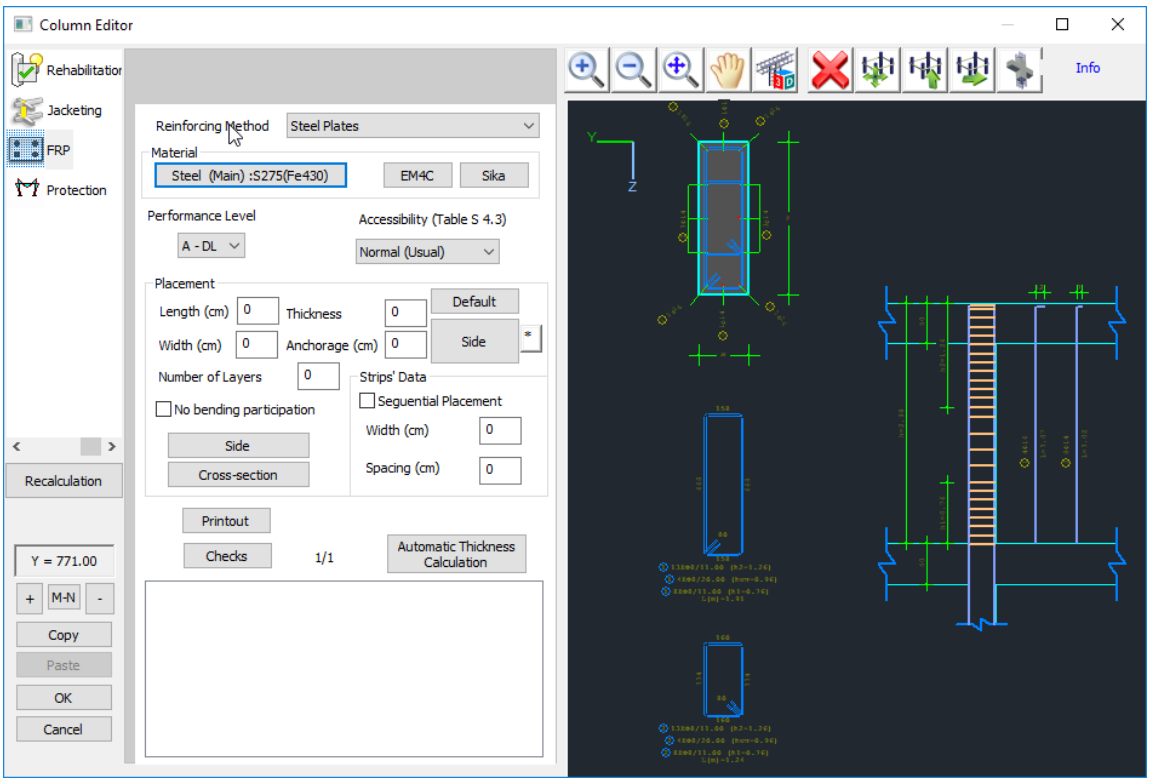


Select the section "Retrofitting methods" and select a level or levels. The corresponding results of the design checks, as well as the calculated number of dowels per level, will be recorded.

12. FRPs – laminates of columns-walls

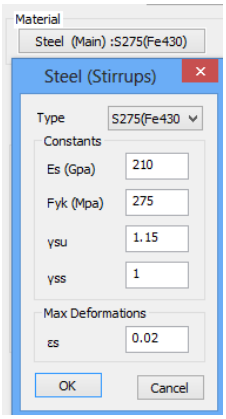
The steel laminates or the fiber reinforcing polymers (FRPs) is a reinforcing method that results in the increase of the bending resistance and the application of confinement reinforcement. The laminates are generally used as additional tensile reinforcement due to the inadequate existing steel reinforcement. The laminates strengthen the tension zone against flexural failure.

⚠ A reinforced concrete cross-section can be strengthened by bending with steel laminates or FRP fabrics. This reinforcing method is applied mainly in beams and slabs and rarely in columns because it is not allowed to be applied in regions under compression. As an exception, it can be applied in regions under compression when that regions resist against another type of failures, e.g. local buckling resistance of the rebar by applying confinement.

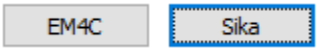


1. Select the reinforcing method; Steel Laminates or FRPs (Fiber reinforced polymers)

2. Select the Material.



⚠ *Furthermore, in SCADA Pro, the techniques and the material considered in each rehabilitation method are enriched with the corresponding material and techniques of the companies' EM4C and Sika. The user has direct access to the library of EM4C and Sika materials by pressing the corresponding button, which appears in the dialog boxes related to column reinforcement.*

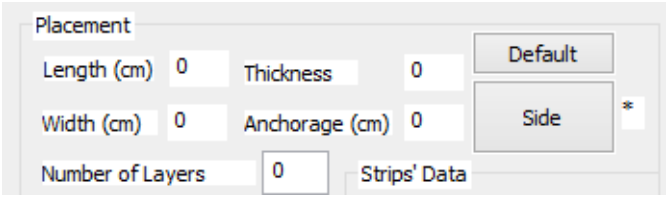
Select one command 

, and then select the appropriate material for each rehabilitation method. Also, select the following button 

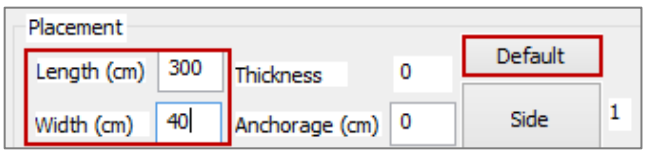
and a PDF file, with an analytical description of the material properties as well as information related to its use, is automatically downloaded.

3. Select the Performance Level and the Accessibility level

4. In the Placement field, select:



Default: This command is used so that the length of the column and the width of the laminate to be filled in automatically. The width of the laminate is equal to the width of the corresponding side of the column by default.



Afterwards, you set the value of the thickness and the length of the laminate in the corresponding fields with two ways:

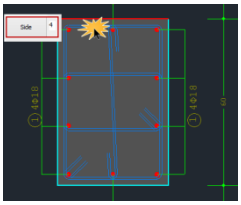
a. For each side: Select the button "Side" and show with the mouse the corresponding side of the column.

b. For the total cross-section: Select the button "Cross-section".

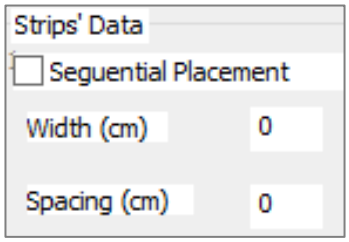
The "Default" command sets the data for all sides of the column. If you want to import laminates in all sides with the same thickness, you enter, at first, the thickness and the anchorage length. Then, click the button "Cross-section".

If you want afterwards to change the thickness of the laminates of the cross-section in total, set a new value for the thickness and press the button "Default" without pressing again the button "Total cross-section". The existing laminates change considering the new thickness value.

Side data: This command is used to show the number of the side selected with the mouse and the data of the reinforcing method applied in the corresponding side.

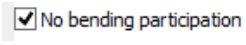


Layers: In this field set the number of the layers.

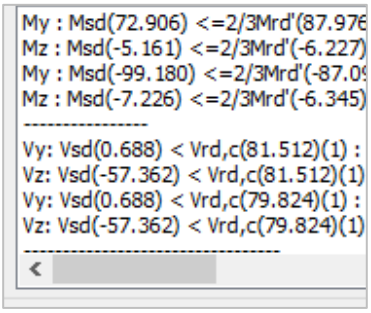


The placement of the laminates can be uniform or in strips; continuous or discontinuous with intermediate spacing.

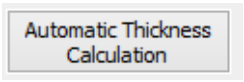
Activate the "Continuous Formulation" in the "Strips' data" field and define the width of the laminate. For considering the discontinuous formulation, deactivate the previous checkbox and define the spacing between the strips.

Activate the checkbox "No participation in bending" 

and the laminate in the corresponding side will not participate in the bending resistance of the reinforced cross-section.



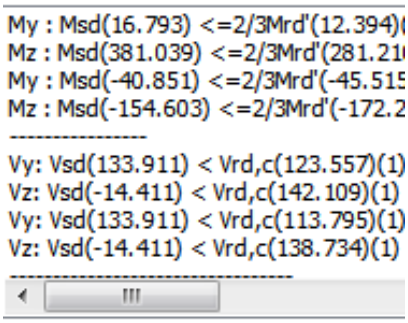
By selecting the "Design checks" command, the program calculates and presents the minimum thickness t_1 and t_2 per side, in the checks' results with respect to the cross-section of the laminate and the type of the material. Then, the thickness t_1 and t_2 are calculated again with reference to the minimum values of t_1 and t_2 and the design checks have to be repeated. Since the calculation of the thickness t_1 and t_2 is an iterative method, select the button "Automatic calculation of the thickness".



Then the program automatically calculates the final minimum thickness t_2, which is presented in the window at the bottom of the dialog box. Then, you have to set this calculated value in the corresponding field and repeat the final design checks.

⚠ The structural adequacy of the laminate or the FRP is reached with the increase of the thickness or the number of the layers.

In the section on the results of the design checks first, the bending resistance checks of the cross-section and the shear resistance check by direction X or Z according to the Code of Structural Interventions are presented.



Furthermore, the results and the value of the ΔM parameter are presented for each side; the difference between the design bending moment and the moment resistance of the initial cross-section is calculated. If the parameter ΔM is positive (the initial cross-section should be reinforced) the thickness values t_1 and t_2 are calculated as described above. The thickness t is defined by the user.

ΔM=45.86
σjd1 = 293995.859
σjd2 = 447795.526
min T(mm) : t=0.400 t1=0.693 t2=0.455

⚠ In the previous example, the thickness *t* is less than the appropriate t_1 και t_2. So the thickness *t* must be equal to 0.7. If you preserve the thickness value *t=0.4* then should be used two layers. The corresponding results are presented below:

ΔM=45.86
σjd1 = 293995.859
σjd2 = 316639.253
min T(mm) : t=0.400 t1=0.347 t2=0.322

So, if you use two layers, then a minimum thickness t=0.35 is needed.

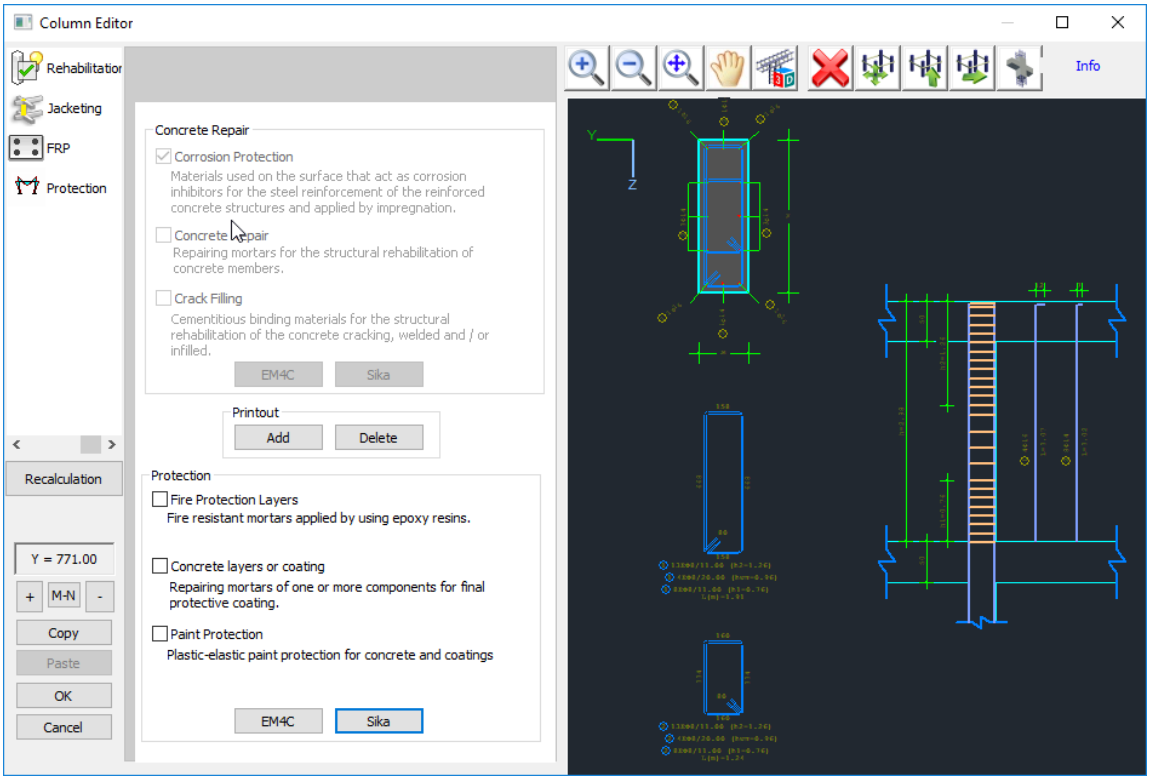
If ΔM=0, then there is no need for reinforcement, so $t_1=t_2=0$.

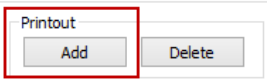
Finally, the shear resistance check, according to EC8, is presented.

Select the "Printout" command to add the results in the corresponding chapter of the report of the study.

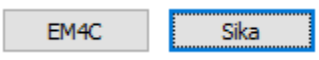
13. Protection of columns-walls

The section "Protection" contains the tools for the application of protection methods in columns.



The user can select one of the three rehabilitation methods by activating the corresponding checkbox. Then, select the "Add" command  and the rehabilitation methods will be included in the final report.

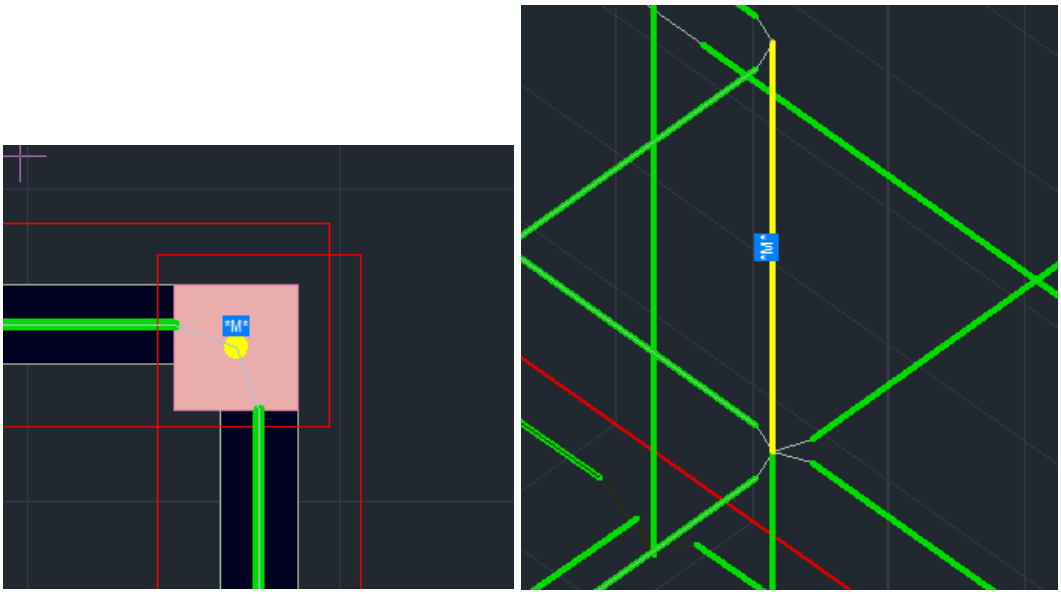
⚠ *Furthermore, in SCADA Pro, the techniques and the material considered in each rehabilitation method are enriched with the corresponding material and techniques of the companies' EM4C and Sika. The user has direct access to the library of EM4C and Sika materials by pressing the corresponding button, which appears in the dialog boxes related to column reinforcement.*

Select one command  , and then select the appropriate material for each rehabilitation method. Also, select the following button  and a PDF file, with an analytical description of the material properties as well as information related to its use, is automatically downloaded.

IMPORTANT NOTE:

The reinforced parts of the columns and walls are pointed out on the screen:

1. In plan view: The node is colored yellow
2. In 3D view: The structural element is colored yellow.



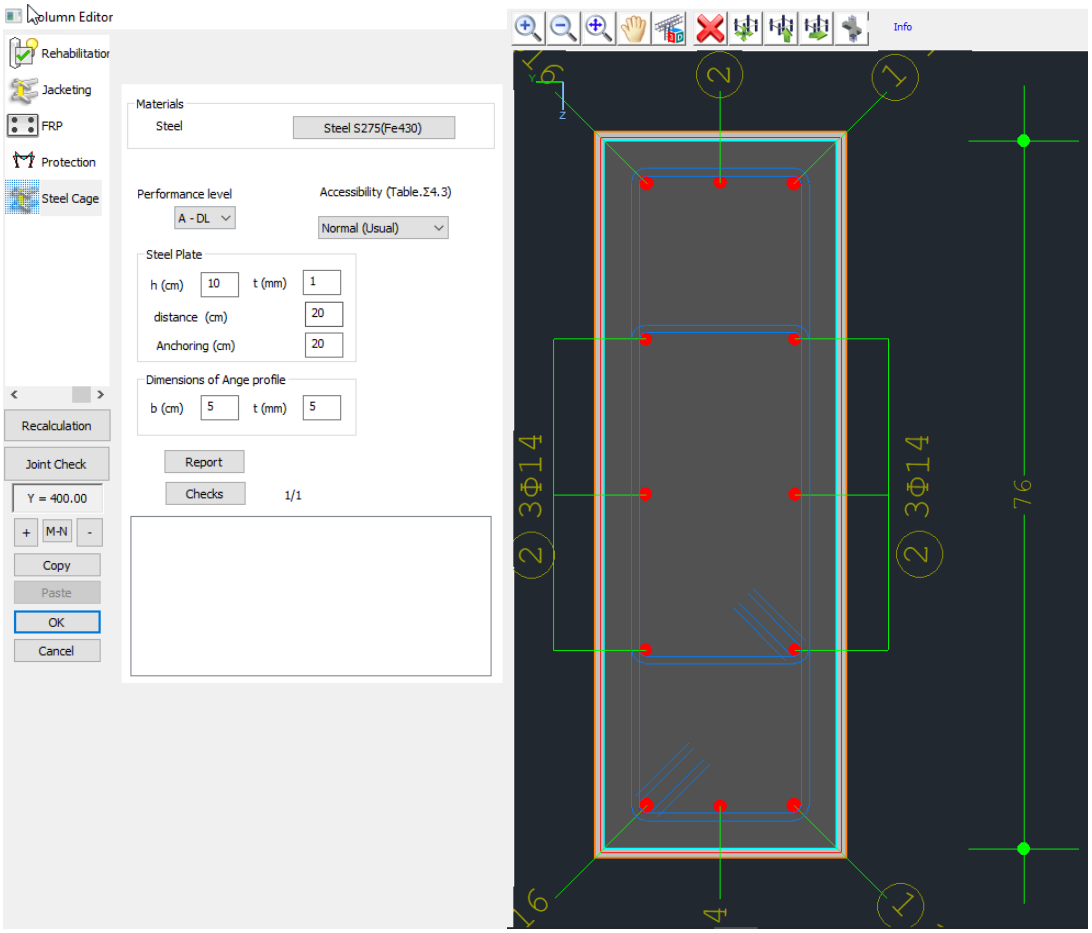
Also, according to the type of the reinforcing method, an indicative letter appears of the reinforcing method:

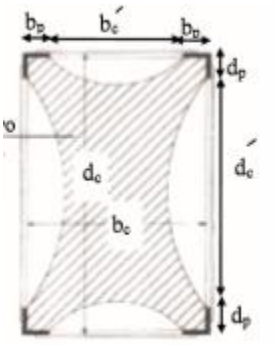
- Concrete Jacket: "J"
- Laminate: "L"
- FRP: "F"

⚠ Prerequisite for the appearance of the label is that you have first selected the "Report" button in the dialog box of the corresponding column reinforcing method.

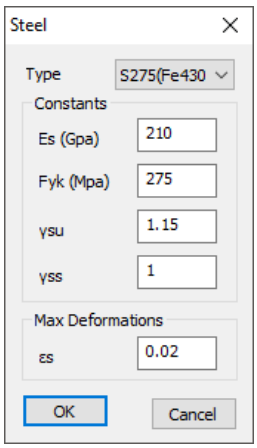
14. Steel Cage

The Steel Cage contains the tools necessary for reinforcing the columns and the walls, as described in Interventions Regulation (ΚΑΝ.ΕΠΕ). The contribution of the cage lies in the confinement as well as the shear strength.

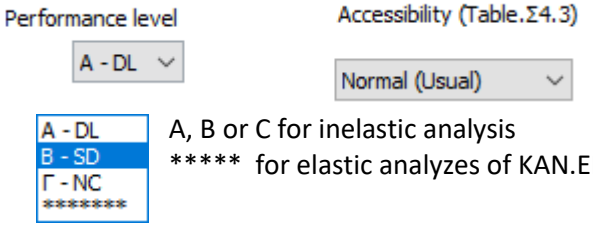

Column Editor
Rehabilitation
Jacketing
FRP
Protection
Steel Cage
Materials
Steel
Steel S275(Fe430)
Performance level
A - DL
Accessibility (Table.Σ4.3)
Normal (Usual)
Steel Plate
h (cm) 10
t (mm) 1
distance (cm) 20
Anchoring (cm) 20
Dimensions of Ange profile
b (cm) 5
t (mm) 5
Report
Checks
1/1
Recalculation
Joint Check
Y = 400.00
M-N
Copy
Paste
OK
Cancel
Info
3Φ14
3Φ14
76


b_c
d_c
d_c
b_c

1. Define Steel Material

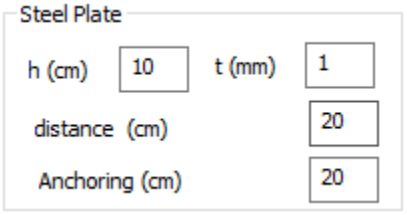


2. Choose Performance Level and Accessibility

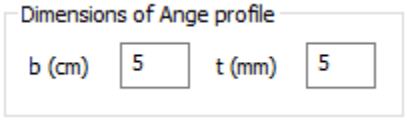


A, B or C for inelastic analysis
***** for elastic analyzes of ΚΑΝ.ΕΠΕ

3. In the field Steel Plate, define the height and the thickness of the plate, the distance between them as well as the anchoring.

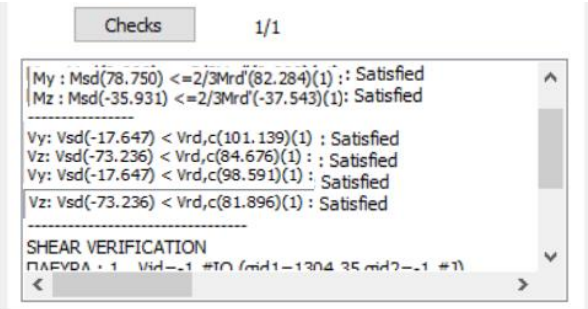


4. In the field Dimensions of Ange profile, define the dimension of angle profile which is considered as square edged equal, as well as its thickness.



5. By choosing Checks, the program calculates and displays the results. At the beginning of the checks, the bending sufficiency check and the shear adequacy check of the

concrete are shown. The shear check of the reinforced element is then displayed.



NOTES:

- The increased values of strength and deformation are displayed on the first page of the reinforcement printout.
- Also, in case of a steel cage in the second page of the reinforcement printout the dimensions of the angular are displayed.
- Finally, the dimensions of the plates are displayed in the next table in the shear check.