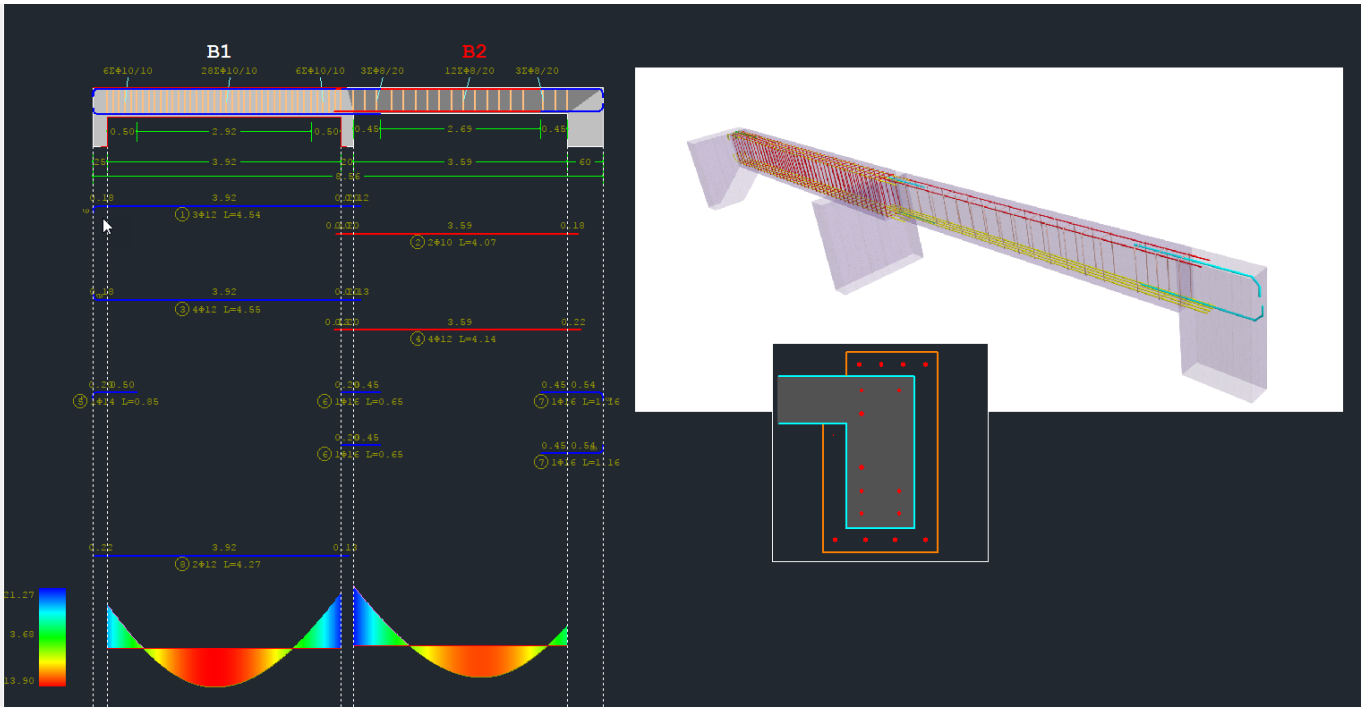
SCADA Pro™
Structural Analysis & Design

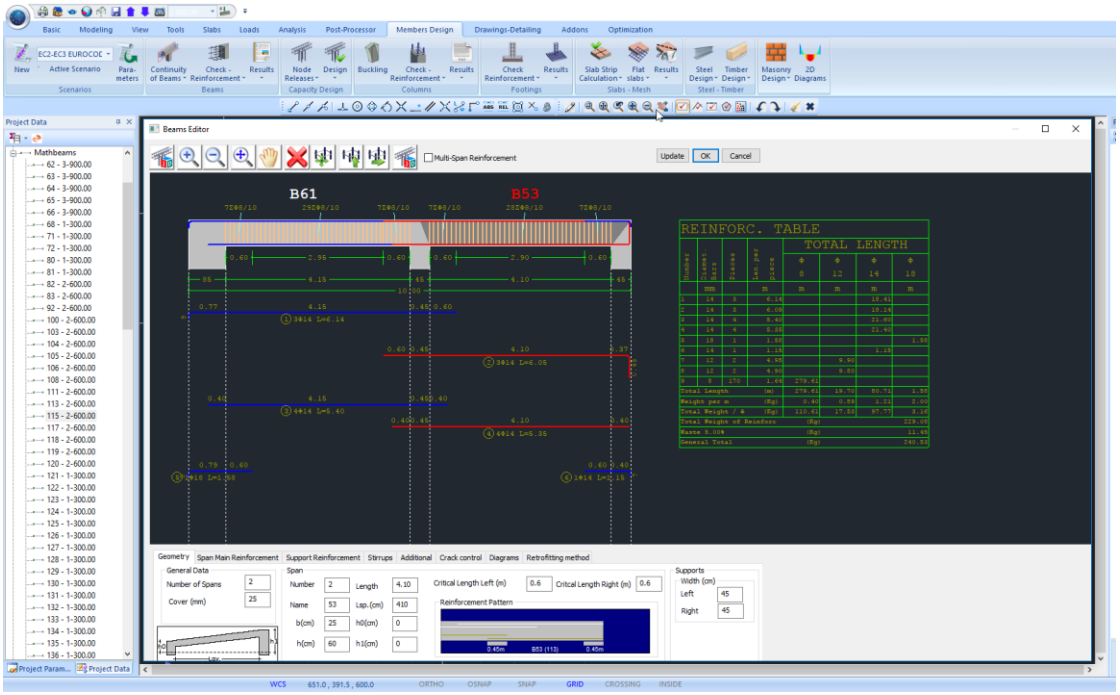
User's Manual
A.BEAM'S DETAILING



ACE HELLAS
INTEGRATED SOLUTIONS

CONTENTS

A. BEAM’S DETAILING



The New Beam’s Editor - “Detailing” of SCADA Pro is part of an innovative new group of tools that helps you manage the beams designing details.

Using the “Detailing” command you can edit, modify, and define cross-section’s details and steel reinforcement. You can also display all the diagrams of the internal forces for each load or load combination or apply reinforcement techniques.

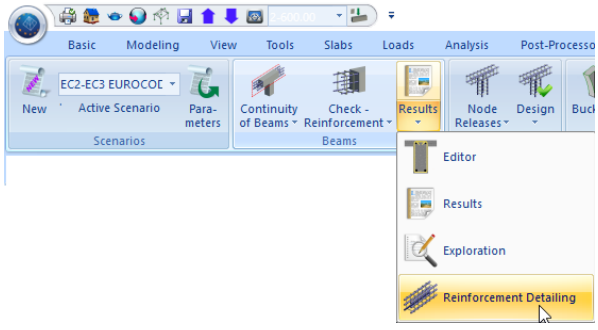
It is an integrated, flexible, easy and very useful tool that saves you a lot of time.

NOTE:

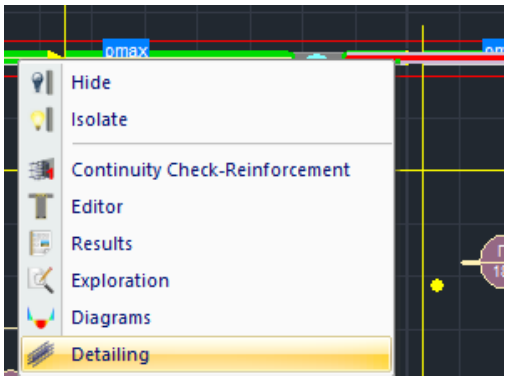
⚠ Prerequisite for accessing the “Detailing” tool is the design of beams to be preceded.

There are two ways to access the “Detailing” tool:

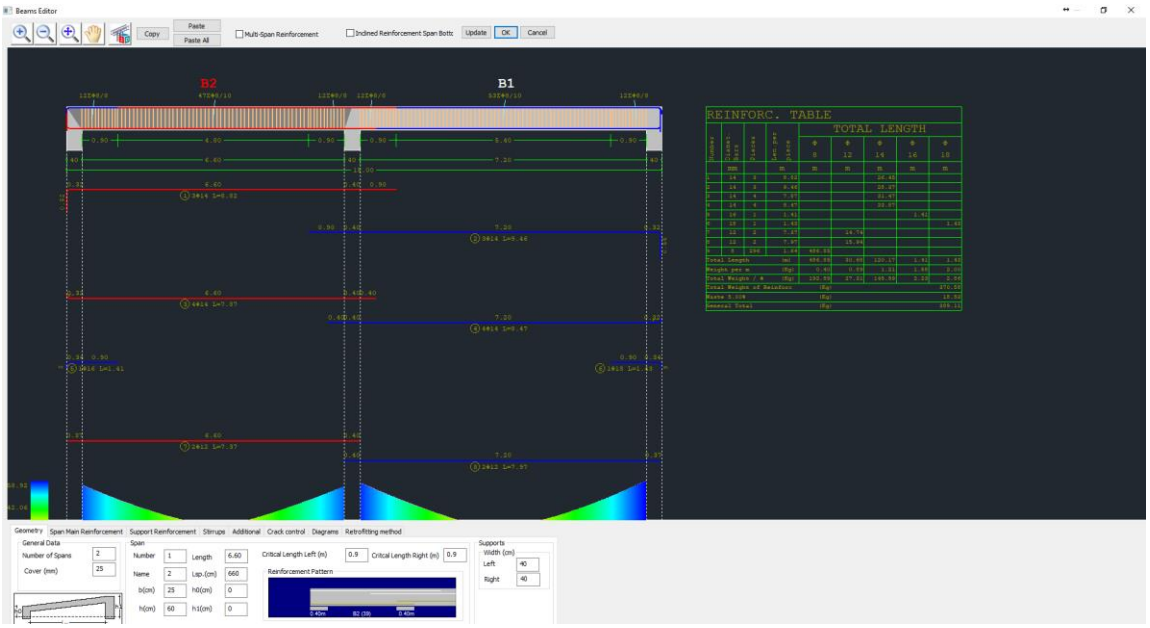
1) Open the “Members Design>> Beams >> Results >> “Reinforcement Detailing” Unit



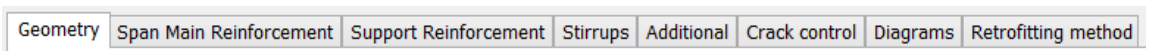
2) While "Members Design" unit is selected, right click on the beam in the interface and the following list opens:



Select the command to open the following dialog box:



The window entitled "Beams Editor" contains a cad interface on the top, with all the steel reinforcement details and the steel reinforcement table, as well as the diagrams of the inertial forces, adapted to the changes made in the parameters. The parameter's fields are located at the bottom of the window consisting of 8 units which are explained in detail in subsequent chapters:



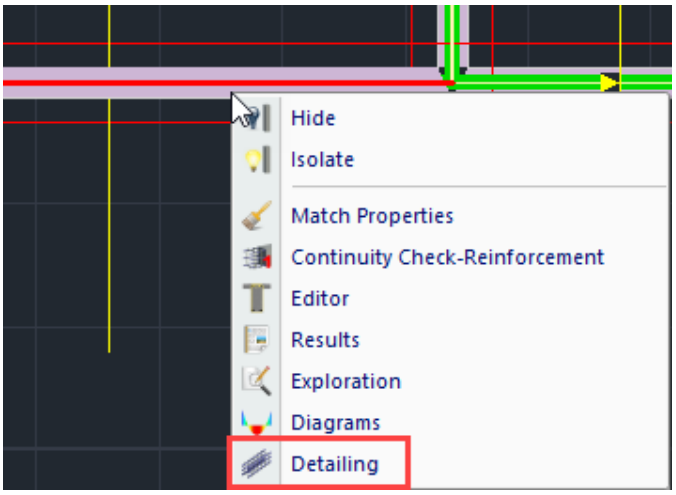
In the following chapters, the units are described one by one.

NOTE:

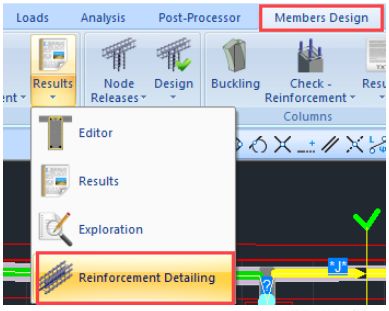
⚠ Prerequisite for the access to the tool "Beams Reinforcement Detailing" is the beam design check.

You can have access to the toll in 2 ways:

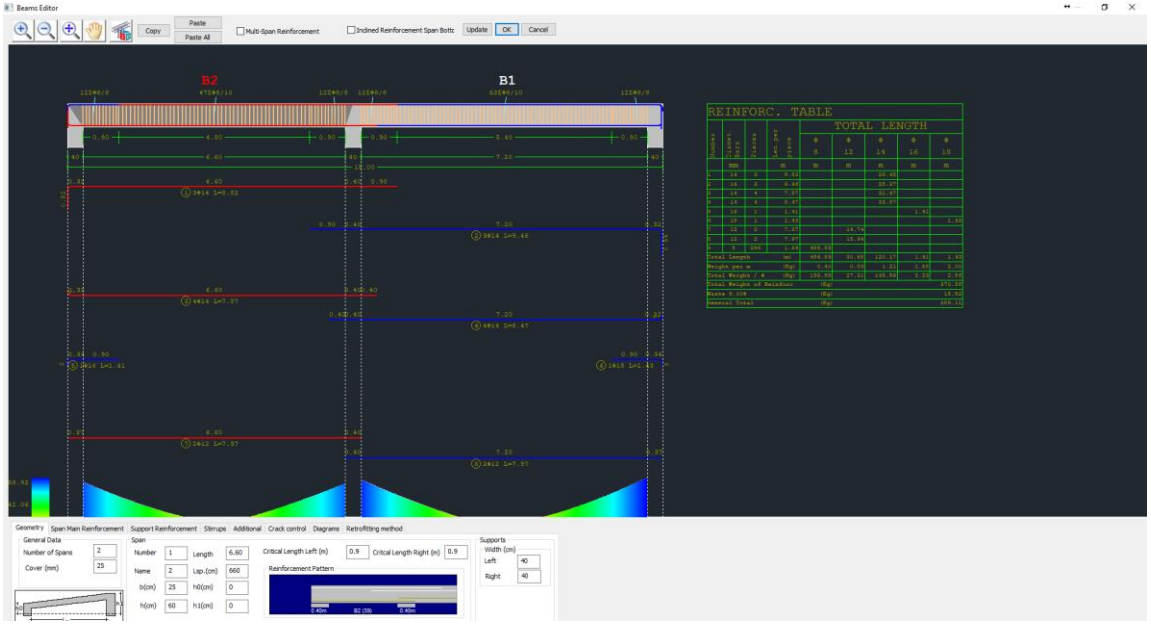
1) Following the path “Members Design>> Beams >> Results >> Reinforcement Detailing”



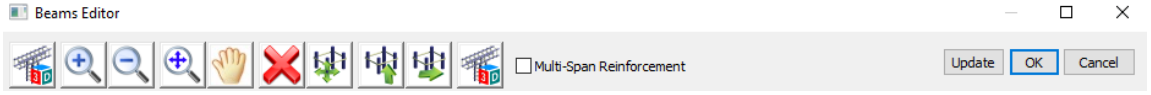
2) In “Members Design” field, right click on the member of the beam,



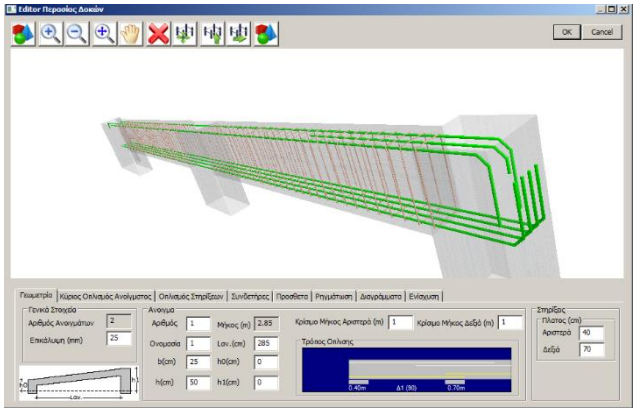
and the dialog window opens



The horizontal bar above the cad interface is used to manage the drawing.



Details:



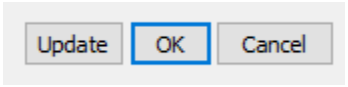
 : This button is used to display beam reinforcement in 3D.

⚠ Use mouse scroll to move, rotate and zoom the design.

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

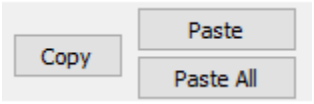
 : This is the “Pan” button.

 : These buttons are the arrows to move the drawing in different directions.



Press the button:
“Update”, to update the modifications you made
“OK” to save the changes you made in the editor and turn back into SCADA Pro’s interface.
“Cancel” to turn back into SCADA Pro’s interface without saving the changes you made in the editor.

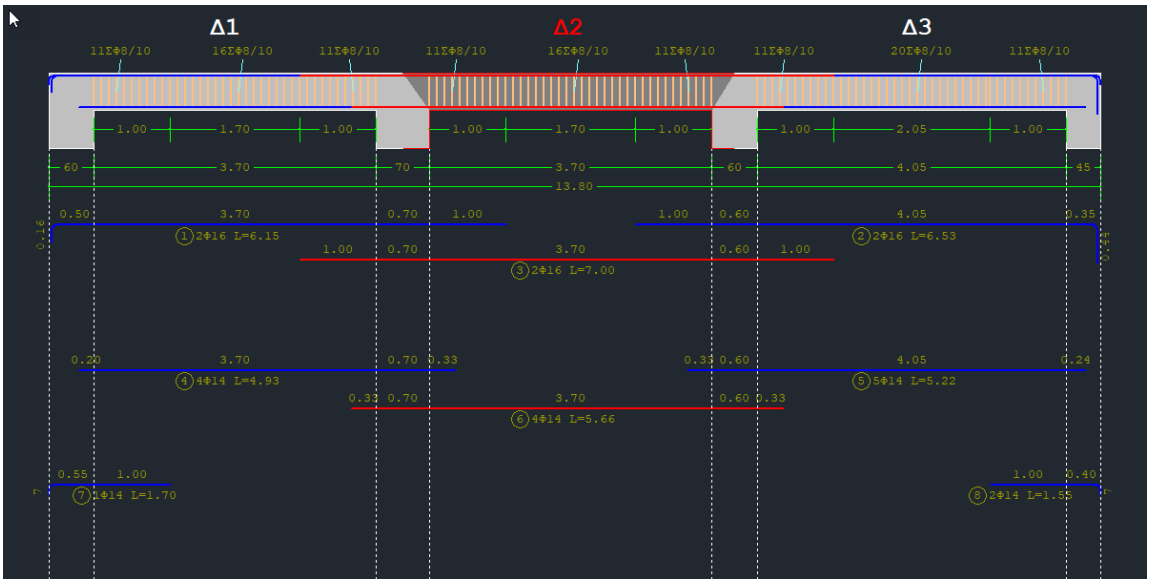
a) COPY-PASTE REINFORCEMENT



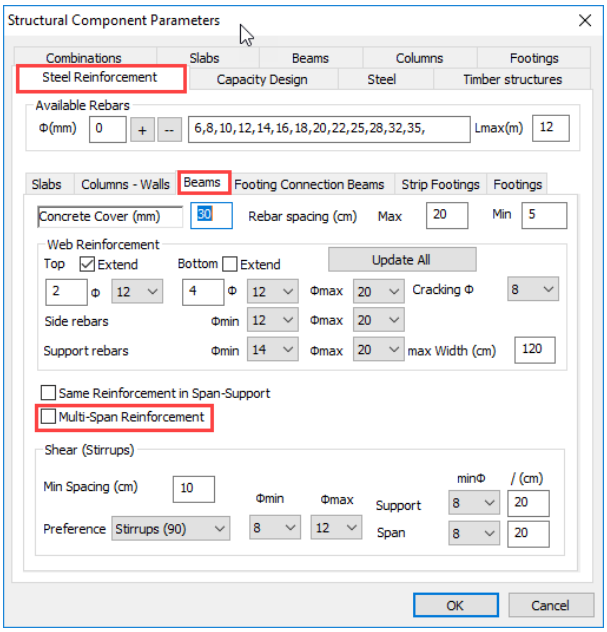
The Copy and Paste or Paste All commands allow the (Copy) of a span’s reinforcement to another span (Paste) or all spans of the beam continuity (Paste all).

USE:
Select a span by left clicking. (The selected span is displayed with red color)
- Click on COPY and then by left clicking, point the span to which the reinforcement will be copied and then click on PASTE.
- Click on COPY and PASTE All in order the reinforcement of the selected span to be copied to all spans of the beam continuity.
- Click on COPY, close the Details window and open the details of another beam continuity in the same or a different level. By left clicking, point the span to which the reinforcement will be copied and then click on PASTE.

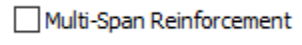
- Click on COPY, close the Details window and open the details of another beam continuity in the same or a different level and then click on PASTE ALL in order the reinforcement of the selected span to be copied to all spans of the new beam continuity.



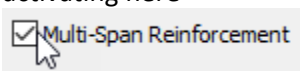
b) MULTI-SPAN REINFORCEMENT



If in the drawing parameters,



is not selected, activating here



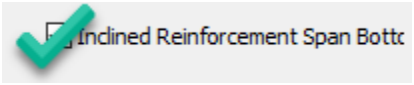
the calculated reinforcement is placed uniformly within the beam, and vice versa (if it was common appears as not).

ATTENTION:

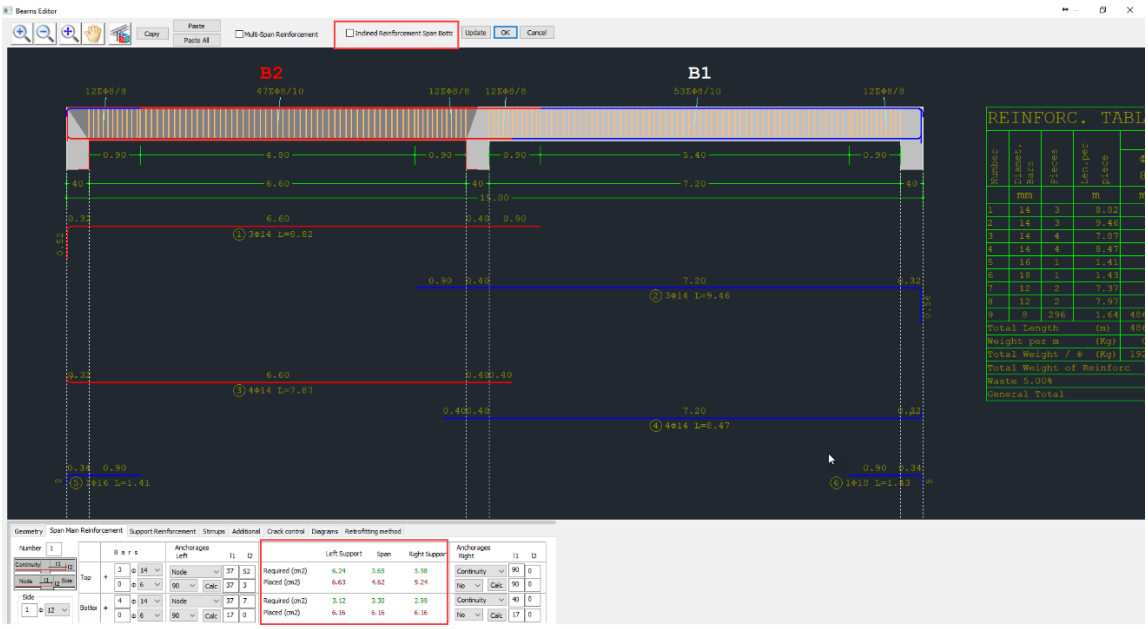
The reinforcement is not recalculated as Common; it simply places the already calculated non-common reinforcement as Common and vice versa.

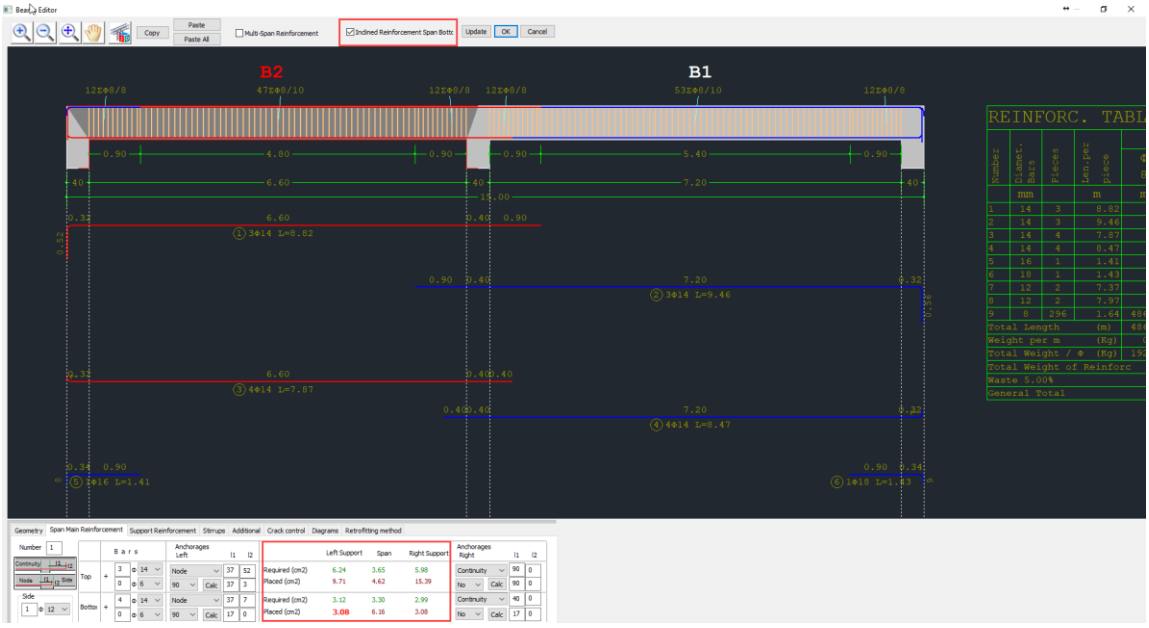
a) INCLINED REINFORCEMENT SPAN BOTTOM

Moreover, by activating the:

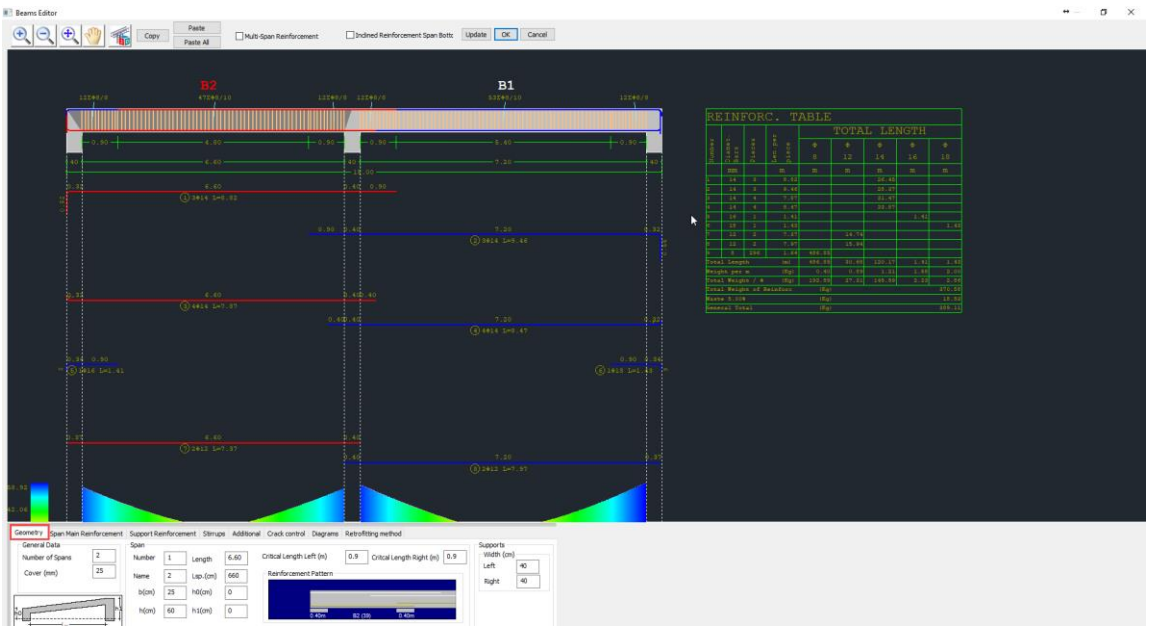


Half of the bottom reinforcement is considered as inclined reinforcement, and as a result, it is added on the top of the supports and removed from the bottom of the supports.

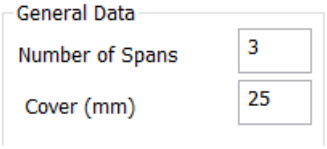




1. Geometry



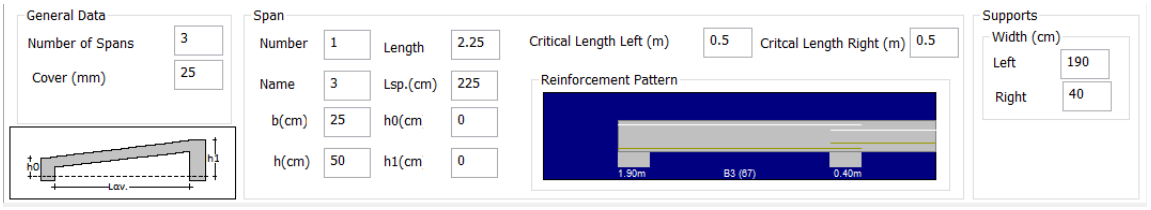
The first unit of "Beams Editor" refers to the geometry. It includes information related to spans and supports of the beam continuity and general data.



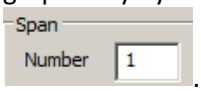
General Data refers to the entire continuity of the beam and includes (i) the number of spans (non-modifiable) and (ii) the cover in mm.

ATTENTION:

*The Cover * listed in General Data is purely designed. It means that it is not taken into account in the calculation of the reinforcement for this and is not mentioned in the issue. If you want to change the Cover, it is right to do it through the reinforcement parameters.*

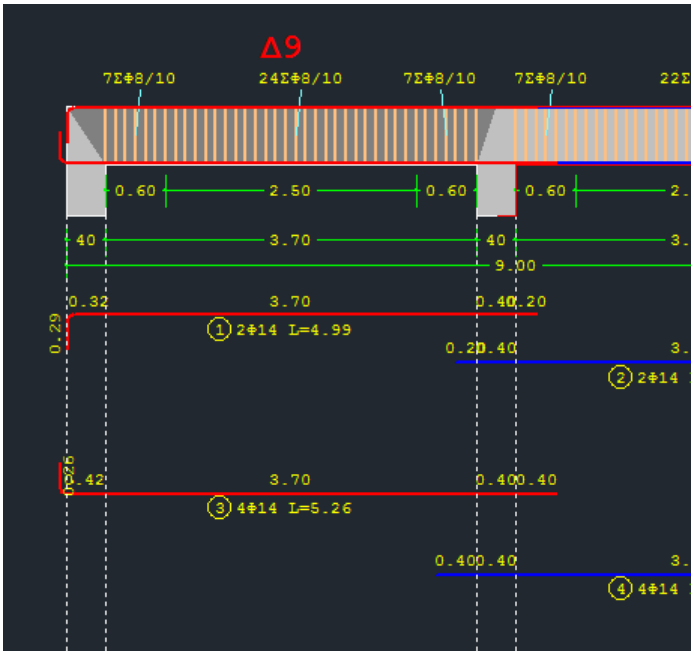


"Span" and "Support" areas can be adjusted in the selected field. You can choose the span graphically by left clicking on the drawing or by typing directly the number in the "Number" field

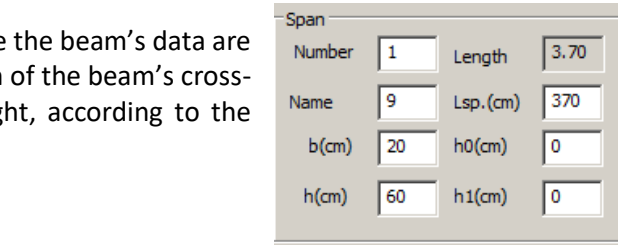


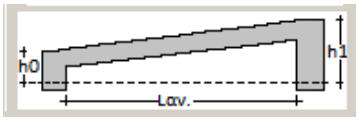
.

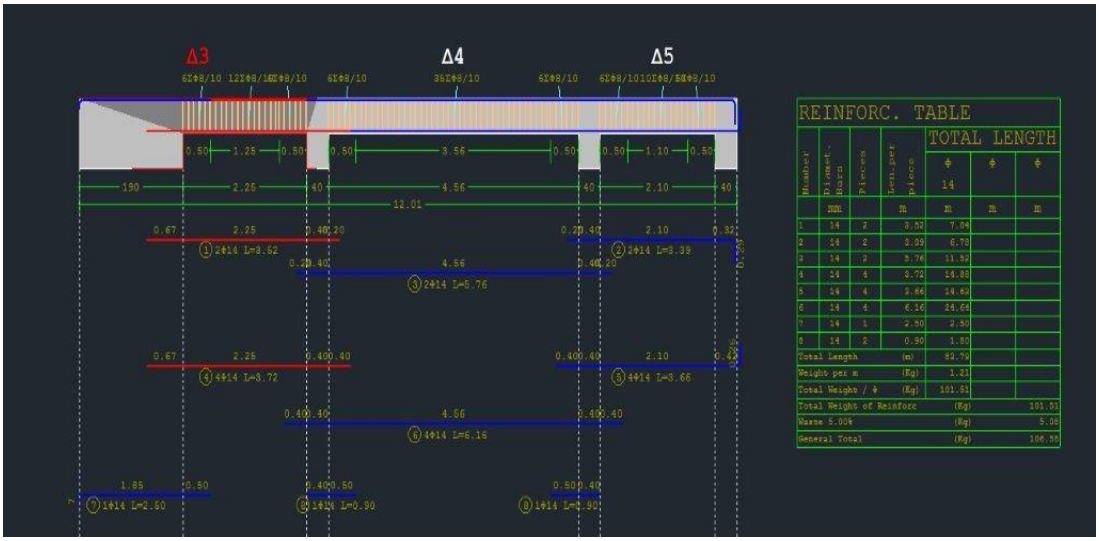
Select the first span. In the selected span the main steel reinforcement and the symbol of the beam turn into red.

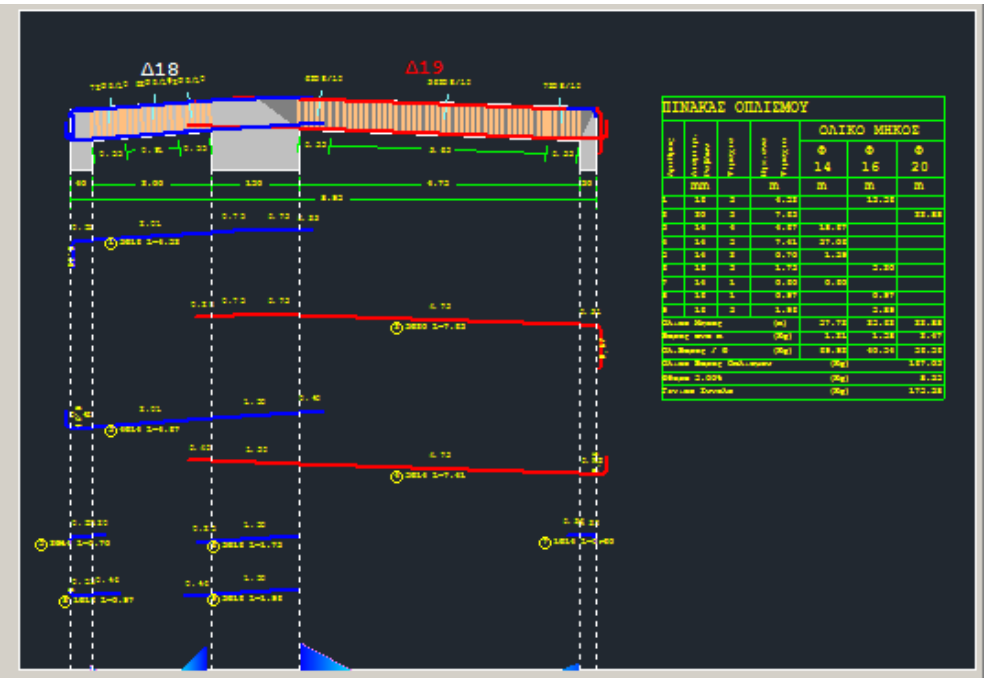


The "Span" area is shown on the right where the beam's data are filled in. You can modify the dimensions b, h of the beam's cross-section as well as the length and the height, according to the following design.





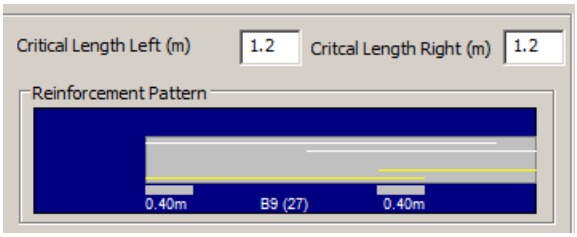




ATTENTION:

Any geometrical modification concerns the design only and does not inform the model calculations.

The “Span” area includes the critical lengths as well, which you can edit and the drawing and the “Reinforcement Pattern” will be updated automatically.

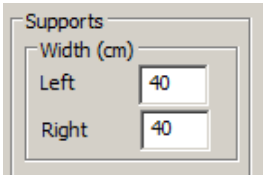


In the picture below notice that the bars on the top, coming from the respective spans, enter the corresponding spans as opposed to the bars on the bottom.

It means that in the calculation of the support reinforcement, the program will consider the top two bars for both left and right span reinforcement, while at the bottom one bar for each span will be considered.

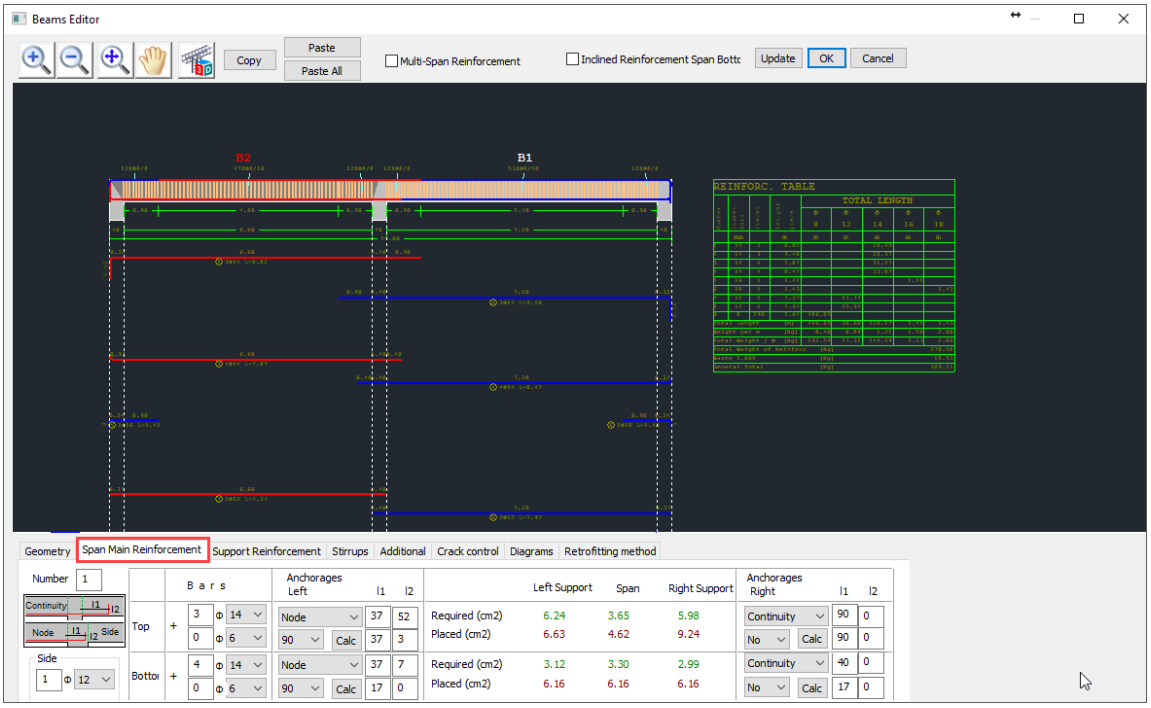
If you want to take into account the bottom bars of both spans, select the yellow lines and then left click (to select the left yellow line, select the left span and for the right yellow line, select the right span) to extend them. So the program will take into account both bars, for top and bottom.

If you want to take into account the bars of one span only, reselect the yellow and the white lines, to produce the form shown in the figure on the left.

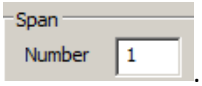


In the “Supports” field you can change the width of the supports on the left and the right, but remember that all geometric modifications concern only the drawing and don’t update the model’s mathematical calculations.

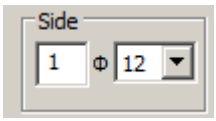
2. Span Main Reinforcement



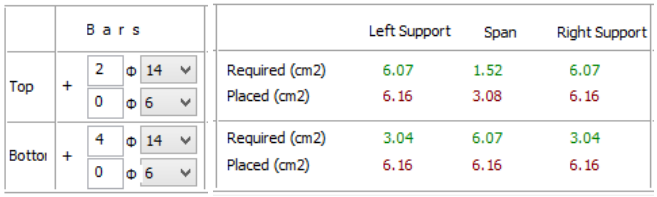
The "Span Main Reinforcement" unit consists of tools useful for the modification of the main steel reinforcement of the selected span.

You can choose the span graphically by left clicking on the drawing or by typing directly the number in the following field 

.

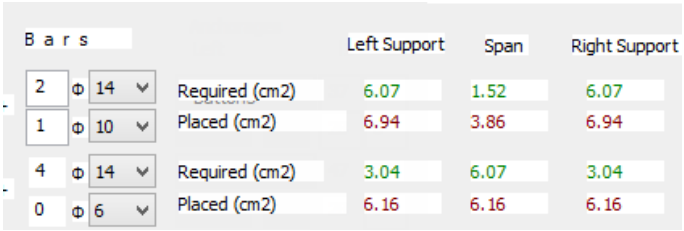


In the "Side" field there are the number and the diameter parameters of the side bars, where you can modify both parameters.



2.1 Bars

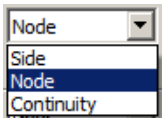
In the "**Bars**" field you define the number and the diameter of the main rebar of the beam at top and bottom and you can modify both of them. You may as well read the rebar area of the required and the placed reinforcement.
The square centimeters of the placed reinforcement are updated automatically according to the user's choice. You can change the number and the diameter or insert two different rebars on the top or bottom.



If the area of provided reinforcement is less or equal to the required, the number of placed square centimeters appears in the table magnified and colored red.

2.2 Anchorages

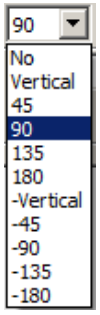
After you define the main steel reinforcement you must calculate the anchorages.

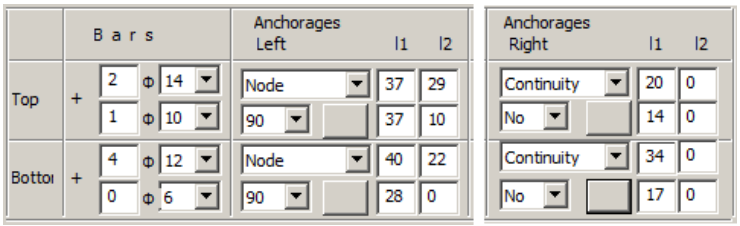
First choose the extension limit from the drop-down list 

according to the following drawing for the calculation of the L_1, on left and right support, separately.

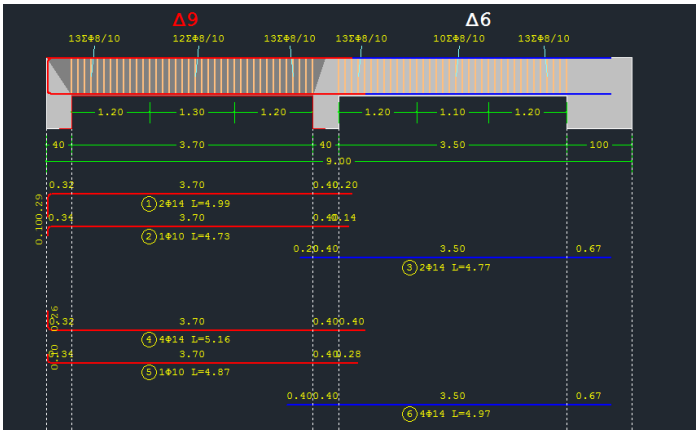
Then select the angle for the calculation of the L_2 (for the side, $L_2 = 0$), on the left support and the right support, separately.

For the calculation of the L_1 and L_2 click the following button  . The program considers all the imposed parameters and the position of the rebar and fills in the following table.





The drawing and the table are updated automatically considering any performed change.



Anchorage length lbd calculation process

The total lbd is calculated and divided into l1 and l2. L1 is the linear anchorage length, and l2 is rotated into the node.

NODE:

EC2 instead of providing a minimum linear anchorage length 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 the DCH the anchorage length to be linear only(exaggerating). According to the above:

1. For the EC2 w / o scenario and all EC with DCL and DCM ductility categories, EC8 does not obey to the minimum linear anchorage length of lb, min but the total length lbd is compared to lb, min σ according to 8.4.4 of the EC2. Therefore, there will never be an error message because, because in case that the anchorage length is greater than the width of the support minus the cover, the rebar will reach the alignment and then return to the node.
2. For the EC with a high ductility, EC8 obeys obeys to the minimum linear anchorage length according to EC8's paragraph 5.6.2.

IMPORTANT NOTE:

In the implementation process of **single beams reinforcement with common rebars of a specific length**, the program places the additional rebars in the supports under criteria.

There are two ways to place the additional supports reinforcement.

- The first way is additional rebars coming from each aperture on either side and positioned on the respective side of the opening.
- The second way is to place a common support rebar.

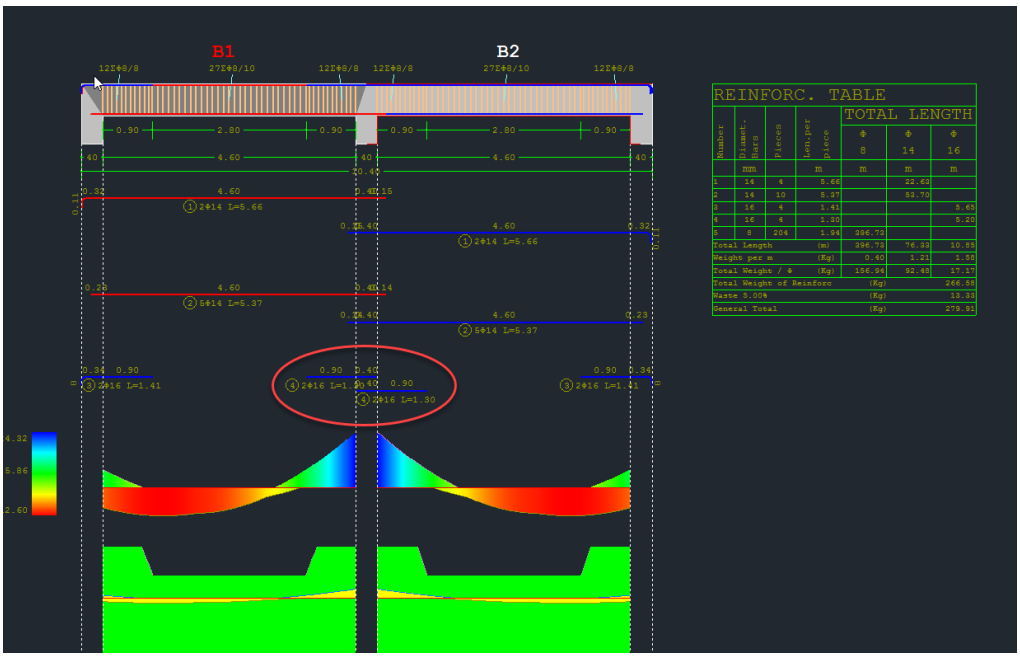


Figure 1

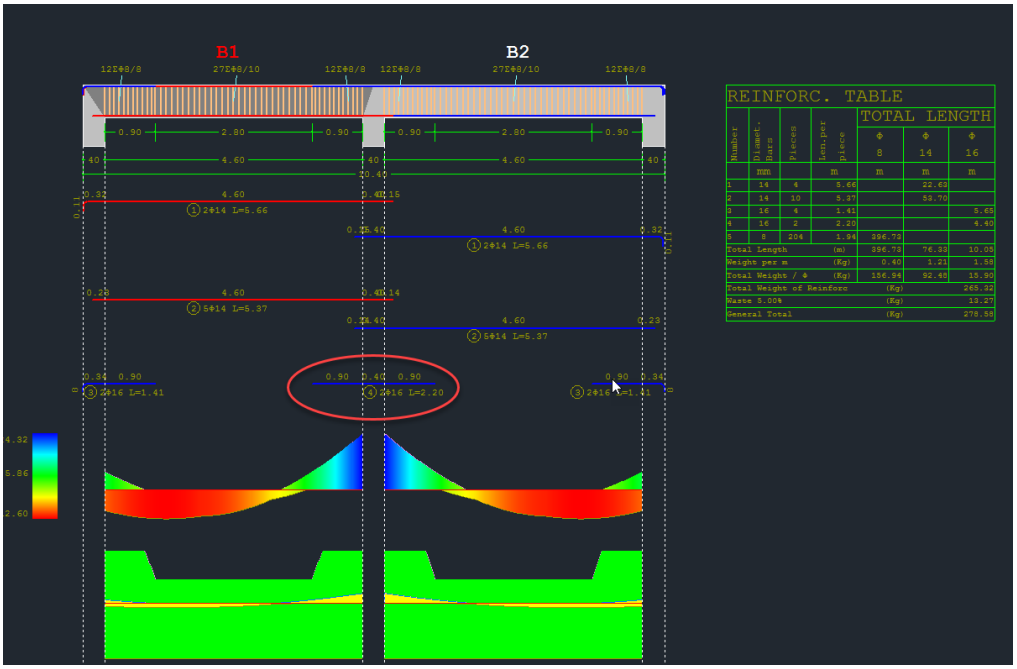
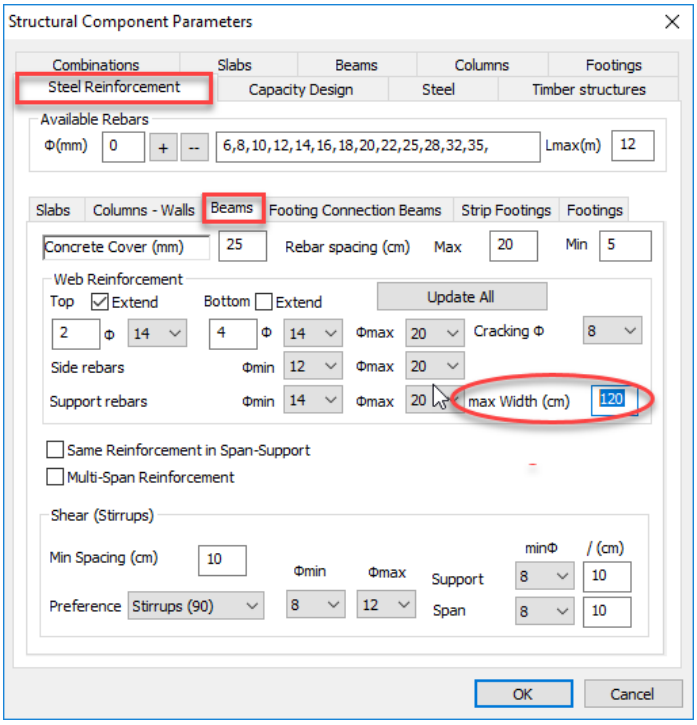


Figure 2

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 is placed separately per side.
If the width of the support is less than max width, then a common rebar to all the support is placed (see Figure 2).

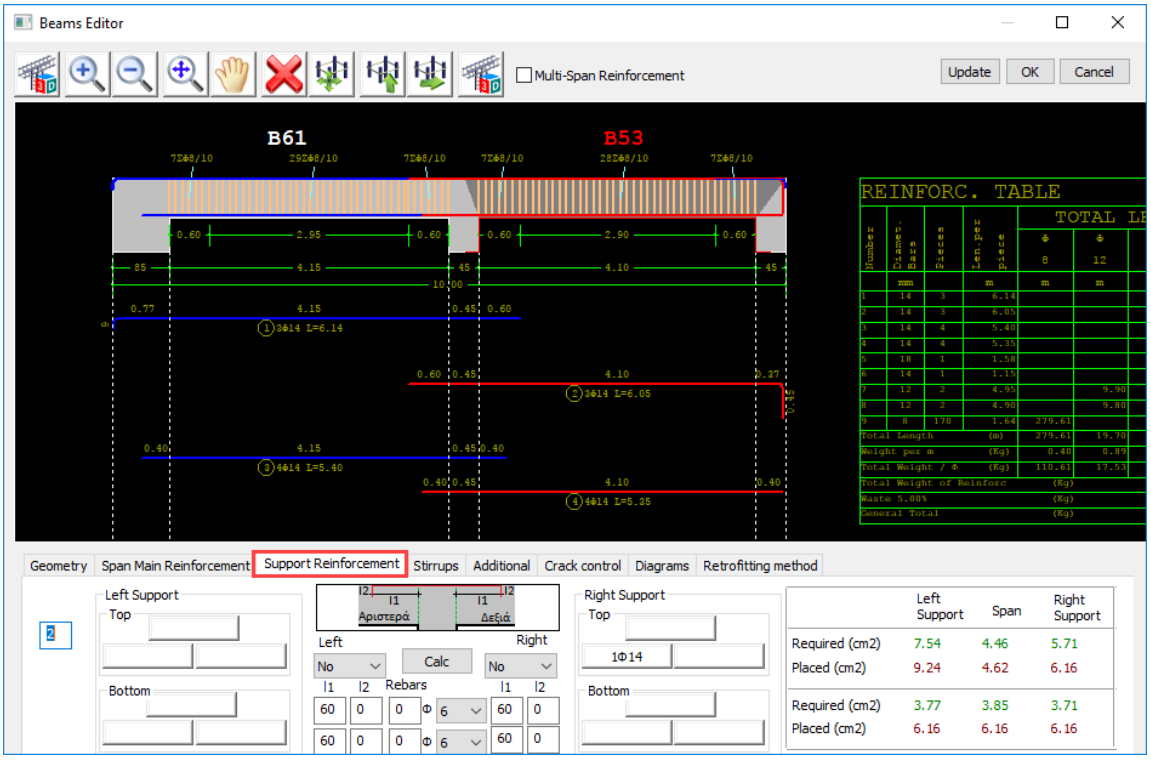
NOTE:
Changing this parameter after the creation of the continuity of the beams requires deletion and re-creation.

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 placed separately additional support rebars. Otherwise, place a common rebar.

CONCLUSION:
Common rebar placed only if satisfied both criteria:

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

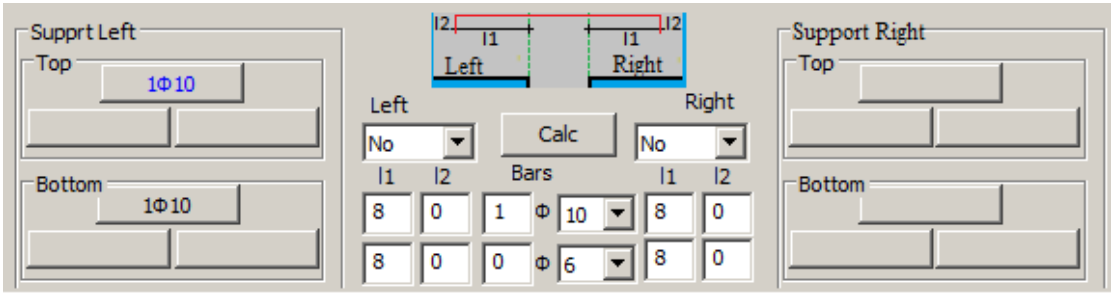
3. Support Reinforcement



The "Support Reinforcement" tab contains tools useful for the modification of the steel reinforcement of the supports of the selected beams.

Choose the span graphically by left clicking on the drawing or by typing directly the number in the following field 

.

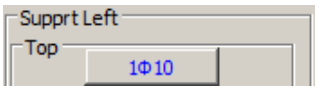


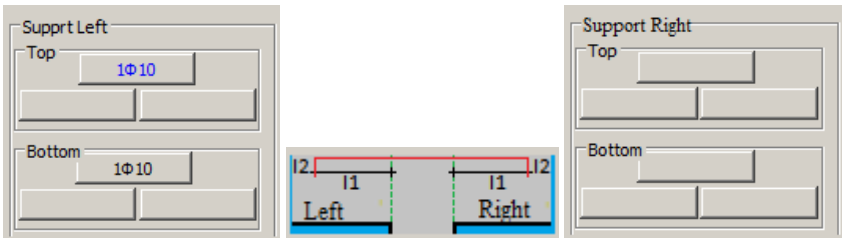
There are two support areas, Left Support and Right Support.
Each one is divided into Top and Bottom, which means Top support rebar and Bottom support rebar, respectively, and contains three tabs that represent the position of the rebar to the support:

3.1 Additional support rebar

1. Common to both spans
2. Only in the left span
3. Only in the right span

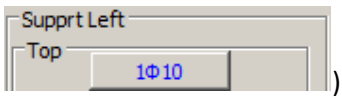
If there is no additional rebar to the support in the specific position, the tab is empty, otherwise, the tab indicates the number and the diameter of the bar in the relative position (e.g. 1Φ10 on the top, common, in the left support).



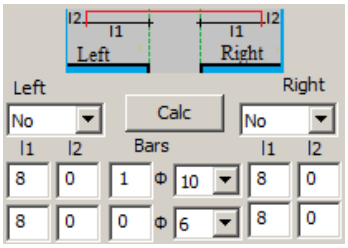


When the buttons are empty it means that there are no additional bars in the respective positions.

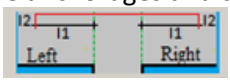
⚠ *To modify an existing additional bar or to add a new one, on the top or bottom, on the left or right support:*

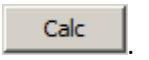
1. First, click on the respective tab (e.g. 

)

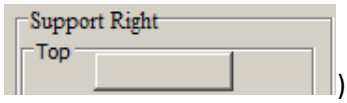
2. Update the following table 

with the characteristics of the support reinforcement in the specific position,

3. Finally, you can:
   - Change the number and the diameter of the rebar
   - Insert more bars
   - Define the anchorages and the angle, and automatically calculate l_1 and l_2 on the left and the right 

by using the button "Calc" 

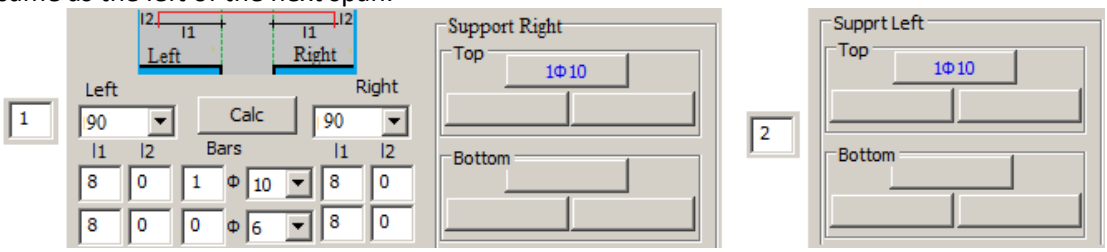
.

4. Click on an empty tab (e.g. 

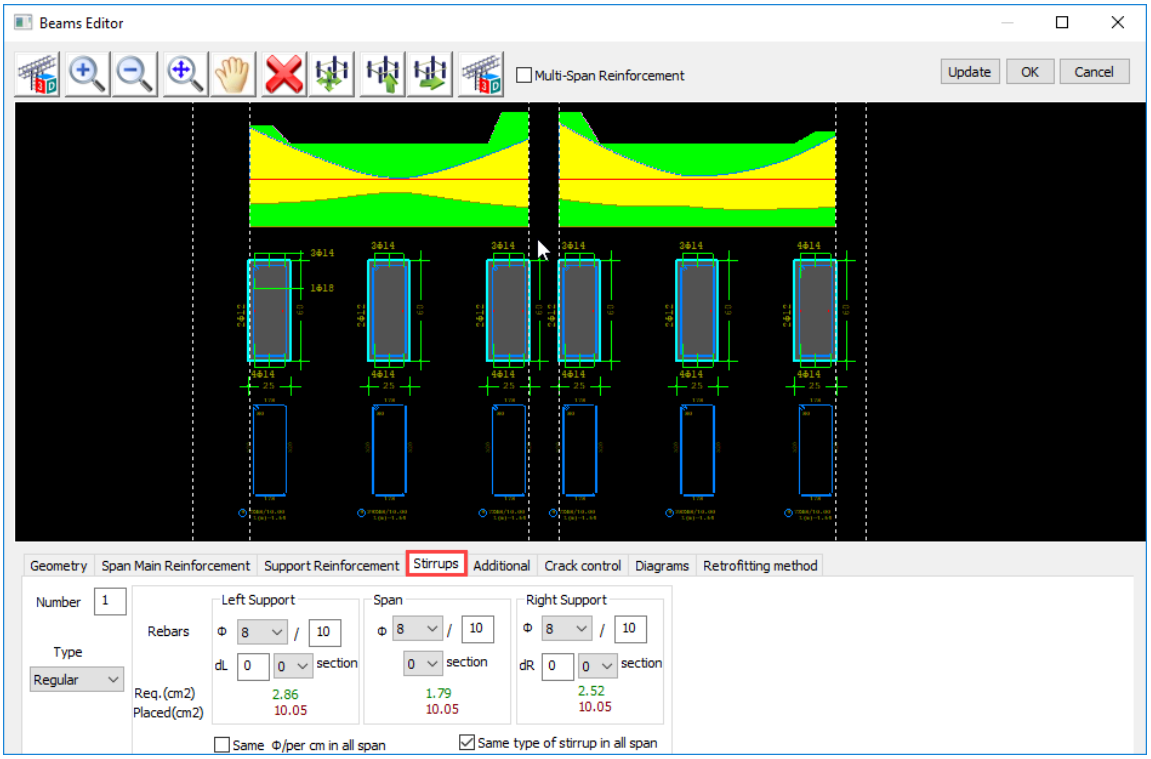
) and insert an additional support rebar on the specific position, following the same procedure.

NOTE:

In a continuity of beams, for the internal supports, the right support of the previous span is the same as the left of the next span.

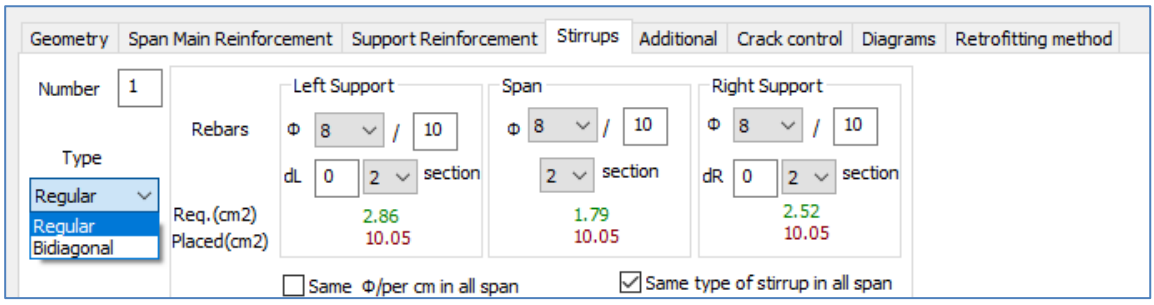


4. Stirrups

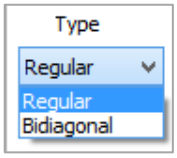


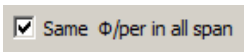
The "Stirrups" tab contains useful tools for the modification or the addition of stirrups on spans and supports of the selected beam.

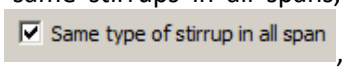
Choose the span graphically by left clicking on the design or by typing directly the number in the following field  to update the corresponding area:



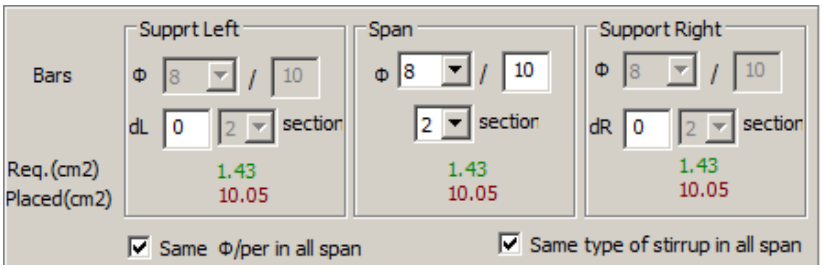
EXAMPLE:

The span with the number "2", contains only one type of stirrups , "Normal" stirrups  , with "2 sections" and "Placed" area greater than the "Required" area.
You can modify the existing stirrups or insert new ones.

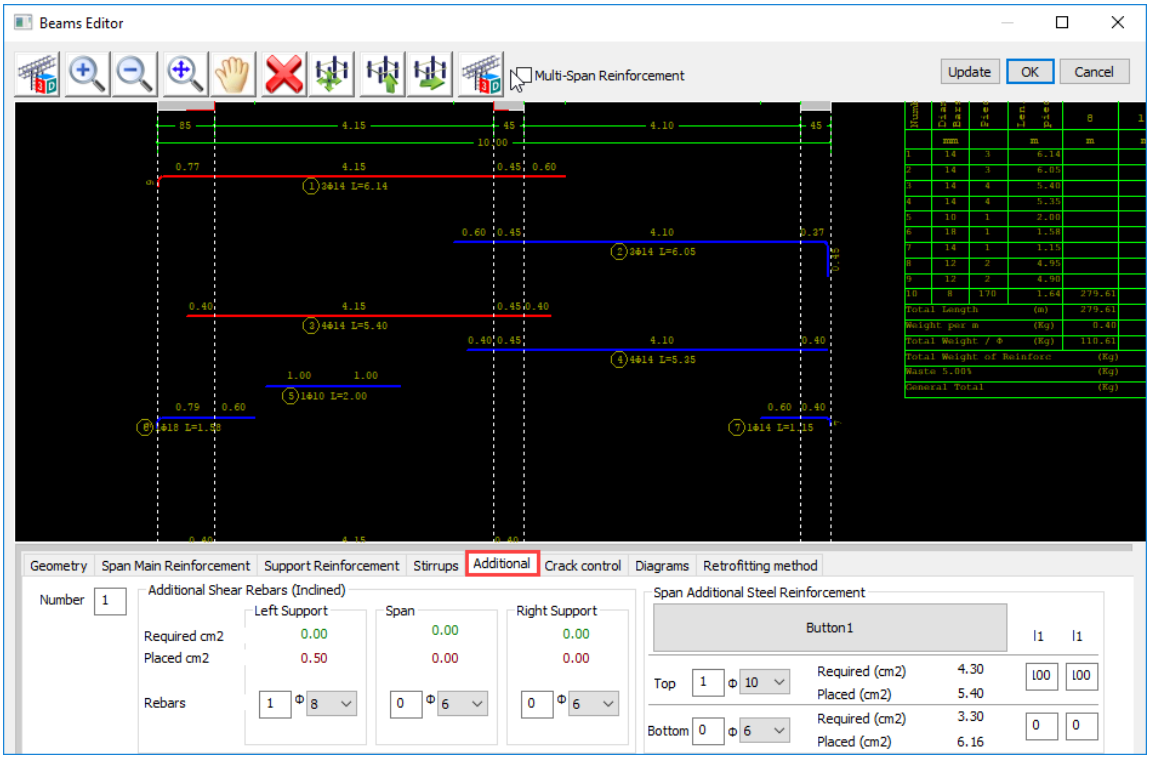
If you decide to use the same stirrups in all spans, activate the corresponding checkboxes 

and 

, and type the characteristics in the span and the same are applied in the supports:

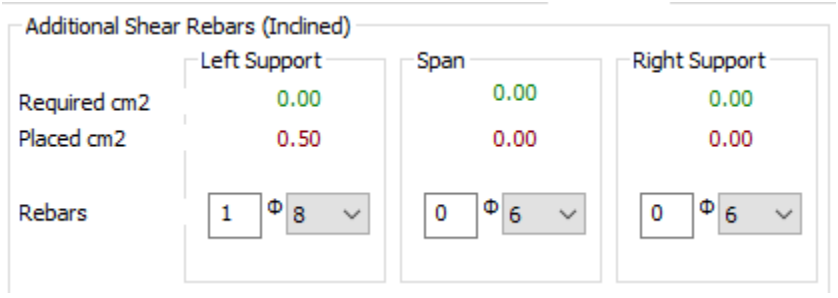


5. Additional



The "Additional" tab contains useful tools to modify or add additional rebar for shear or bending resistance, on spans and supports of the selected beam.
Choose the span graphically by left clicking on the drawing or by typing the number directly in the following field  to update the corresponding area:

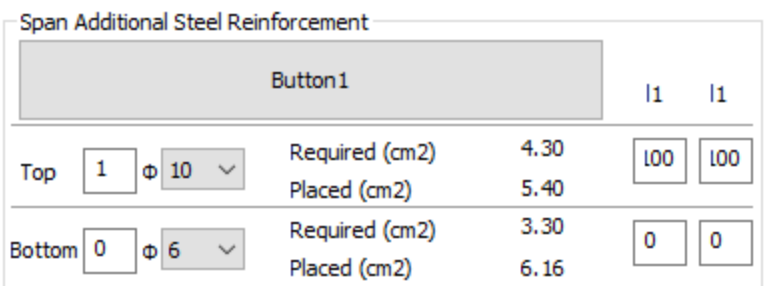
5.1 Additional bars for Shear effects (Inclined)



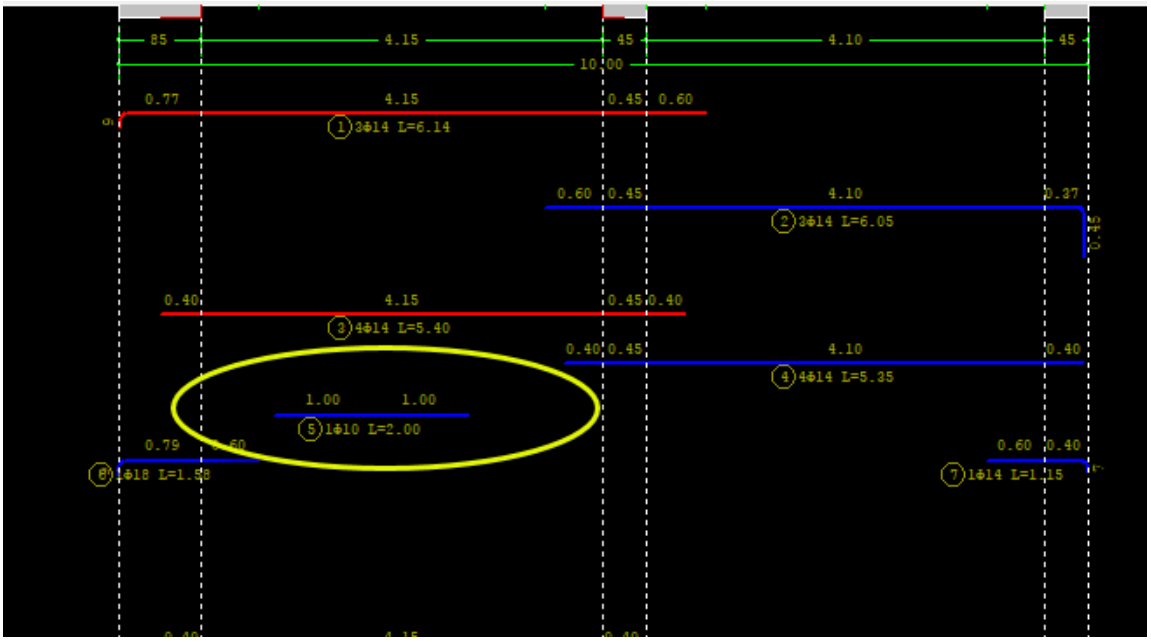
The field "Shear Additional" is automatically updated considering the required and placed steel reinforcement for shear effects.

You can intervene by changing the number and the diameter in both supports and span. Placed reinforcement is automatically updated.

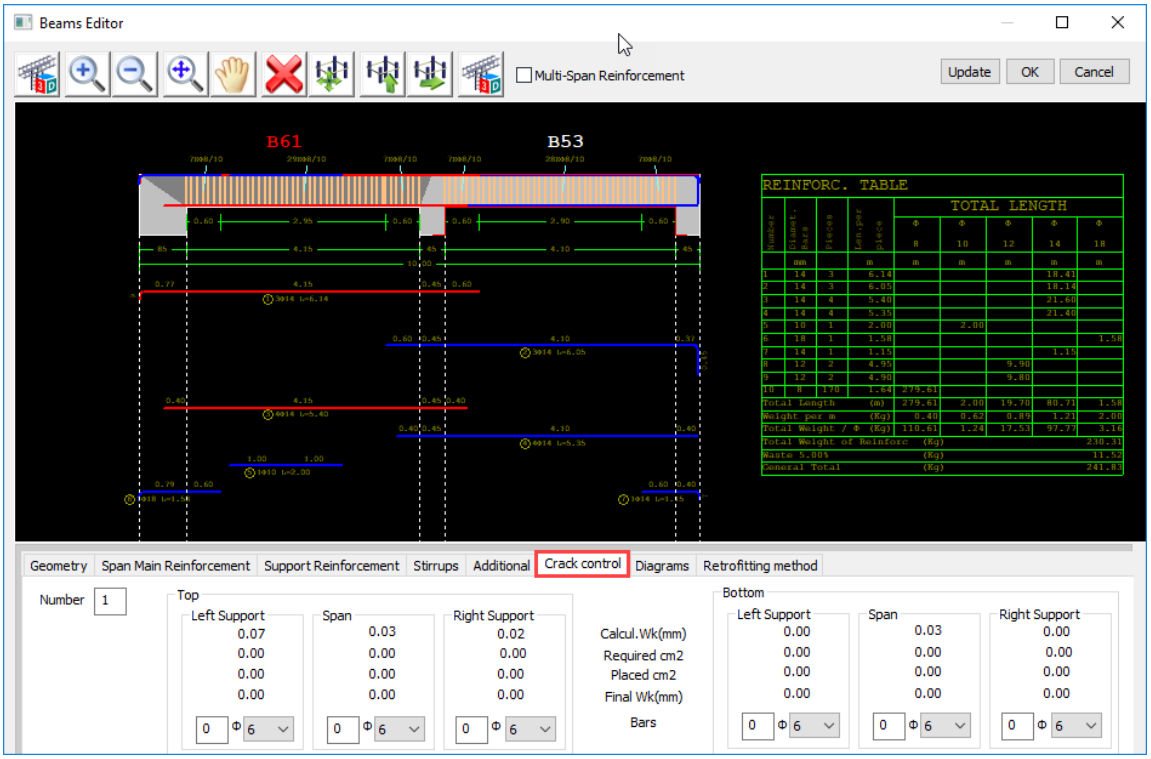
5.2 Additional bars for bending effects



The “Additional Span Reinforcement” field is automatically updated considering the required and placed steel reinforcement for bending effects, in the top and bottom.

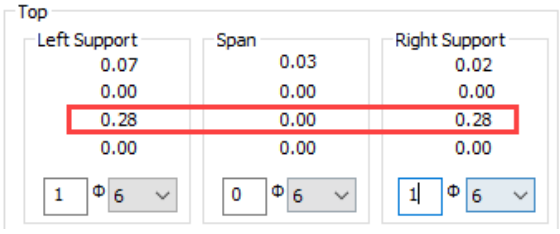


6. Crack control

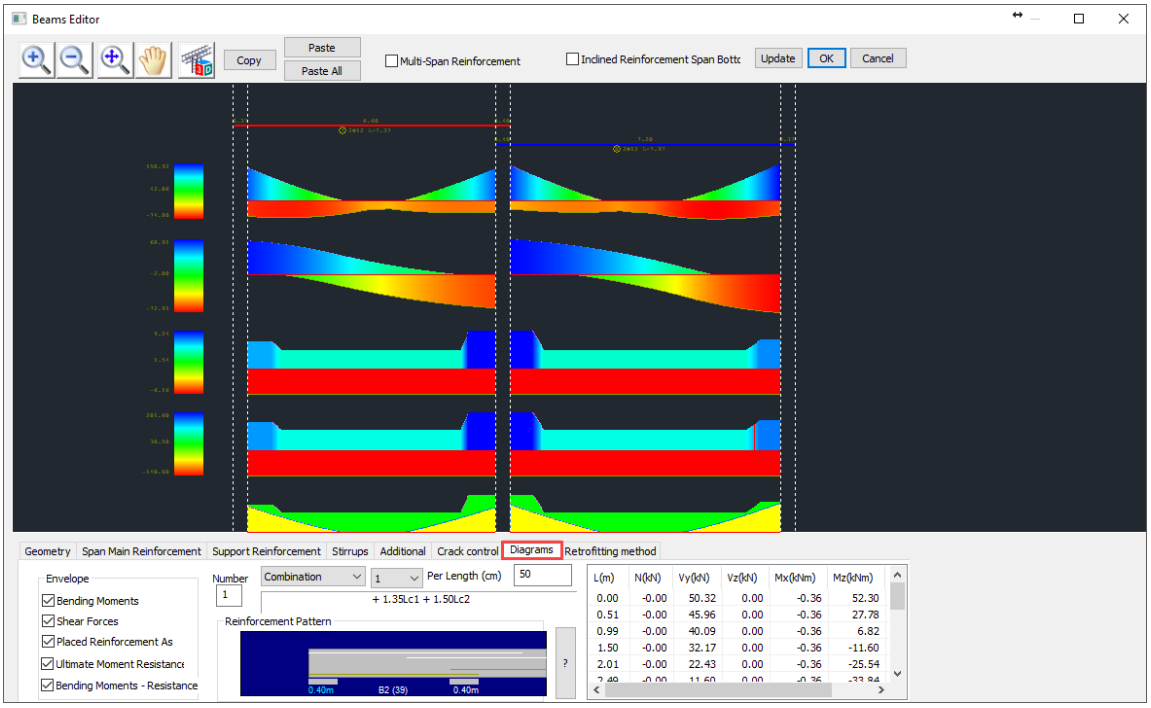


"Crack control" unit contains tools useful for the modification or addition of rebar against the concrete cracking, on spans and supports of the selected beam, on the top and bottom.
Choose the span graphically by left clicking on the design or by typing directly the number in the following field  to update the corresponding area:

The calculated W_k, the required and the placed steel reinforcement, the final W_k and the characteristics of the placed rebar in the correspondent position are updated automatically.
You can intervene by changing the number and the diameter in both supports and the span, on the top and the bottom. Placed steel reinforcement is automatically updated.



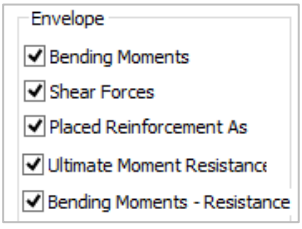
7. Diagrams



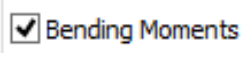
In the "Diagrams" unit you can find useful information about the:

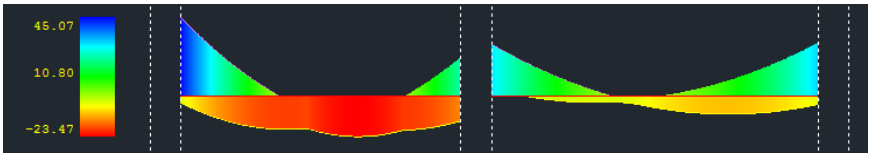
- Envelope diagrams of Bending Moments, Shear Forces, Placed Reinforcement, Ultimate Moment Resistance, Bending Moments – Resistance of all spans.
- Internal forces for each load and each combination, by specifying the path ("Per Length").

7.1 Diagrams

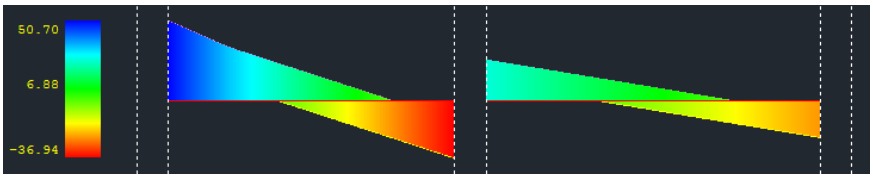


Activate a checkbox so that the corresponding diagram to be displayed on the cad interface:

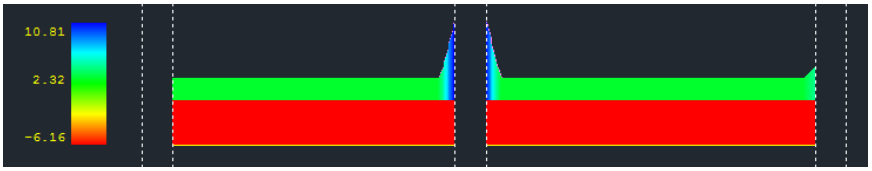
1. 



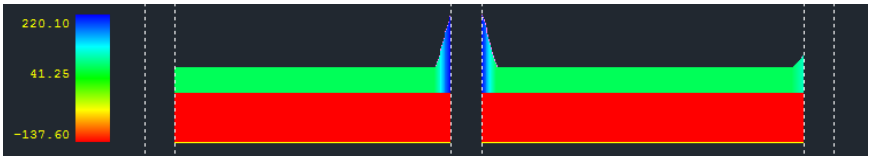
2. ☑ Shear Forces



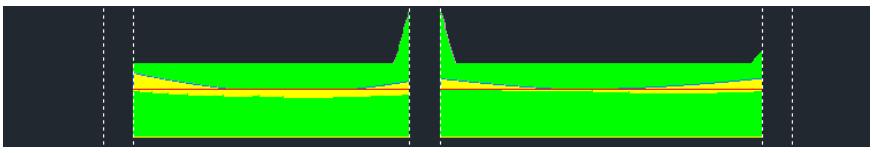
3. ☑ Placed Reinforcement As



4. ☑ Ultimate Moment Resistance



5. ☑ Bending Moments - Resistance

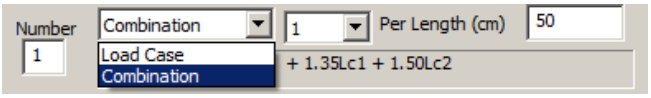


The color bar on the left helps you to easily find the corresponding values of the diagrams.

7.2 Internal Forces

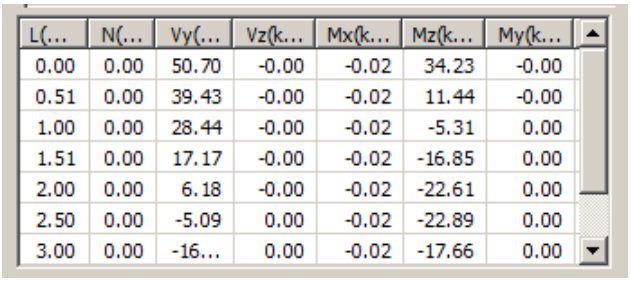
Choose the span graphically by left clicking on the drawing or by typing directly the number in the “Number” field 

.

In the following area 

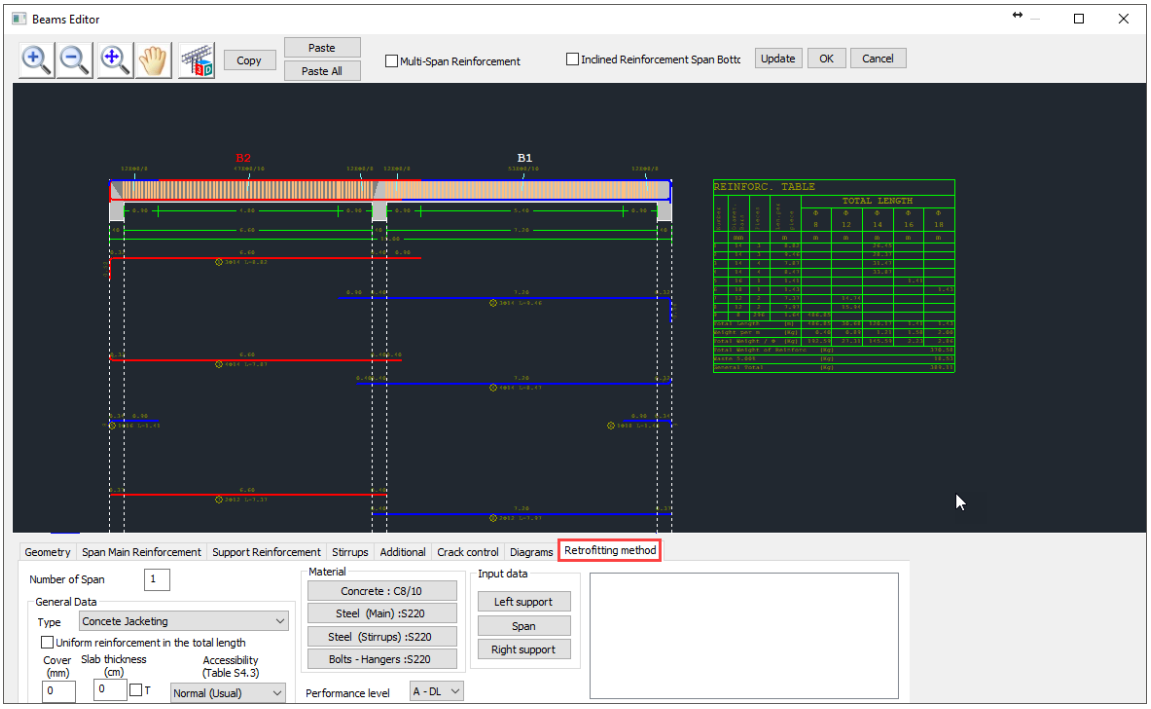
choose the load case or the load combination and the corresponding number, and type the length of the beam where the inertial forces will be calculated.

You can read the values of all the internal forces, calculated for the specific Load Case or Load Combination, in the corresponding length in the table below.



L(...	N(...	Vy(...	Vz(k...	Mx(k...	Mz(k...	My(k...
0.00	0.00	50.70	-0.00	-0.02	34.23	-0.00
0.51	0.00	39.43	-0.00	-0.02	11.44	-0.00
1.00	0.00	28.44	-0.00	-0.02	-5.31	0.00
1.51	0.00	17.17	-0.00	-0.02	-16.85	0.00
2.00	0.00	6.18	-0.00	-0.02	-22.61	0.00
2.50	0.00	-5.09	0.00	-0.02	-22.89	0.00
3.00	0.00	-16...	0.00	-0.02	-17.66	0.00

8. Retrofitting methods



The “Retrofitting methods” section contains the tool for the seismic reinforcement and rehabilitation of the beams according to the Code of Structural Interventions.

First, the calculated steel reinforcement is adjusted to the existing one, then you proceed with the definition of the structural reinforcing techniques.

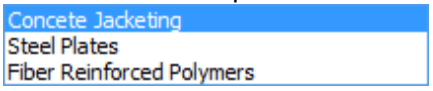
Select the span graphically or by left click on the span in the cad interface. Otherwise, set the number of the span in the “Number of Span” field 

.

In the “Beams editor”, the beam is always depicted by the input step direction. To quickly locate the beam of interest among the rest structural elements, the numbering and the visibility of the local axes for all beams is suggested. You can select the corresponding beam by setting its serial number in the editor utility. To define the left and right support of the beam, use the direction of the local axis xx’. The start and the end of the beam element in the editor’s interface are defined concerning the local axis xx’ direction, without considering the beam’s orientation in the plan view.

“General Data”

Select from the drop-down list the structural reinforcing method applied to the selected beam.



1. The activation of the "Uniform reinforcement in the total length" checkbox/means that the reinforcement, applied in each critical cross-section, will be designed (supports, span) considering the less favorable values of the inertial forces. The less favorable inertial forces are derived from the comparison of the inertial forces in all cross-sections of the beam. Otherwise, if the checkbox remains inactive, the inertial forces of each critical region examined will be used for the design of the corresponding retrofitting method.

2. In both options above, the structural reinforcement must be applied in the three critical sections of the beam (left/right support and span).

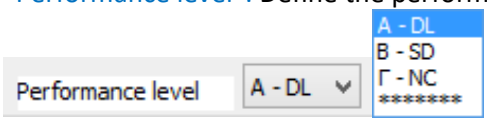
3. In case of the concrete jacket and additional concrete layers, type a value in the field "Cover (mm)" to define the concrete cover of the cross-section.

4. Activate the checkbox "T" 

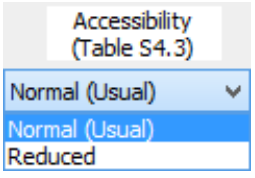
and type the thickness of the plate in the "Plate thickness (cm)" field in case of a T-shaped cross-section. For a Γ-shaped cross-section, type the thickness of the plate without activating the checkbox "T".

5. For zero value as plate thickness, a rectangular cross-section will be considered, no matter of the checkbox "T" status.

"Performance level": Define the performance level of the structure.

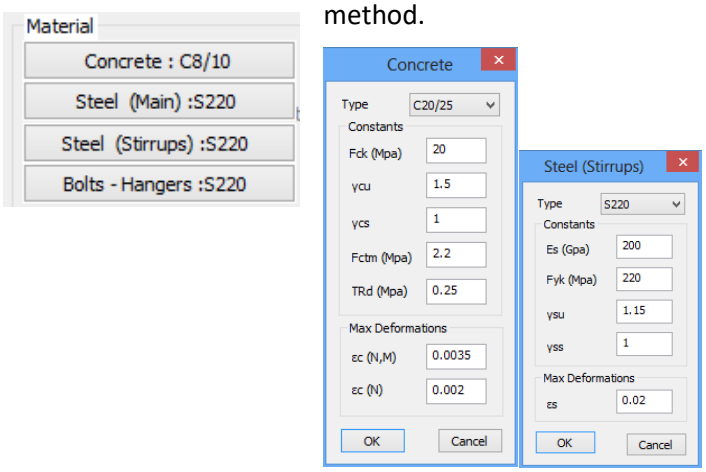


"Accessibility": Define the accesibilty of the area where the reinforcement is applied according to the § 4.5.3.2 of the Structural Interventions Code.

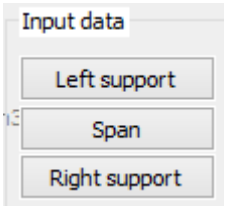


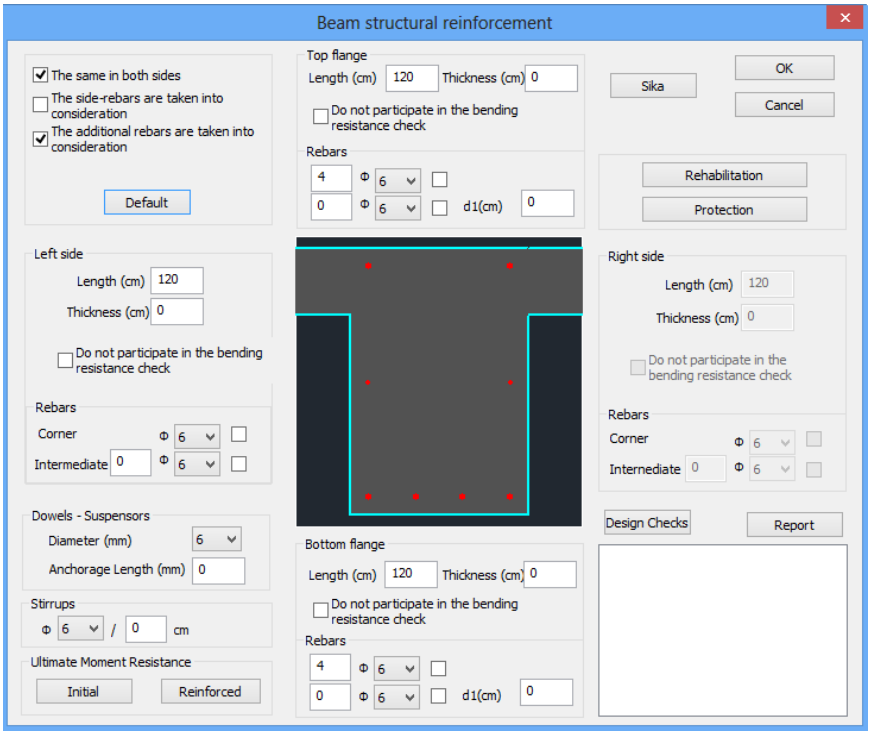
8.1 Additional concrete layers – concrete jackets

"Materials": Select the type of material for each structural component of the reinforcing method.



"Input Data": Define the input data of the concrete jacket in the two supports and the span of the beam.

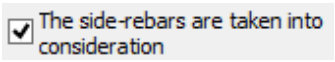




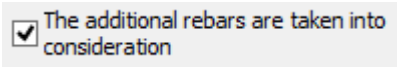
In the dialog box "Beam structural reinforcement" that appears, you add the concrete jacket per side (top, bottom, left the side, right side). There is also a preview window, part of the dialog box, where the settings are depicted.

The calculations and the design checks of the beam are the same with the checks of the column presented in the relative chapter.

For the existing side rebars to be taken into consideration, in the calculation of the ultimate moment resistance of the cross-section, you activate the following checkbox:

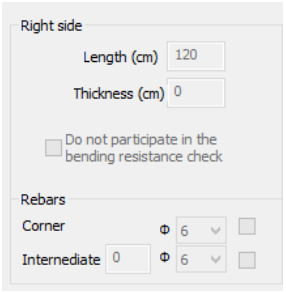


For the existing additional steel reinforcement in the supports of the beam to be taken into consideration, in the calculation of the ultimate moment resistance of the cross-section, you activate the following checkbox:



If you activate the following checkbox 

then, the "Right Side" field is deactivated and receives the same input parameters as they are defined in the "Left Side" field.



If you select the button "Default", then, the corresponding length of the supports and the span are automatically defined. The support's length is the critical length and the span's length is the total length of the beam minus the critical of the two supports. The calculated lengths correspond to the reinforcing length.

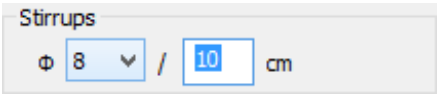
In the "Ultimate Moments Resistance" field, if you select the button "Initial" or "Reinforced", then the ultimate bending resistance of the existing or the reinforced cross-section, respectively, is calculated.

NOTES:

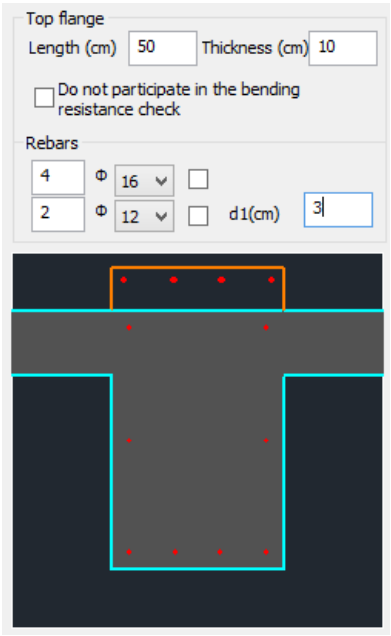
- *Beams reinforcement offers two options concerning the calculation of the Ultimate Moment Resistant: "Initial" and "Reinforced". Press the "Reinforced" button to calculate the new M-N diagrams and update also the relative λ ratios.*
- *When you finish with the reinforcement, go back to the Analysis field and open the Checks, to update also the print outs with the new λ ratios.*

In the "Dowels - Suspensors" field you define the input parameters of the dowels or the suspensors; diameter and anchorage length.

In the "Stirrups" field, set the diameter and the spacing of the stirrups of the concrete jacket.



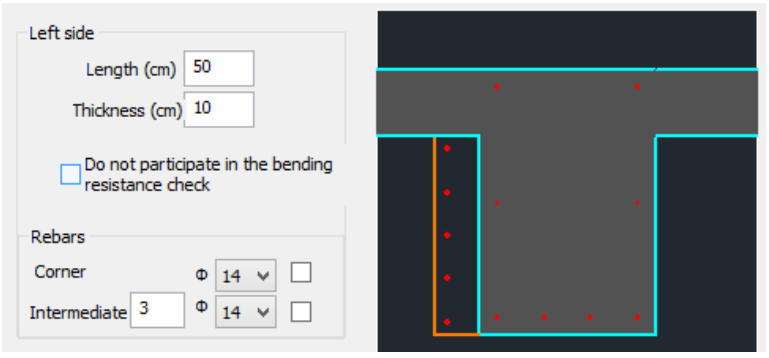
Input parameters in the fields "Top" and "Bottom"



If you activate the "Not considered in the bending resistance check" checkbox, then the corresponding concrete jacket or the additional concrete layer are not taken into consideration in the calculation process of the reinforced cross-section.
In the first line of the input parameters of the steel reinforcement, you define the number of the rebars of the first/basic rebars' layer and their diameter. For more than one layers of rebars, you define in the second line, the number of the rebar layers, the diameter of the rebars and the spacing d_1 between the rebar layers.
The rebar layers have always two rebars.
The activation of the checkbox, on the right of the diameter's drop-down list 

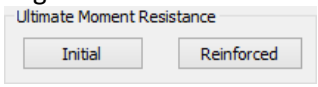
, means that the corresponding steel reinforcement is not taken into account in the calculation of the ultimate moment resistance.

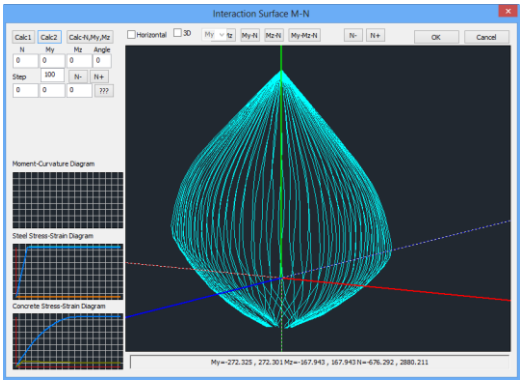
Input parameters in the fields "Left Side" and "Right Side"

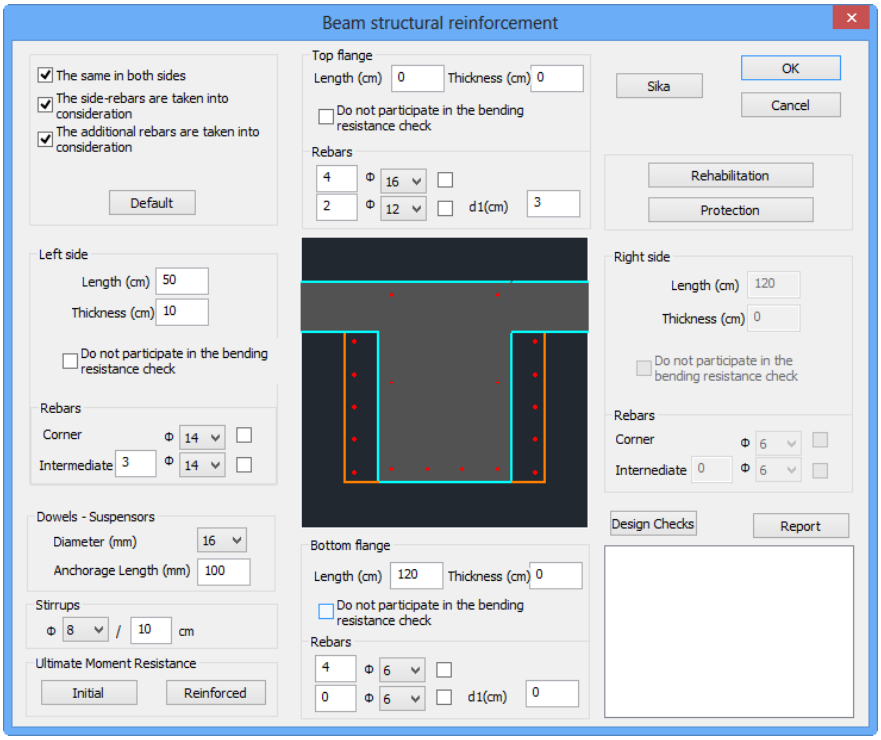


In the "Rebars" field you define the diameter of the corner rebars of the corresponding side, as well as, the number and the diameter of the intermediate rebars. The rest parameters are the same as the parameters of the top and bottom of the cross-section.

In the "Ultimate Moment Resistance" field, select the "Initial" of "Reinforced" button. Then, a new dialog box opens, where the interaction diagram of the uniaxial bending moment and the axial force is depicted for the existing and the reinforced cross-section, respectively.

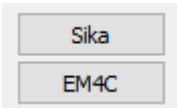






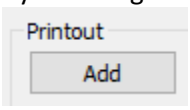
Select the "Design checks" command and the program performs all the design checks for the concrete jacket in every reinforced side with a concrete jacket (according to the Code of Structural Interventions) and so the appropriate number of dowels is calculated. The design checks and their corresponding results are similar to those considered for the columns.
The results of the design checks are presented on the bottom of the window.
Finally, select the "Report" command and the results of the design checks will be added in the corresponding chapter of the Report.

⚠ *After any modification on the parameters of the concrete jacket, select the command "Report", so that the report will be updated with the new data as appropriate.*

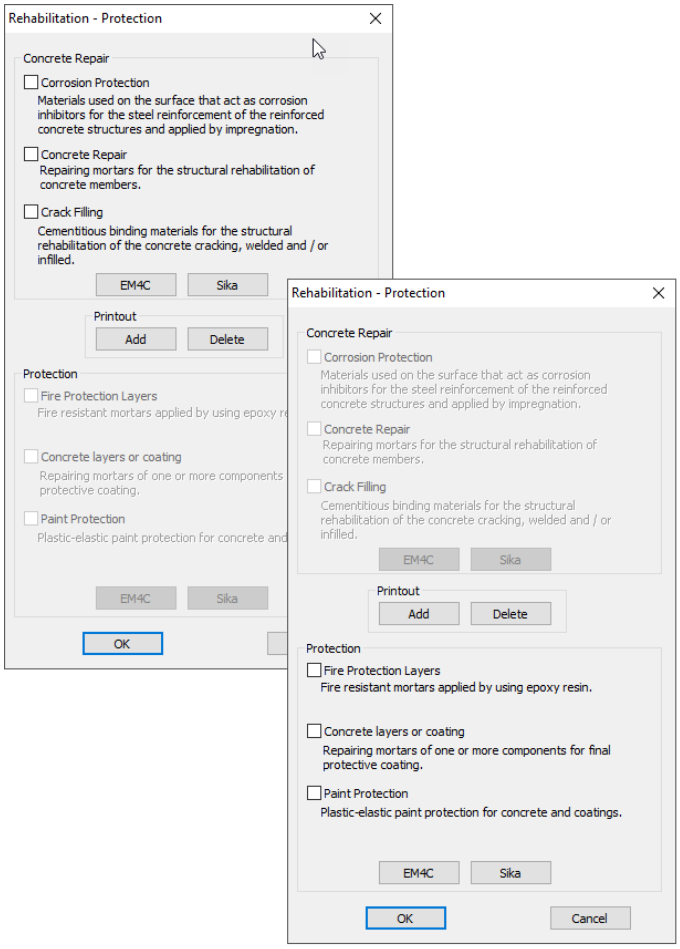
Furthermore, in SCADA Pro, the techniques and the material considered in each retrofitting method are enriched with the corresponding material and techniques of the company Sika and EM4C. The user has direct access to the library of Sika and/or EM4C materials by pressing the relative button 

, which appears in the dialog boxes related to beam reinforcement. The buttons "Rehabilitation" and "Protection" correspond to a dialog box each with tools related to the rehabilitation and protection of the beam structural elements according to the Code of Structural Interventions. (ΚΑΝ.ΕΠΕ).

The user can select one of the three rehabilitation and protection methods by activating the corresponding checkbox.

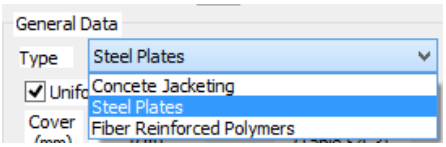
By selecting the command 

the corresponding information will be imported in the final report of the study.

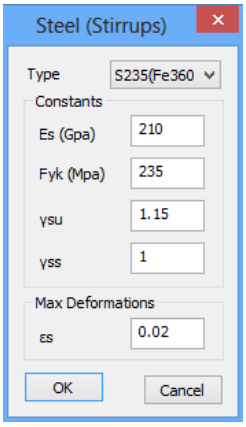


8.2 Steel laminates – fiber reinforcement polymers (FRP)

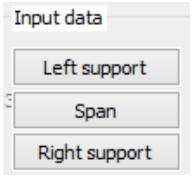
For these two retrofitting methods, the same procedure is followed as previously.
For the steel laminates or FRPs, select the corresponding layer from the drop-down list.



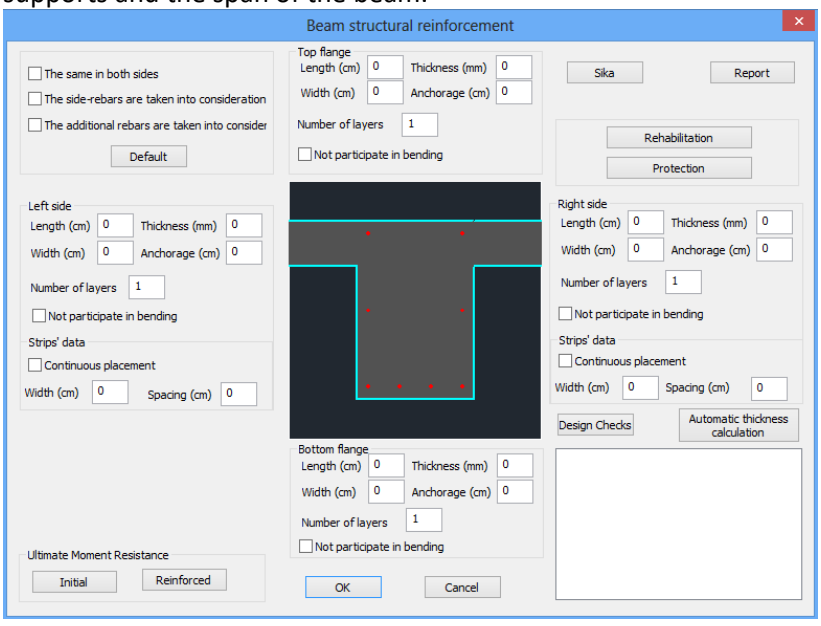
"Material": Select the type of the steel for the laminates and the fiber reinforced polymers.



The performance objective and the accessibility level are defined following the same procedure as for the definition of the concrete jacket.

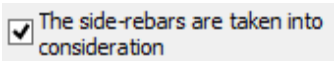


"Input data": Set the input data for the "Steel Laminates" or the "FRPs" for the supports and the span of the beam.

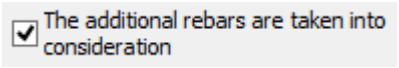


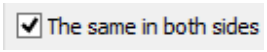
In the dialog box "Beam Reinforcement" that appears, the input data per side of the cross-section (top, bottom, left the side, right side) are defined. There is also a window in the dialog box, where the results of the design checks are summarized.
The calculations and the design checks of the beam are the same with the checks for the column presented in Chapter A.

For the existing side rebars to be taken into consideration in the calculation of the ultimate moment resistance of the cross-section, you activate the following checkbox:

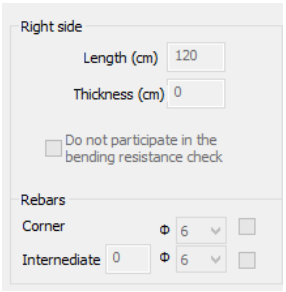


For the existing additional steel reinforcement in the supports of the beam to be taken into consideration in the calculation of the ultimate moment resistance of the cross-section, you activate the following checkbox:

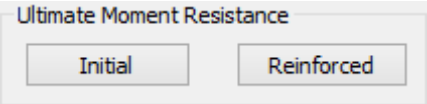


If you activate the following checkbox 

then, the "Right Side" field is deactivated and receives the same input parameters as they are defined in the "Left Side" field.

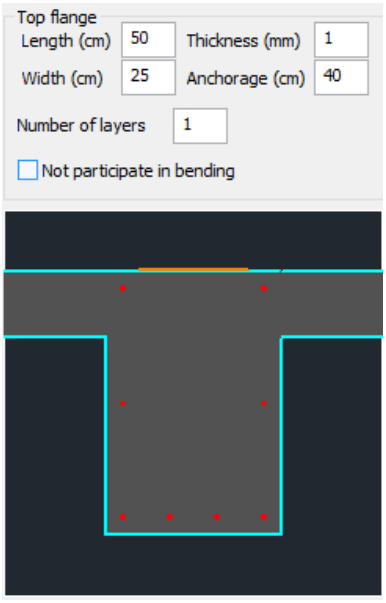


If you select the **"Default"** button, then, the corresponding length of the supports and the span are automatically defined. The support's length is the critical length and the span's length is the total length of the beam minus the critical of the two supports. The calculated lengths correspond to the reinforcing length.



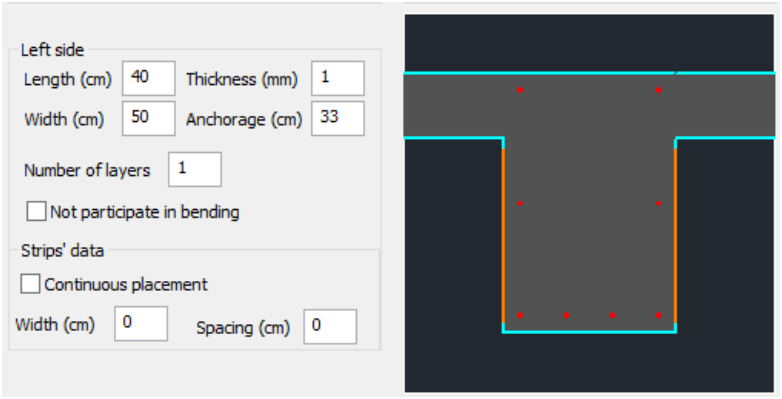
In the "Ultimate Moments Resistance" field, if you select the button "Initial" or "Reinforced", then the ultimate bending resistance is calculated of the existing or the reinforced cross-section, respectively.

Input parameters in the fields "Top" and "Bottom"



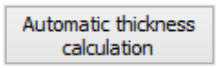
If you activate the "Not considered in the bending resistance check" checkbox, the corresponding concrete jacket or the additional concrete layer are not taken into consideration in the calculation process of the reinforced cross-section.
The length of the laminate is defined as the length of the concrete jacket. The width of the laminate is defined at first equal to the width of the corresponding side. The definition of the anchorage length is mandatory and the program at first sets a value by default and it can be modified by the user. The number of the layers is the number of the layers of the reinforcement.

Input parameters in the fields "Left Side" and "Right Side"



The definition of the reinforcement's geometry is defined with a similar way to the one of placing reinforcement on the top and the bottom of the cross-section. The checkbox about the non-consideration of the laminates of the left and right side in the bending resistance of the cross-section is already activated because they provide strength to the cross-section mainly against shear failure.

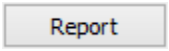
The placement of the laminates can be uniform or in strips; continuous or discontinuous with intermediate spacing. Activate the "Continuous Formulation" in the "Strips' data" field and define the width of the laminate. For considering the discontinuous formulation, deactivate the previous checkbox and define the spacing between the strips.

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 concerning the cross-section of the laminate and the type of the material. Then, the thickness t_1 and t_2 are calculated again concerning 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 calculates automatically the final minimum thickness t_2, which is presented in the window in the bottom of the dialog box. Then, you have to set this calculated value in the corresponding field and repeat the final design checks.

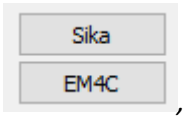
⚠ *The structural adequacy of the laminate or the FRP is reached with the increase of the thickness or the number of the layers*.



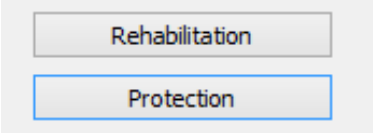
Finally, select the command "Report" and the results of the design checks will be added in the corresponding chapter of the Report.

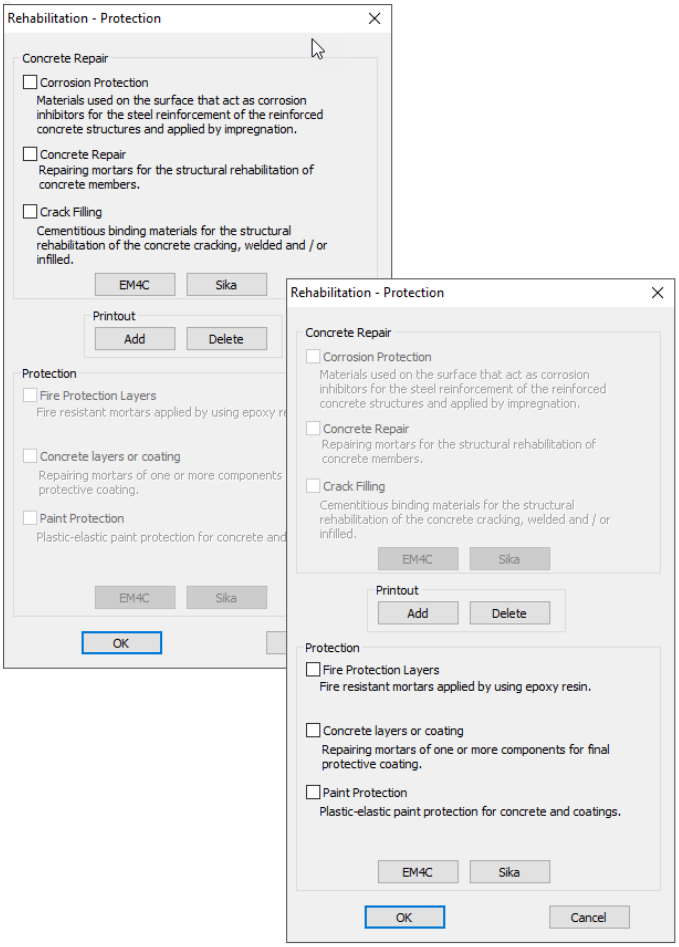
⚠ *For each modification in the steel laminate or the FRPs, repeat the selection of the command "Report" in order the final report of the study to be updated.*

NOTES:

Furthermore, in SCADA Pro, the techniques and the material considered in each retrofitting method are enriched with the corresponding material and techniques of the company Sika and EM4C. The user has direct access to the library of Sika and/or EM4C materials by pressing the relative button 

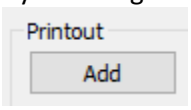
, which appears in the dialog boxes related to beam reinforcement. The buttons "Rehabilitation" and "Protection" correspond to a dialog box each with tools related to the rehabilitation and protection of the beam structural elements according to the Code of Structural Interventions.





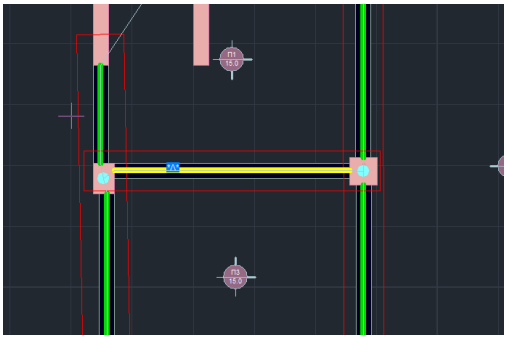
The user can select one of the three rehabilitation and protection methods by activating the corresponding checkbox.

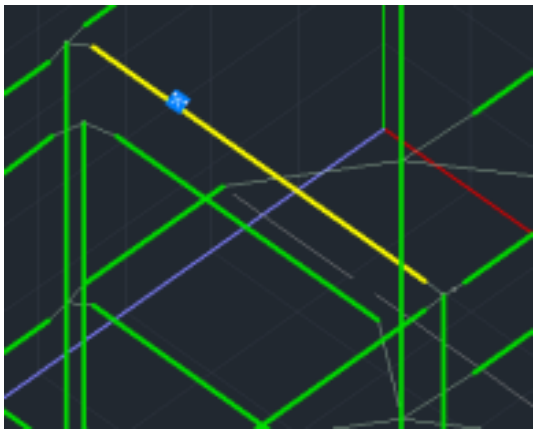
By selecting the command



the corresponding information will be imported in the final report of the study.

The reinforced beams are indicated on the screen like the reinforced columns:
The corresponding structural member is colored yellow:

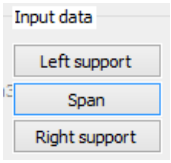




Also, according to the type of the reinforcing method, an indicative letter appears:

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

The following definition takes place according to the local axes of the beam (according to the direction that it was drawn: from left to right and vice versa). For that reason always activate the view of the local axes before the input of the reinforcing method.



⚠ *Prerequisite for the appearance of the label is that you have first selected the “Report” button in the dialog box of the corresponding beam reinforcing method.*