# SCADA Pro - WHAT'S NEW 2017

**Note**

ACE-HELLAS in the context of developing and optimizing its products, and in particular SCADA Pro, has created the new SCADA Pro 2017 with new enhanced capabilities.

Version SCADA Pro17
May 2017

## What's NEW in SCADA Pro 17?

1. **New way of calculating beam reinforcement. Reinforcing with common bars of a specific length.**
2. **More combinations for the slab's design.**
3. **New storage capability of dimensioning parameters.**
4. **Slabs Deflection Control according EC2.**
5. **Red mark on slab symbol when thickness is less than permitted by regulation**
6. **Members diagrams, nodes displacements, plate element internal forces, slab strip diagrams view.**
7. **New possibility of color display of intensive sizes on linear and surface elements, based on the sign.**
8. **New display capability of the deformation values based on the color gradation**
9. **Textile Reinforced Mortar(TRM) (ACI549.4R-13 and Triantafillou & Antonopoulos methods).**
10. **Load bearing masonry check based on Stress Failure Criterion.**
11. **Pehabilitation are enriched method of beams and columns, using EM4C and Sika materials.**
12. **3D DXF-DWG import. Automatic attribution of the cross sections on the drawing lines. Lines and arcs identification.**
13. **Member Correspondence, to assign the calculated wind and snow loads.**
14. **New ability to check steel members at shorter times.**
15. **New command group for merging steel elements.**
16. **A new warning symbol for inefficiency of the anchoring length in the beams.**
17. **New command to merge the nodes.**
18. **New, supervisory and comprehensive print out of the results of buckling and deformation checks (total and per member) of steel structures.**
19. **Recognition of arcs from dwg 2D-3D.**
20. **SCADA connection with REVIT via ifc.**
21. **Import of beams detailing in Drawings without needing to open them in editor.**

22. Create separate files with support reactions (* .rea inside scenarios' folder) and two new files in the anal folder: combdispl.txt and combforce.txt. The first contains movements, rotations from combinations, and the second the corresponding intensive forces.

23. New type of design scenario including the EC2 provisions but not the EC8 provisions. In this scenario, the columns' stirrups do not continue in the node.

24. Jacketing for circular cross section.

25. The uniform height distribution of the earthquake in the Eurocode 8 scenario is added.

26. More distinct Attribute points of beams and posts (larger and red).

## The NEW ADD-ONS of SCADA Pro 17

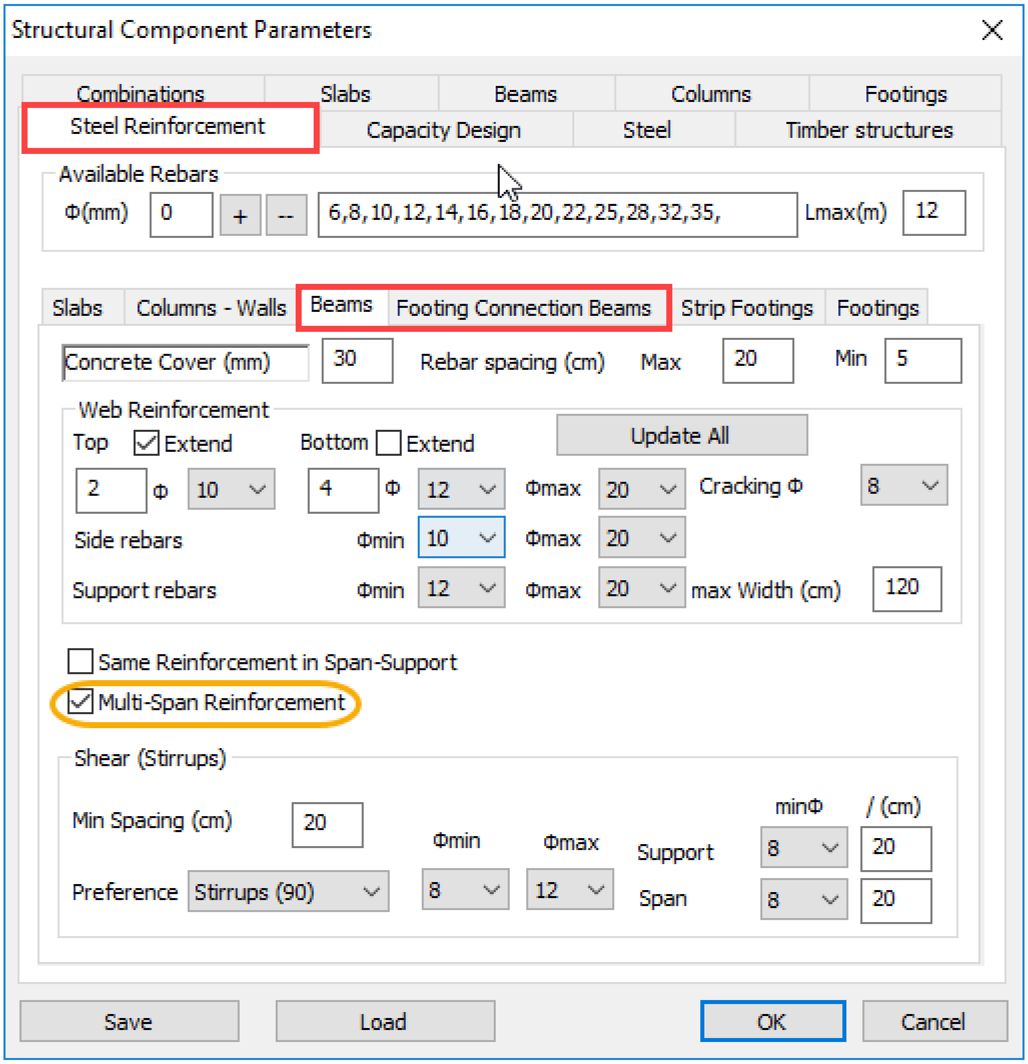
27. SCADA Pro OCP.

28. Modeling and dimensioning of flat slabs.

29. Punching shear checks.

## What's NEW in SCADA Pro 17?

### 1. New way of calculating beam reinforcement. Reinforcing with common bars of a specific length

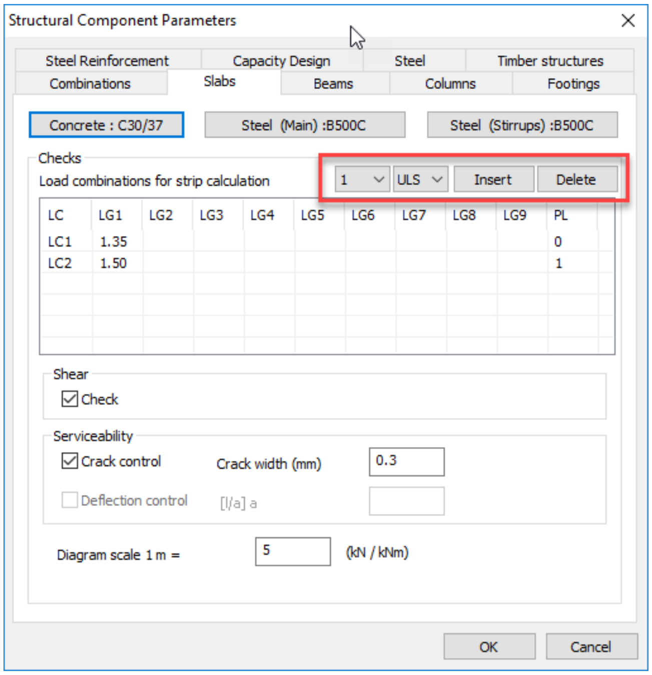


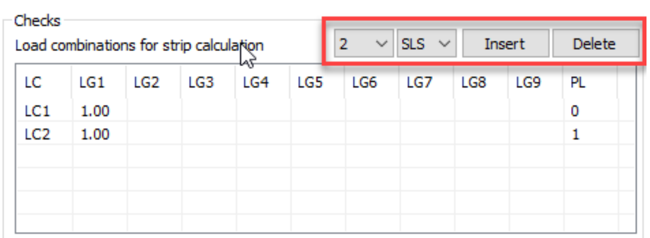

Activate the following checkbox ☑ Multi-Span Reinforcement so that the reinforcement in the openings is common to the entire beam.

The program, taking into account the reinforcement requirements across the beam (supports and openings), places a common bar aimed at the most economical solution.

## 2. More combinations for the slab's design

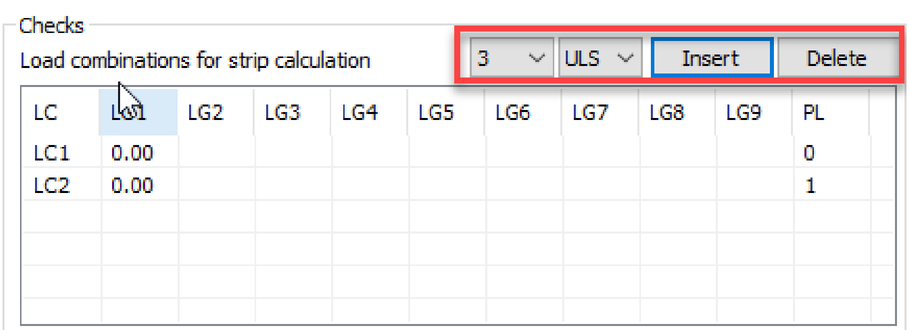
In the new version of Scada Pro there is the feature of introducing more combinations for the slab's design. In the design parameters window, inside Slabs Tab:




There are 2 default combinations, one for the Ultimate and one for the Serviceability Limit States.

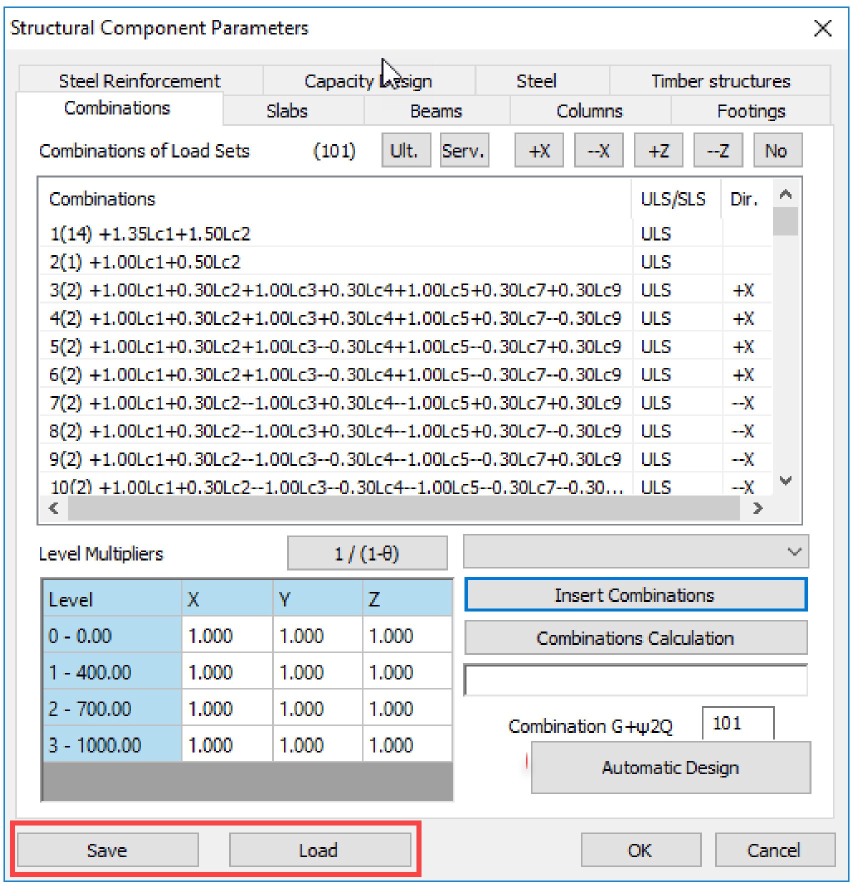
To create a new combination, press, "Insert". The new combination is the combination number 3 and the coefficients are all 0.



Change the coefficients and select the Limit State.
Same way to define as much combinations you like, or to modify the ones you already create. The program will use the combination with the worst moment regarding the USL combinations and correspondingly will make deformations checks by the functionality combinations. Using "Delete" button you can delete the created combinations. Only 1 & 2 default combinations cannot be deleted.

## 3. New storage capability of dimensioning parameters

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




Once you configure the dimensioning parameters, you can now save them to a file in order to use them in your next projects.
Press "Save" and type a name.
Use "Load" command to apply the parameters already saved.

⚠ **ATTENTION**

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

## 4. Slabs Deflection Control according EC2.

In the newest SCADA Pro version contains also the slabs Deflection Control.

Deflection control based on 7.4.2 and 7.4.3 of EC2 is presented at the end of the results of each slab.
The results of the two checks are shown separately.

DEFLECTION CONTROL (EC2 7.4.2 & 7.4.3)

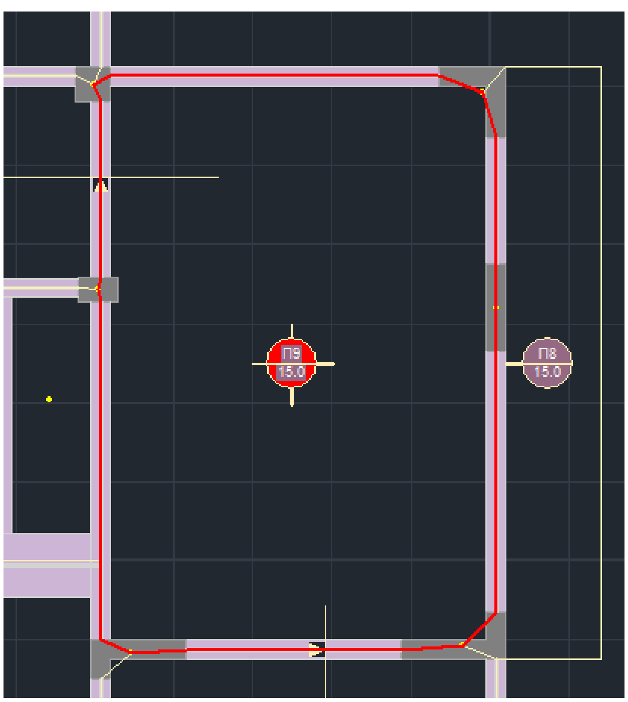
| l/d | l/d perm. | Suf. | Suggested.min thick. hs(mm) | Max. M (kNm) | dul (mm) | a | l/a (perm.) (mm) | Suf. |
|---|---|---|---|---|---|---|---|---|
| 34.59 | 80.10 | YES | 77 | -7.64 | 0.42 | 250 | 18.40 | YES |

From the first check results a minimum recommended thickness, that can not be proposed for the initial slab recognition, because its reinforcement is required to calculate it.

⚠ **Note:**

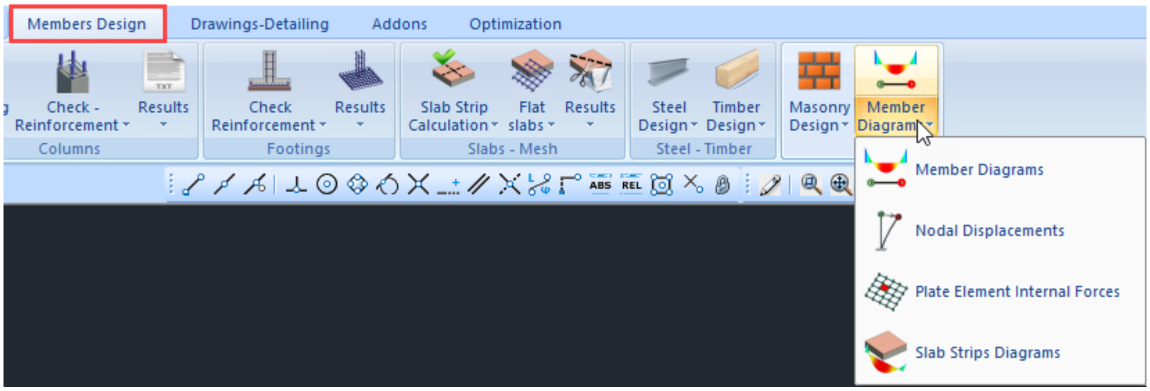
The calculation of the sizes of the first check does not involve intensive forces, while the second check considers the serviceability combination (s).

## 5. Red mark on slab symbol when thickness is less than permitted by regulation

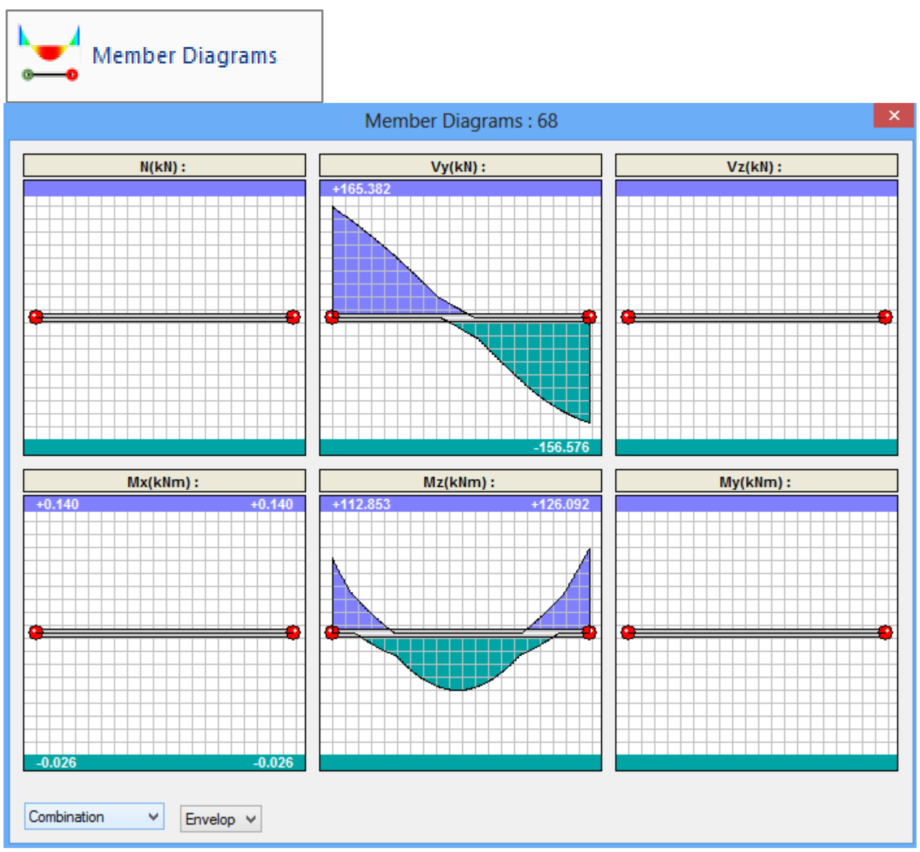


## 6. Members diagrams, nodes displacements, plate element internal forces, slab strip diagrams view

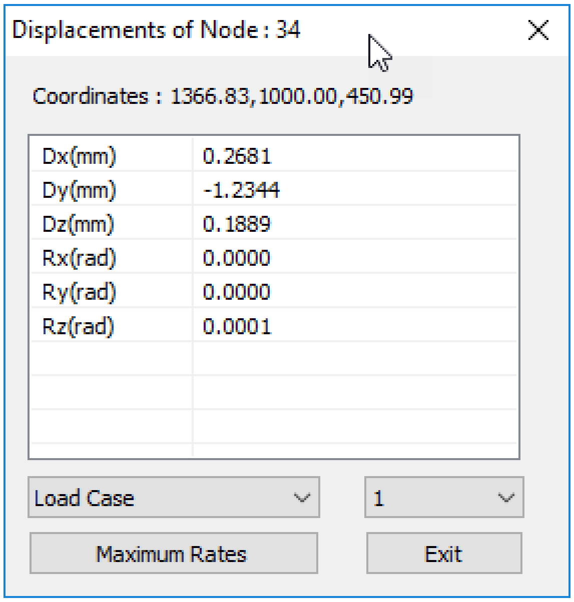
It is now possible to see not only the Members diagrams, but also nodes displacements, plate element internal forces and slab strip diagrams view.
Commands selection can be through Members Design>Members Diagrams




Or with right mouse click on a member.
Each command opens the respective dialog box:




Nodal Displacements


Displacements of Node : 34
Coordinates : 1366.83,1000.00,450.99
Dx(mm) 0.2681
Dy(mm) -1.2344
Dz(mm) 0.1889
Rx(rad) 0.0000
Ry(rad) 0.0000
Rz(rad) 0.0001
Load Case
1
Maximum Rates
Exit

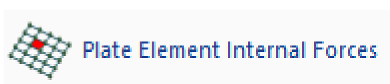


Plate Element Internal Forces

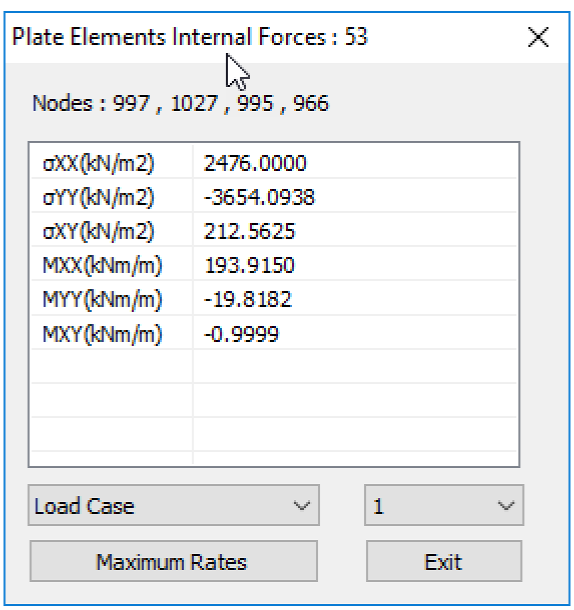
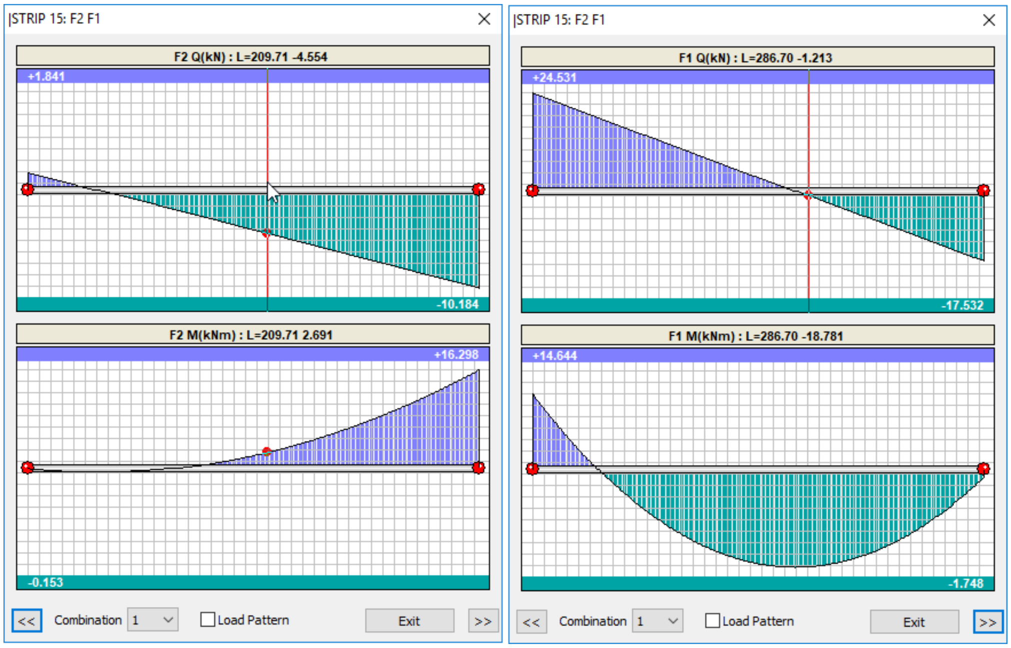
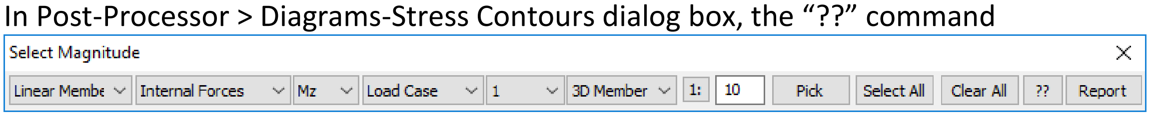


Plate Elements Internal Forces : 53
Nodes : 997 , 1027 , 995 , 966
σXX(kN/m2) 2476.0000
σYY(kN/m2) -3654.0938
σXY(kN/m2) 212.5625
MXX(kNm/m) 193.9150
MYY(kNm/m) -19.8182
MXY(kNm/m) -0.9999
Load Case
1
Maximum Rates
Exit

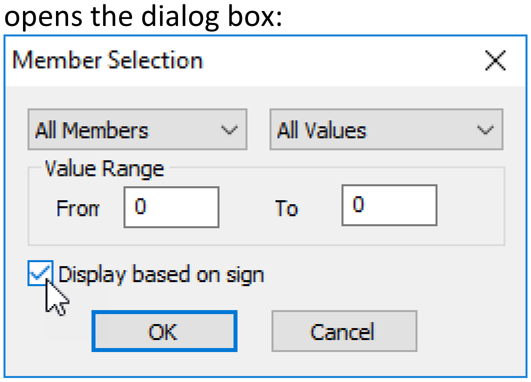




## 7. New possibility of color display of intensive sizes on linear and surface elements, based on the sign.

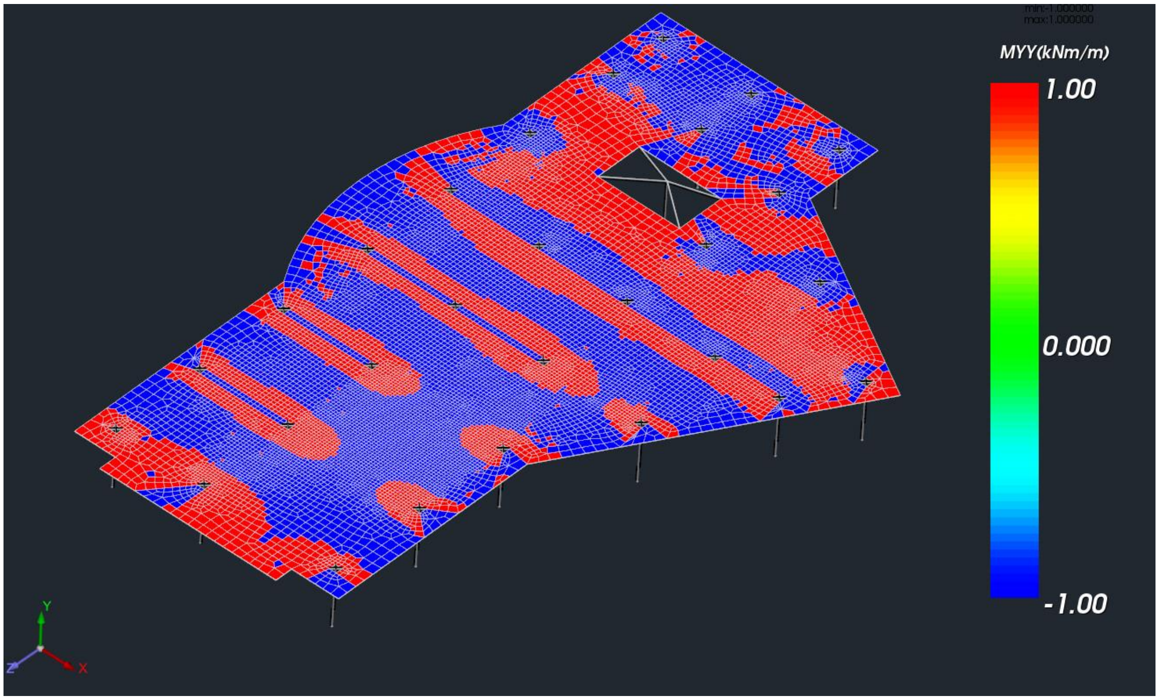
In Post-Processor > Diagrams-Stress Contours dialog box, the "??" command




opens the dialog box:

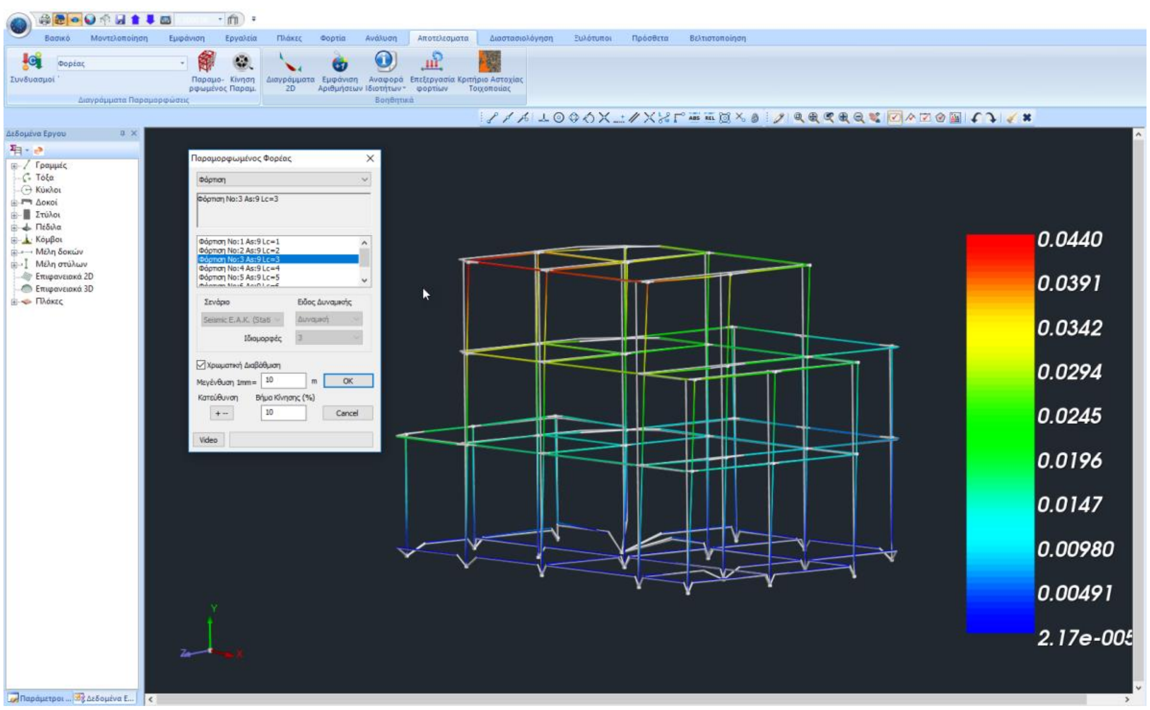



There is now the "Display based on sign" new option. Activating the checkbox, the selected size values display with two different colors, one for the positive values and one for the negative.

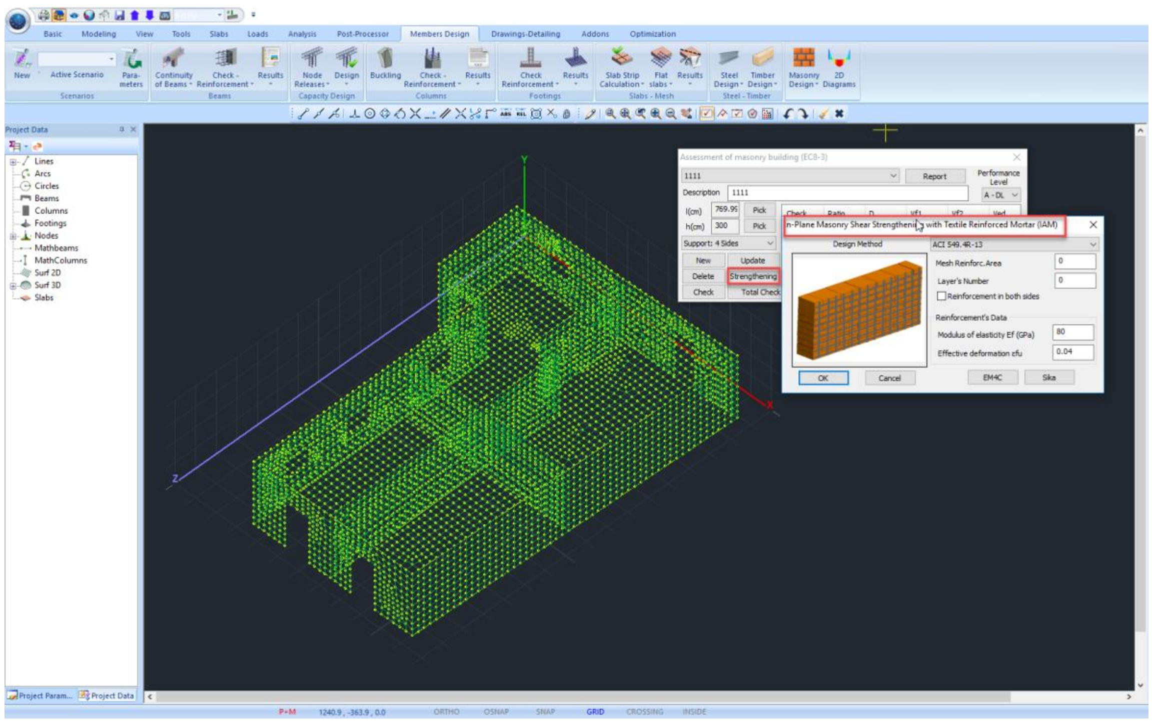



⚠ This option works for all sizes and elements (linear and surface)

## 8. New display capability of the deformation values based on the color gradation



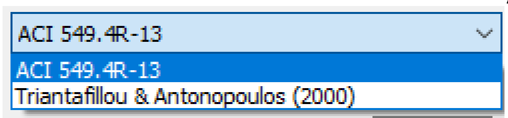
## 9. Textile Reinforced Mortar(TRM) (ACI549.4R-13 and Triantafillou & Antonopoulos methods).



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

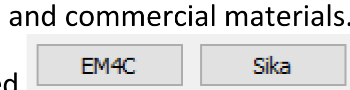
Select the "Design Method".

To SCADA Pro contains 2 methods and you can select between

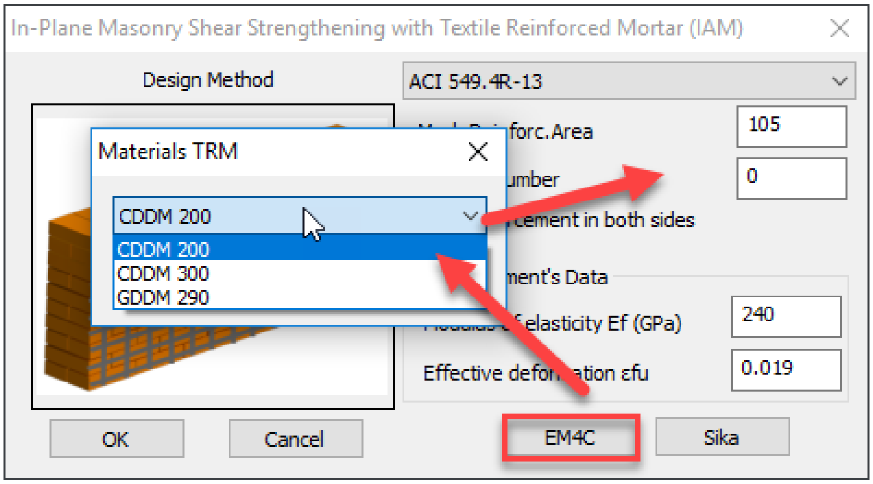


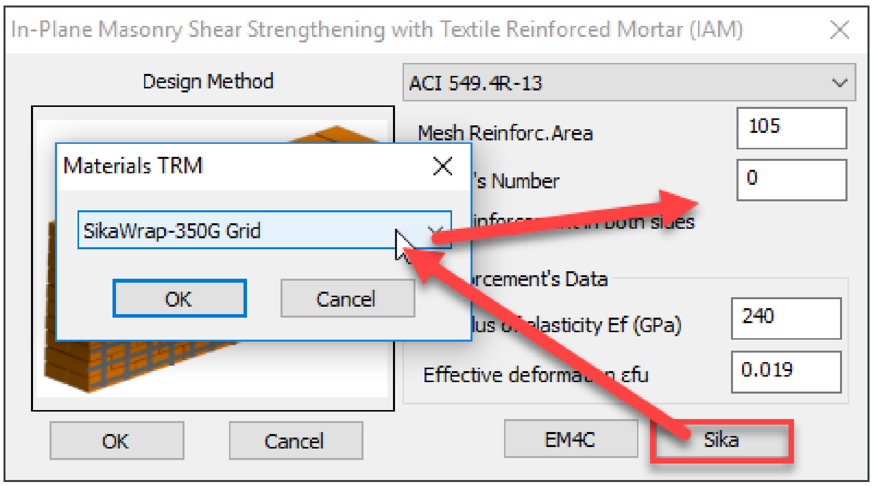

Specify the characteristics of the mesh, based on catalogs and commercial materials.

⚠ In SCADA Pro company materials have been introduced




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

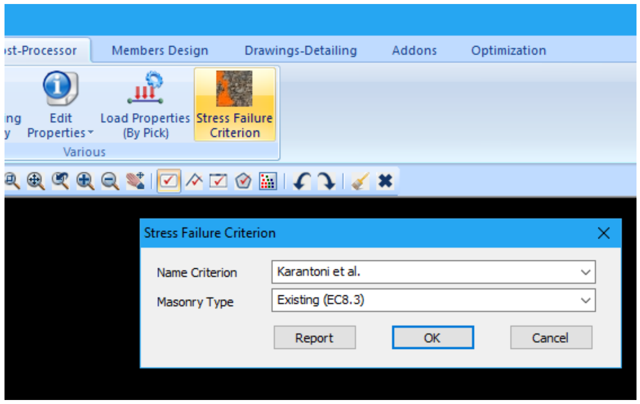


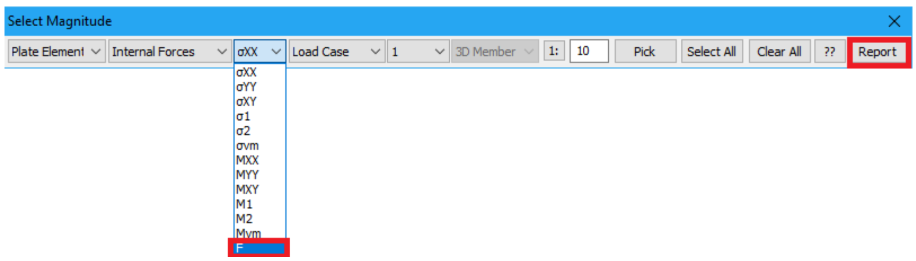

## 10. Load bearing masonry check based on Stress Failure Criterion

In the new version of SCADA Pro the check of load bearing masonry in terms of stresses based on the Karantoni et al. stress failure criterion is added.
The check in terms of stresses is performed in both curved and planar walls for existing or new masonry.
The check command is located in the **Post-Processor** tab where:

- Initially the masonry type is selected (Existing/New).

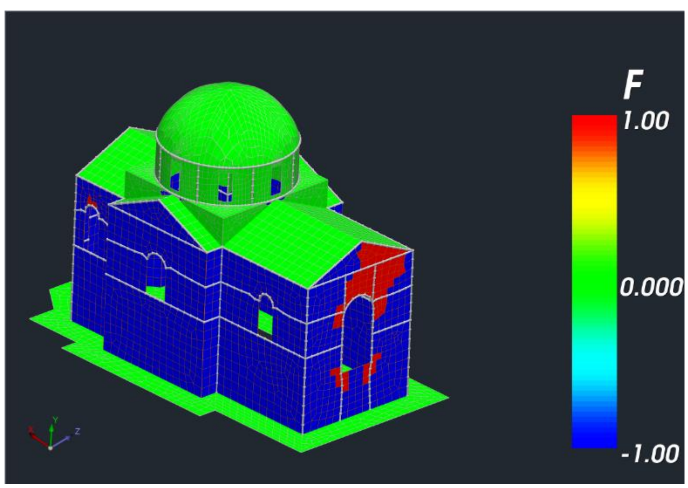



- In the following, from the surface elements stresses bar the F entity is selected.




The model is colored according to the value of the criterion:

- BLUE for sufficiency
- RED for insufficiency
- GREEN for members of different material




For better evaluation of the check results two additional options exist:

- The values of F are written in a txt file for all the surface elements through the Report command.

```
Name    Comb.        F
-----------------------

****** Plegma - S30 ******
  696       4       -0.549
  697       4       -0.573
  698       4       -0.625
  699       4       -0.731
  700       4       -0.798
  701       4       -0.807
  702       4       -0.761
  703       4       -0.748
  704       4       -0.679
  705       4       -0.333
  706       4       -0.519
  707       4       -0.338
```

- A detailed report for every mesh is created.

**Page : 1**

**Stress Failure Criterion**

**Name of Criterion** Karantoni et al.
**Masonry Type** Existing (EC8.3)
**Criterion Description** $F = \alpha J_2/f_w^2 + \lambda J_2^{(1/2)}/f_w + \beta I_1/f_w - 1$
SUFFICIENCY : Για F < 0
INSUFFICIENCY : Για F >= 0

**Mesh Check**

**Mesh Name : Plegma S21** Material : Masonry stone wall - M2 50 cm

Compressive strength $f_w$ = 0.000 (N/mm²) $\gamma_M$ = 2.20 / 1.50
Tensile strength $f_{wt}$ = 0.000 (N/mm²) CF = 1.35
Equal biaxial comp. strength $f_{wc\_b}$ = 0.000 (N/mm²)

Criterion Parameters : $\alpha$ = 0.665 b = 1.650 $c_1$ = 13.765 $\lambda_1$ = 0.581
$\beta$ = 3.835 f = 0.085 $c_2$ = 0.959 $\lambda_2$ = 0.995

| Number of elements | Total Area (m²) | Number of elements that fail | Total Failure Area (%) | Critical Combination | | | |
|---|---|---|---|---|---|---|---|
| | | | | ID. | Number of elements that fail | Total Failure Area (%) | $F_{max}$ |
| 128 | 8.64 | 0 | 0.00 | 37 | 0 | 0.00 | -0.31 |

##########################################################################################

**Mesh Name : Plegma S22** Material : Masonry stone wall - M2 50 cm

Compressive strength $f_w$ = 0.000 (N/mm²) $\gamma_M$ = 2.20 / 1.50
Tensile strength $f_{wt}$ = 0.000 (N/mm²) CF = 1.35
Equal biaxial comp. strength $f_{wc\_b}$ = 0.000 (N/mm²)

Criterion Parameters : $\alpha$ = 0.665 b = 1.650 $c_1$ = 13.765 $\lambda_1$ = 0.581
$\beta$ = 3.835 f = 0.085 $c_2$ = 0.959 $\lambda_2$ = 0.995

| Number of elements | Total Area (m²) | Number of elements that fail | Total Failure Area (%) | Critical Combination | | | |
|---|---|---|---|---|---|---|---|
| | | | | ID. | Number of elements that fail | Total Failure Area (%) | $F_{max}$ |
| 146 | 7.13 | 7 | 7.75 | 35 | 4 | 4.37 | 0.44 |

##########################################################################################

## 11. Rehabilitation methods of beams and columns are enriched, using EM4C and Sika materials

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 in 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 [EM4C] [Sika] , 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.

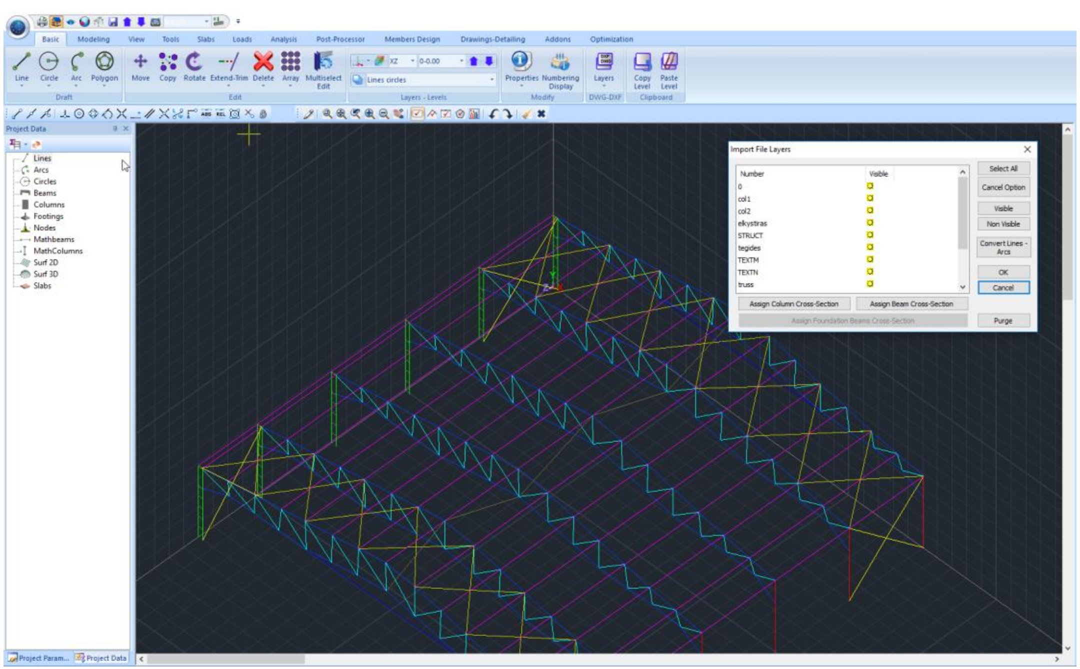
## 12. 3D DXF-DWG import. Automatic attribution of the cross sections on the drawing lines. Lines and arcs identification.

SCADA Pro17 gives you the opportunity, not only to import a 3D dwg-dxf file, but also to assign automatically the respective sections to the drawing lines.

First select the 3D display of Scada 

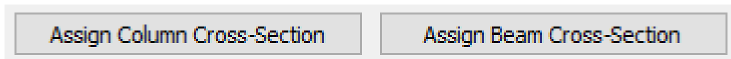
and through the command Insert, insert the 3D design. Selecting 

opens the list with all project's layers.

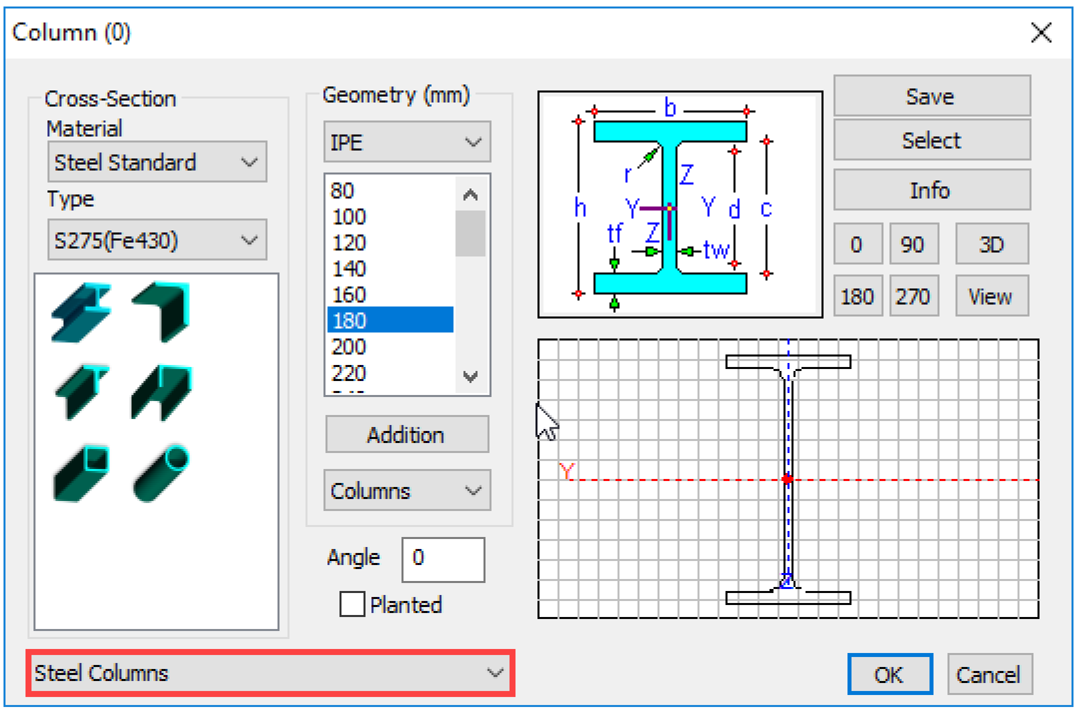


For the automatic attribution of the cross sections on the drawing lines, is necessary the correct matching of the lines in the respective layers.

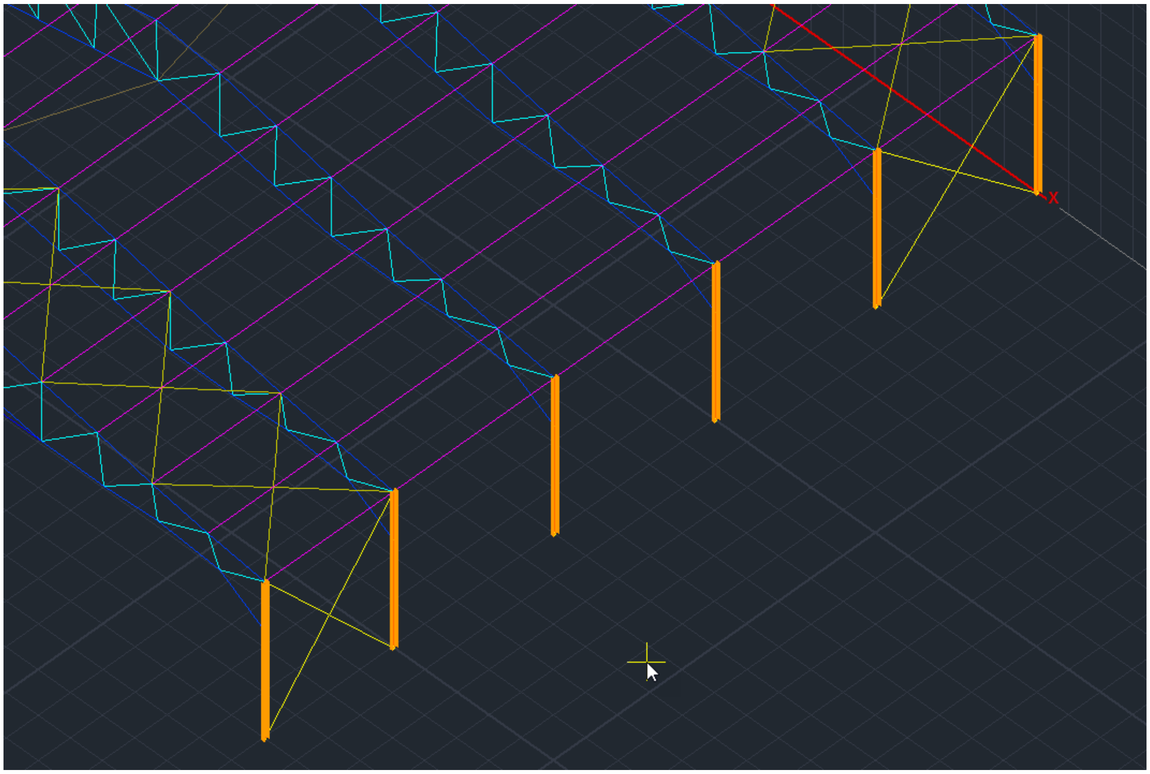
Choose a layer from the list and depending on whether includes columns or beams, select the corresponding command at the bottom of the window




Automatically opens the cross sections window, to select the one that will be assigned on all the lines of the selected layer. Also, choose the angle, the insertion point and the Scada's layer in which they will join.




Members created automatically containing physical and mathematical properties.

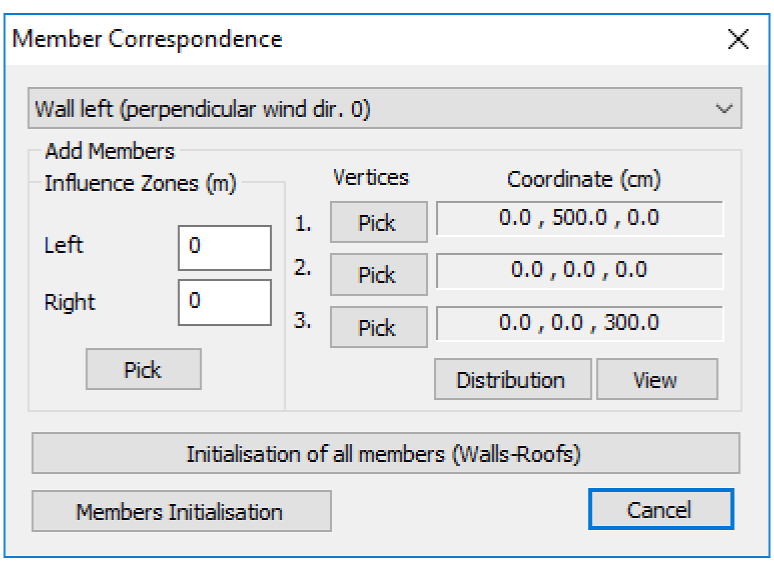


## 13. Member Correspondence, to assign the calculated wind and snow loads

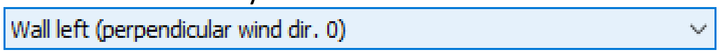
In the new version of SCADA Pro, completed and integrated the automatic calculation of influence zones for linear members in order to make the distribution of wind and snow loads.

⚠ *Remind that until now the automatic distribution was only for the structures derived from Templates. Now enable this distribution on any surface.*

By selecting the command now opens the following dialog box



The part on the old definition of the influence zones did not change but added to the right a new part to define the area with three points.
The definition always concerns the active area




*Better to start the procedure by pressing the "Members Initialization" button.*

Indicate the point graphically with the following particularity:

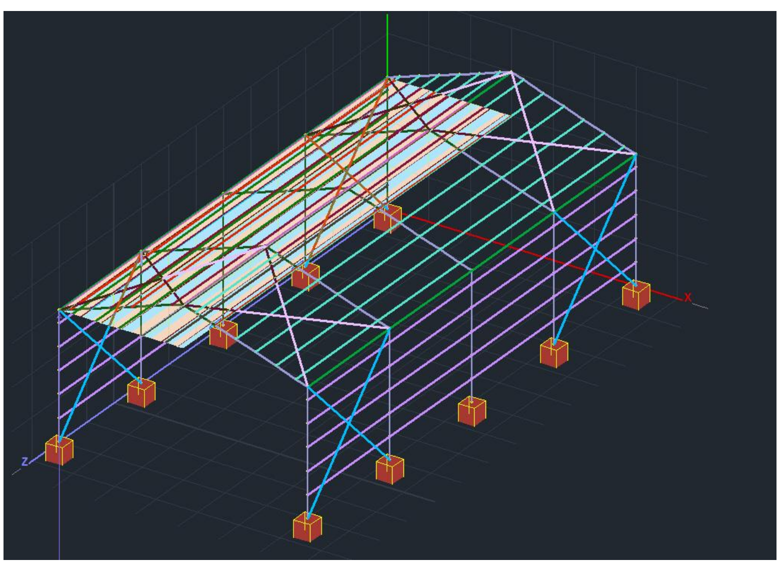
- The first two points define the direction by which the automatic calculation of influence surfaces made for items which are parallel to this direction.
  Note also that the distribution will be for all linear members belonging to this level and are parallel to the first direction.
- After you define the three points, press the "Distribution" button and the program automatically makes the distribution and displays it.

Similarly for the other walls.

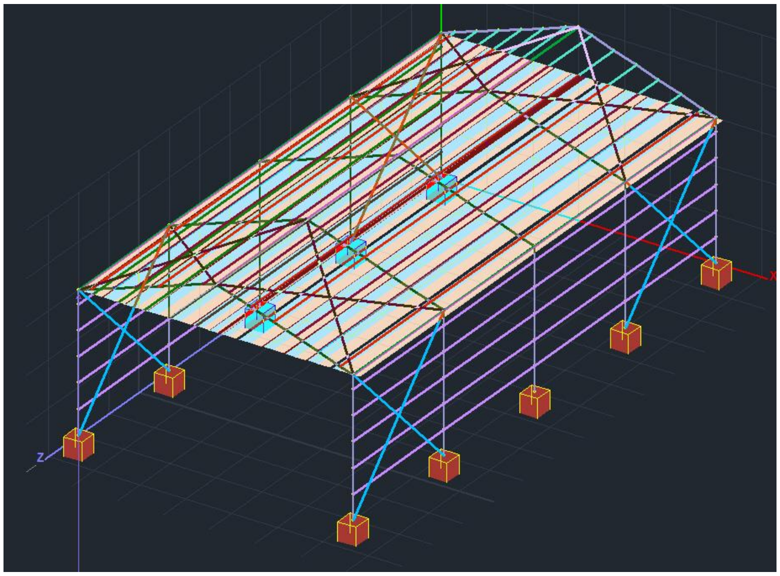
Concerning the roofs, the definition can be sequentially.
First select the roof


, then you must define the individual areas.

First define the left slope indicating graphically the 3 points

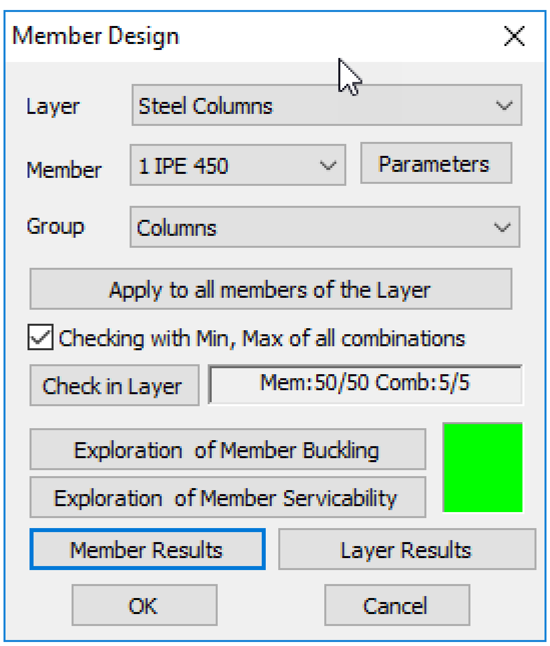


and then the right. The overall result is the following



- ⚠ Finally it is worth noting that if the walls are properly defined there is NO need of more definition. Just select each wall and press «Distribution». The distribution becomes and simultaneously displays on the linear members belonging to this wall.
- ⚠ Same for the flat roofs only.

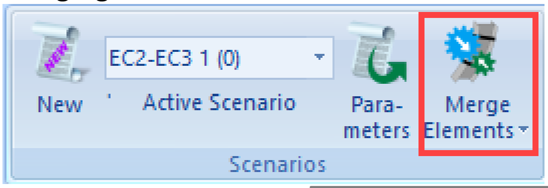
### 14. New ability to check steel members at shorter times.




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

## 15. New command group for merging steel elements.

In “Members Design” added a new command group that provides steel elements merging.





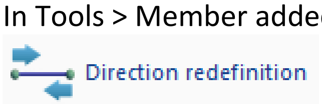
Merge elements means that, either automatically or manually, the individual parts of a single element, merge in each buckling direction. Meaning that, the buckling length is considered computationally, not the actual length of the element, but the unified from the beginning to the end of the column or beam, respectively.

In addition, in the presentation of the results, for these merged elements, the worst results display only once and not for each individual part, as it was so far.

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

## 16. New command for member’s direction redefinition.

In Tools > Member added a new command named “Direction Redefinition”.




This command should be used when one or both of the following messages appear in the Model Checks Reports:

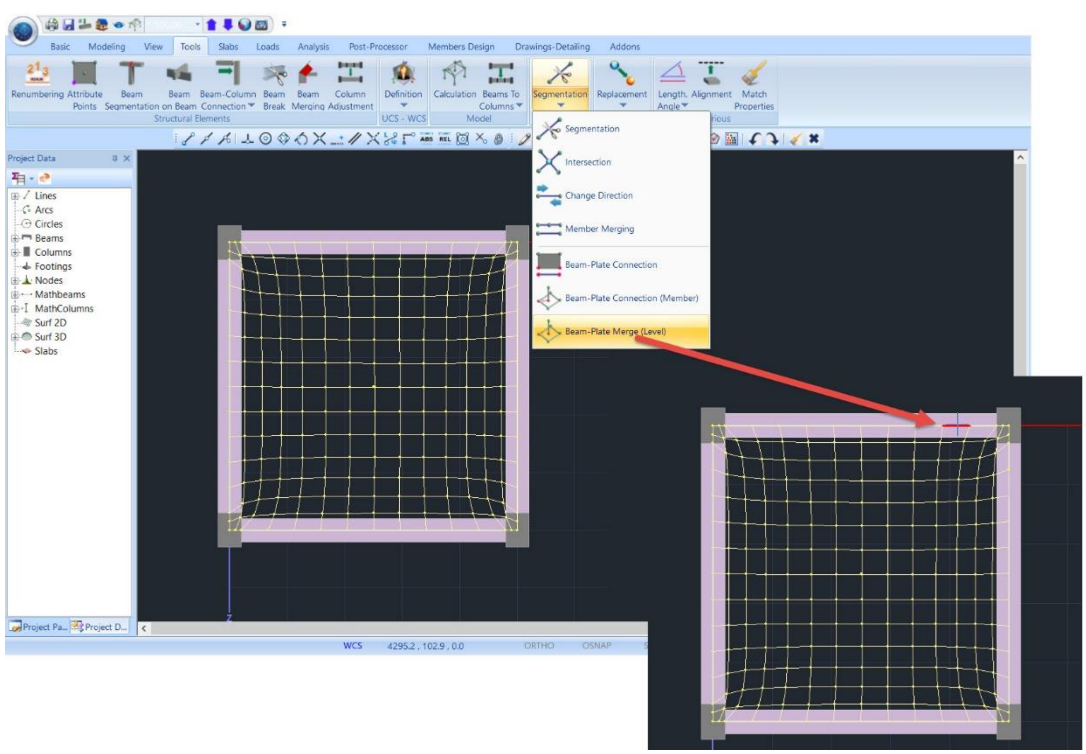
Error1678: The column 21 has been assigned with wrong orientation
There are members with wrong local axis

The first one, which concerns only the columns, has to do with the direction of their placement (the correct direction is from the bottom to the top),
while the second is a general message concerning beams and columns, and especially for the beams, appears when they are not placed with the program conversion, from left to right and from top to bottom.

So when the above messages appear, using the "Direction Redefinition" command the program corrects automatically the orientation for the entire model.

## 17. New command that Merge Beams


Where the beams include surface elements is a need to break the members of the beams in order to ensure the necessary connections between the linear and surface elements.



Consequently, breaking the beam in small portions creates the need for unification in order to be able to dimensioned as a single member. This is accomplished by using the command Merge Beams. 


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

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

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

## 18. A new warning symbol for inefficiency of the anchoring length in the beams.

The way of calculating the lbd anchorage lengths for different regulations is summarized below:

The total lbd is calculated and divided into l1 and l2. L1 is the linear anchorage length, and l2 is the one that rotates in the node.

**NOTES:**

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

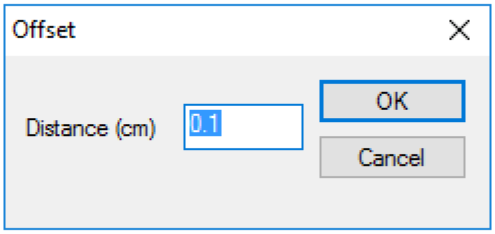
Based on the above, resulting:

1. For EC2 w/o EC8 scenario as well as for all EC with DCL and DCM ductility classes, it does not obey a minimum linear anchorage length lb,min, but checks the total length lbd with lb,min according to 8.4.4 Of the EC2.
   So, there will never be an error message because if the anchorage length is bigger than the width of the support, the bar will reach the face and then turn to the node.
2. For EC with DCH high ductility class, it obeys the minimum linear anchorage length according to 5.6.2 of EC8 . The error message will be displayed.

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

## 19. New command to merge the nodes

### Merge




Command to merge the nodes in small distances between them.
Select the command and specify a distance value. Nodes at a distance less or equal to this will be merged, resulting in a single node.

## 20. New, supervisory and comprehensive print out of the results of buckling and deformation checks (total and per member) of steel structures

## 21. Recognition of arcs from dwg 2D-3D

## 22. SCADA connection with REVIT via ifc

## 23. Import of beams detailing in Drawings without needing to open them in editor

## 24. Create separate files with support reactions (* .rea inside scenarios’ folder) and two new files in the anal folder: combdispl.txt and combforce.txt. The first contains movements, rotations from combinations, and the second the corresponding intensive forces.

## 25. New type of design scenario including the EC2 provisions but not the EC8 provisions. In this scenario, the columns’ stirrups do not continue in the node.

## 26. Jacketing for circular cross section

## 27. The uniform height distribution of the earthquake in the Eurocode 8 scenario is added

## 28. More distinct Attribute points of beams and posts (larger and red)

## The NEW ADD-ONS of SCADA Pro 17

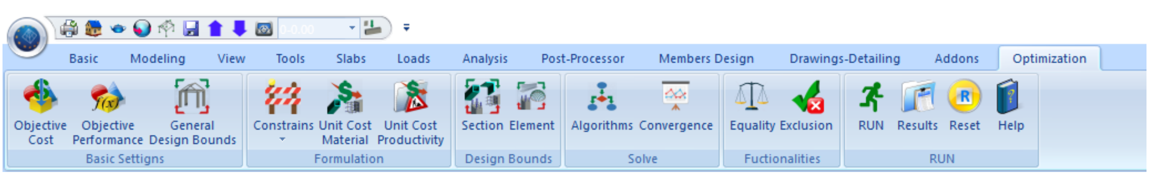
### 1. SCADA Pro OCP

SCADA Pro OCP is a new module of SCADA Pro that represents an advanced, real-world, optimum design computing platform for real-world civil engineering structures. The main goal for the implementation of SCADA Pro OCP module is to **reduce construction and material cost** consistent with the required performance, reliability, quality and safety within an innovative technological framework.

SCADA Pro OCP module provide:

- **Advanced** and **easy** to use ribbon-based interface makes structural optimization a **single-click** process allowing numerous customizable options.
- The option of selecting **multiple criteria** related to the cost of the structure such as construction cost, construction of materials, and environmental cost during the life span of the structure. These criteria used either as "Objectives" or as "Constraints', which are taken into account in addition to those imposed by the design code regulations.
- Solve optimization problems by selecting the solution algorithm from a list of **deterministic** and **probabilistic search numerical algorithms** that are at the cutting edge of technology, which can replace the "trial and error" traditional process of structural design and with an optimized solution quickly obtained.
- Πολλαπλές επιλογές για τον καθορισμό του εύρους των **μεταβλητών σχεδιασμού** και την ομαδοποίησή τους είτε σε **επίπεδο διατομής ή σε επίπεδο δομικού στοιχείου.**
- Multiple options for determining the value range of the **design variables** and grouping them either **by type of cross-section or by type of structural element.**
- **Comparison of the initial design with the optimum one.**

Commands are grouped into sections according to their type of use.

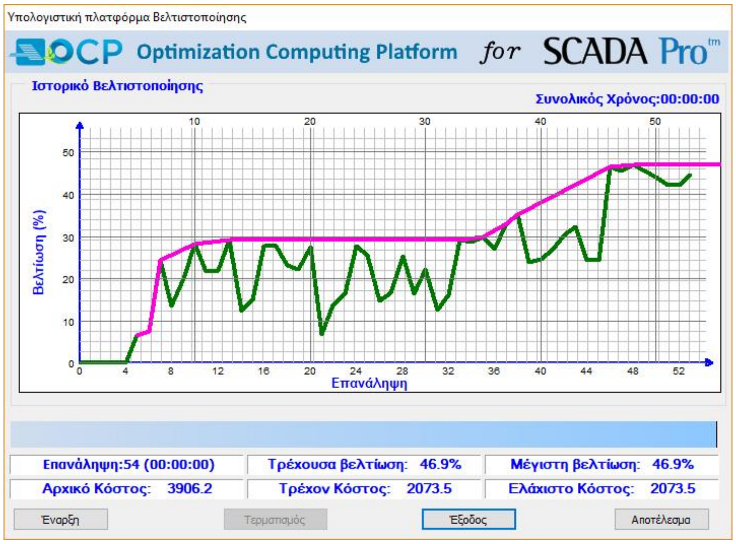



The optimization process is performed in two stages:

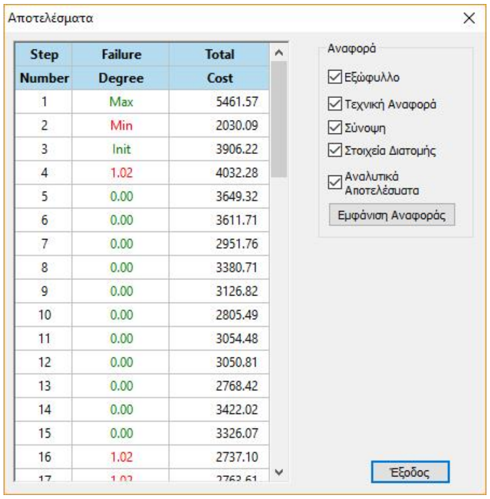
• The first stage is the definition of the basic parameters of the project:
(a) the optimization criterion (objective function)

(b) additional constraints in terms of cost or performance apart from the design code checks
(c) the limits of variation of the cross-sectional dimensions (design bounds) of the structural elements (design variables).

• In the second phase is the definition of the optimization algorithm, the convergence criterion of the iterative solution process and its immediate execution.

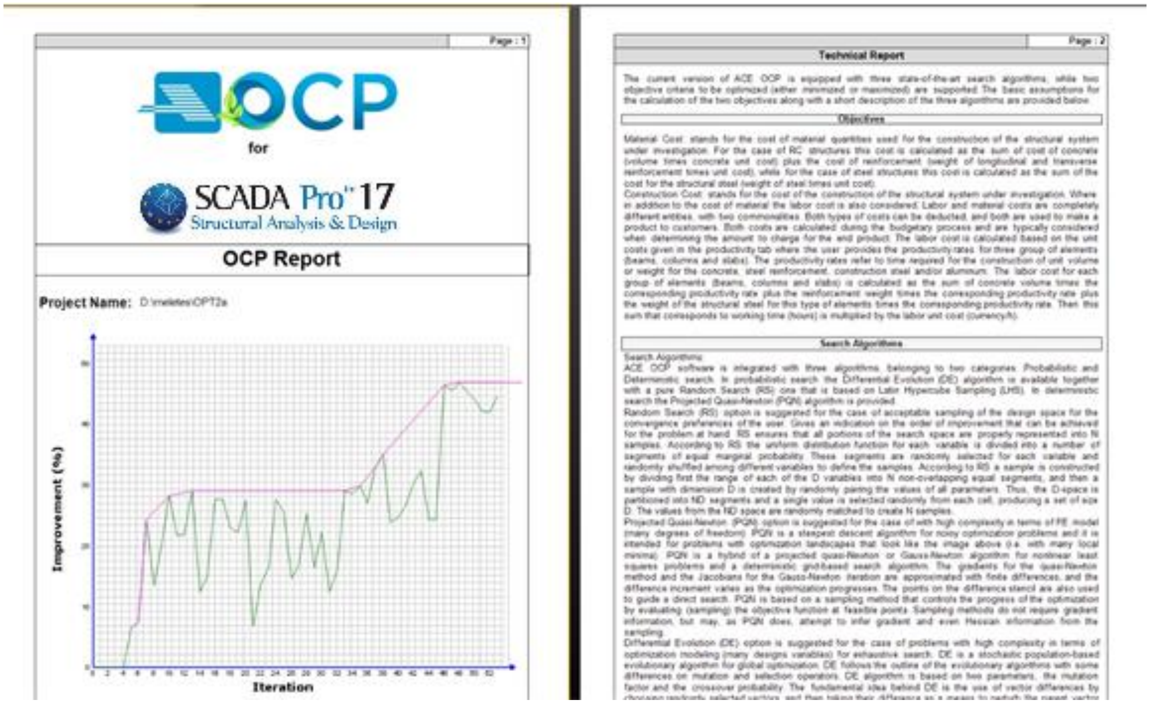



After completing the iterative optimization process, it is possible to analyze the results obtained with parallel comparison of the initial parameter values with the optimum values.

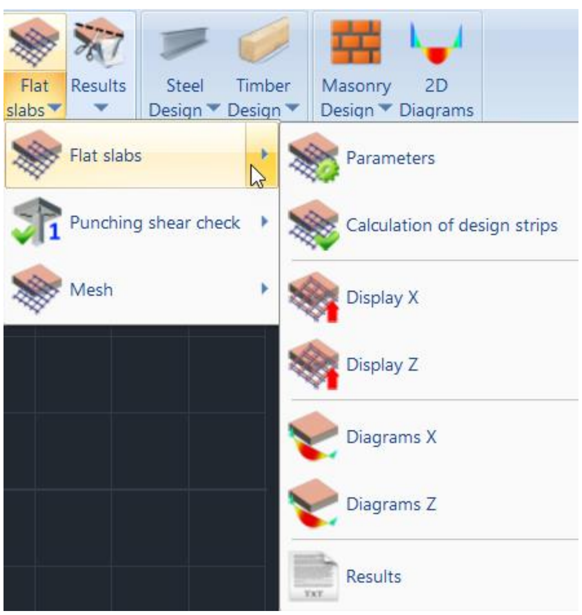



| Step Number | Failure Degree | Total Cost |
|---|---|---|
| 1 | Max | 5461.57 |
| 2 | Min | 2030.09 |
| 3 | Init | 3906.22 |
| 4 | 1.02 | 4032.28 |
| 5 | 0.00 | 3649.32 |
| 6 | 0.00 | 3611.71 |
| 7 | 0.00 | 2951.76 |
| 8 | 0.00 | 3380.71 |
| 9 | 0.00 | 3126.82 |
| 10 | 0.00 | 2805.49 |
| 11 | 0.00 | 3054.48 |
| 12 | 0.00 | 3050.81 |
| 13 | 0.00 | 2768.42 |
| 14 | 0.00 | 3422.02 |
| 15 | 0.00 | 3326.07 |
| 16 | 1.02 | 2737.10 |

There is also the option of exporting results, analyzing all the parameters of the optimization problem and the optimization history diagram; the value variation diagram of the optimization criterion (objective function) in each step of the iterative process.




## 2. Modeling and dimensioning of flat slabs

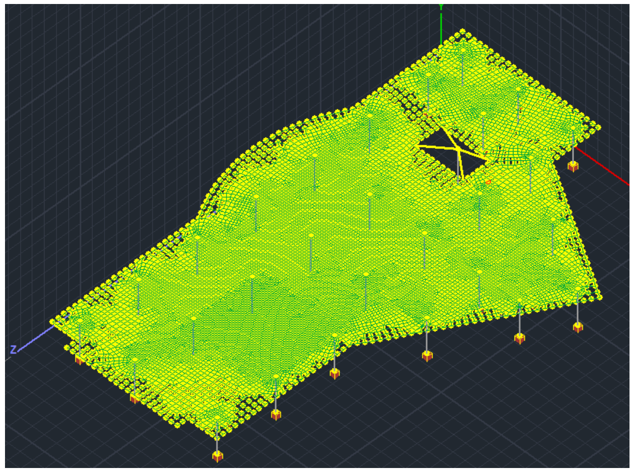



The new version of SCADA Pro offers the possibility of creating flat slabs (slabs without the presence of beams) with the finite element method.

The procedure for the modeling of flat slabs requires:

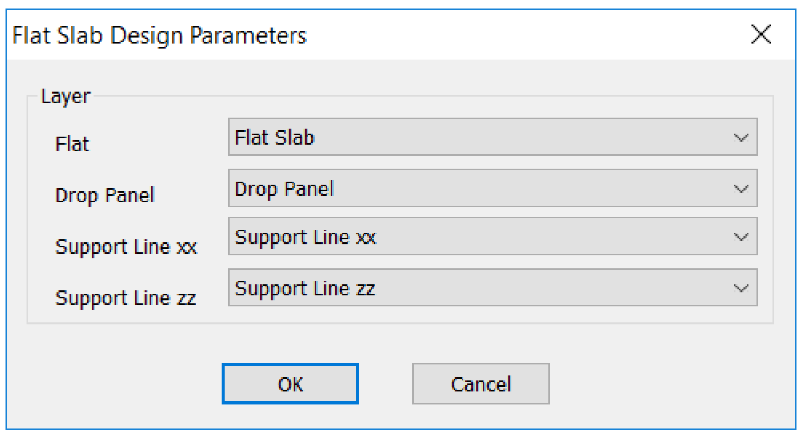
- the 3D Mesh definition,
- the External Boundary creation ,

- the Holes automatically creation in place of the Columns,
- the Mesh calculation and the mathematical model calculation.



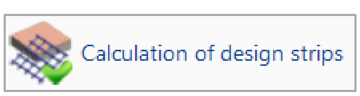
The command "Flat slabs" includes the commands:

## ❖ Parameters


In the dialog box you define the correlation between Layers.

## ❖ Calculation of design strips

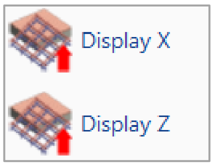



According to Annex I of EC2 flat slab is divided into design strips. These are the areas that are automatically created by the program on both sides of the Support Line, according to Figure I.1 of EC2.

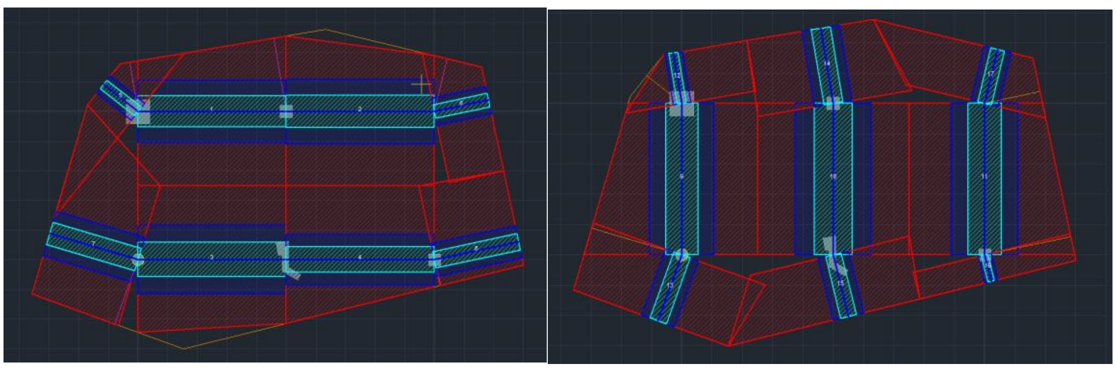
Select the command Calculation of design strips and the program automatically creates them.

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

### ❖ Display X, Z



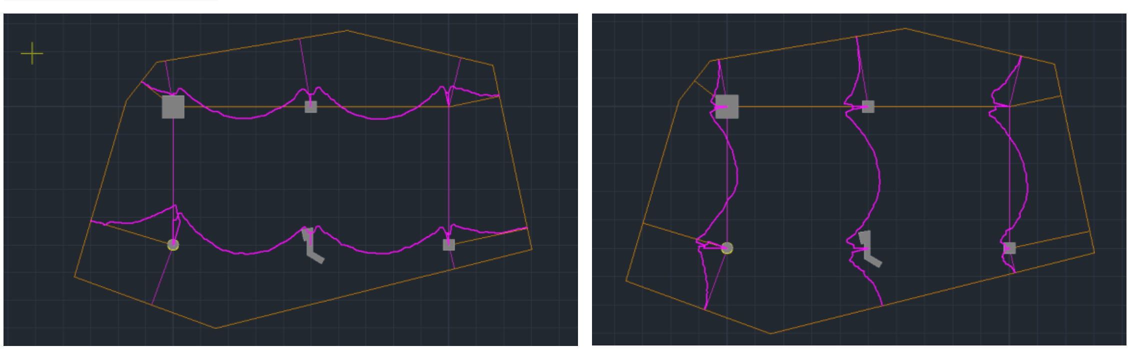

Select to display the Design Strips in both directions.



Design Strips along the X and Z axes

### ❖ Diagrams X, Z


Select Diagrams in both directions to see the corresponding diagrams.



## ❖ Results


This command opens the Results file through the Report.

Each page concerns a Strip Line.
Initially described the characteristics of the Strip.

**Page : 1**

**Strip Calculations**

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

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

| Analysis Results and Reinforcement | | | | | | | | | | Top |
|---|---|---|---|---|---|---|---|---|---|---|
| | 125.00 cm ($L_{start}$) | | | | | 250.00 cm ($L_{centre}$) | | | | |
| Zone | M (kNm) | Width (cm) | $A_{s,req}$ (cm²/m) | $A_{s,prvd}$ (cm²/m) | Φ/s | M (kNm) | Width (cm) | $A_{s,req}$ (cm²/m) | $A_{s,prvd}$ (cm²/m) | Φ/s |
| Left | -33.909 | 86.5 | 5.660 | 5.660 | 8/8 | | 127.5 | | 1.415 | 8/20 |
| L-C | -45.708 | 52.7 | 9.744 | 9.744 | 8/5 | | 52.7 | | 2.606 | 8/19 |
| Center | -170.151 | 105.3 | 19.129 | 19.129 | 12/5 | | 105.3 | | 4.782 | 8/10 |
| R-C | -44.750 | 52.7 | 9.530 | 9.530 | 8/5 | | 52.7 | | 2.382 | 8/20 |
| Right | -72.082 | 144.7 | 5.478 | 5.478 | 8/9 | | 144.7 | | 1.369 | 8/20 |

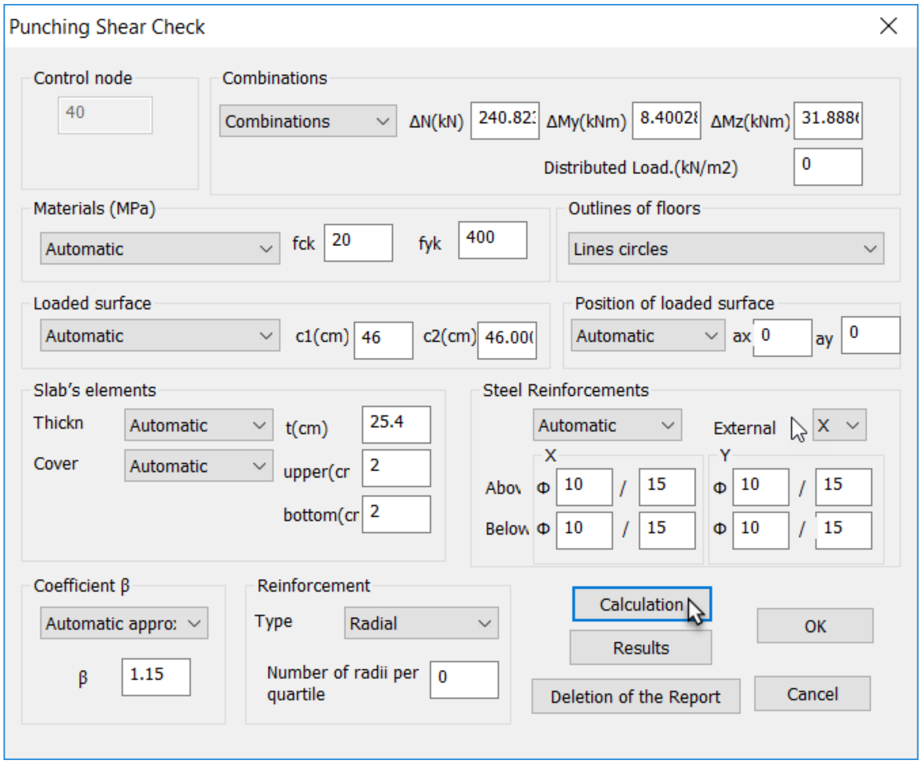
| | 125.00 cm ($L_{end}$) | | | | |
|---|---|---|---|---|---|
| Zone | M (kNm) | Width (cm) | $A_{s,req}$ (cm²/m) | $A_{s,prvd}$ (cm²/m) | Φ/s |
| Left | -44.707 | 149.7 | 3.284 | 3.904 | 8/12 |
| L-C | -48.725 | 52.7 | 10.423 | 10.423 | 10/7 |
| Center | -162.632 | 105.3 | 18.175 | 18.175 | 12/6 |
| R-C | -42.495 | 52.7 | 9.027 | 9.027 | 8/5 |
| Right | -43.124 | 144.7 | 3.240 | 3.904 | 8/12 |

| Analysis Results and Reinforcement | | | | | | | | | | Bottom |
|---|---|---|---|---|---|---|---|---|---|---|
| | 125.00 cm ($L_{start}$) | | | | | 250.00 cm ($L_{centre}$) | | | | |
| Zone | M (kNm) | Width (cm) | $A_{s,req}$ (cm²/m) | $A_{s,prvd}$ (cm²/m) | Φ/s | M (kNm) | Width (cm) | $A_{s,req}$ (cm²/m) | $A_{s,prvd}$ (cm²/m) | Φ/s |
| Left | 11.110 | 86.5 | 1.380 | 1.380 | 8/20 | 52.042 | 127.5 | 5.346 | 5.346 | 8/9 |
| L-C | 2.346 | 52.7 | 0.475 | 1.438 | 8/20 | 27.508 | 52.7 | 5.752 | 5.752 | 8/8 |
| Center | 9.371 | 105.3 | 0.952 | 1.438 | 8/20 | 55.015 | 105.3 | 5.752 | 5.752 | 8/8 |
| R-C | 2.362 | 52.7 | 0.478 | 1.438 | 8/20 | 27.508 | 52.7 | 5.752 | 5.752 | 8/8 |
| Right | | 144.7 | | 0.999 | 8/20 | 52.965 | 144.7 | 3.995 | 3.995 | 8/12 |

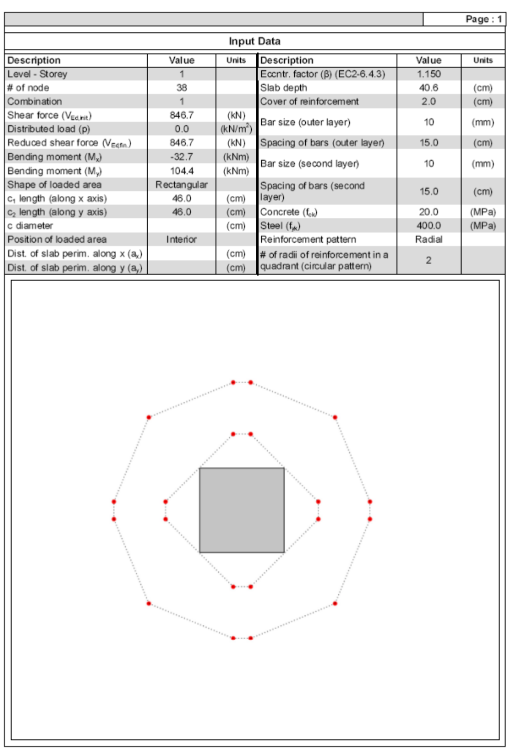
| | 125.00 cm ($L_{end}$) | | | | |
|---|---|---|---|---|---|
| Zone | M (kNm) | Width (cm) | $A_{s,req}$ (cm²/m) | $A_{s,prvd}$ (cm²/m) | Φ/s |
| Left | | 149.7 | | 1.337 | 8/20 |
| L-C | | 52.7 | | 1.438 | 8/20 |
| Center | 7.314 | 105.3 | 0.742 | 1.438 | 8/20 |
| R-C | 8.729 | 52.7 | 1.786 | 1.786 | 8/20 |
| Right | 27.511 | 144.7 | 2.052 | 2.052 | 8/20 |

### 3. Punching shear checks

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



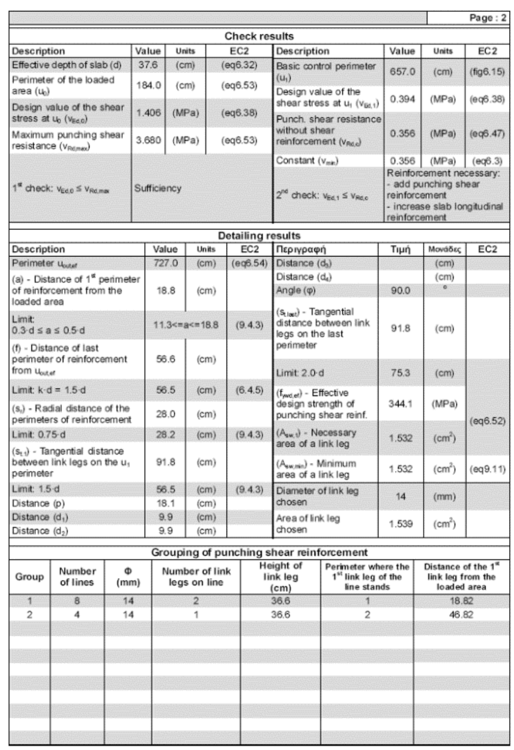

- Production of complete results print out, with detailed reinforcement layout plan (radial or cross), whenever required.
- Details designing.



Page : 1

**Input Data**

| Description | Value | Units | Description | Value | Units |
|---|---|---|---|---|---|
| Level - Storey | 1 | | Eccntr. factor (β) (EC2-6.4.3) | 1.150 | |
| # of node | 38 | | Slab depth | 40.6 | (cm) |
| Combination | 1 | | Cover of reinforcement | 2.0 | (cm) |
| Shear force ($V_{Ed,init}$) | 846.7 | (kN) | Bar size (outer layer) | 10 | (mm) |
| Distributed load (p) | 0.0 | (kN/m²) | | | |
| Reduced shear force ($V_{Ed,fin.}$) | 846.7 | (kN) | Spacing of bars (outer layer) | 15.0 | (cm) |
| Bending moment ($M_x$) | -32.7 | (kNm) | Bar size (second layer) | 10 | (mm) |
| Bending moment ($M_y$) | 104.4 | (kNm) | | | |
| Shape of loaded area | Rectangular | | Spacing of bars (second layer) | 15.0 | (cm) |
| $c_1$ length (along x axis) | 46.0 | (cm) | | | |
| $c_2$ length (along y axis) | 46.0 | (cm) | Concrete ($f_{ck}$) | 20.0 | (MPa) |
| c diameter | | (cm) | Steel ($f_{yk}$) | 400.0 | (MPa) |
| Position of loaded area | Interior | | Reinforcement pattern | Radial | |
| Dist. of slab perim. along x ($a_x$) | | (cm) | # of radii of reinforcement in a quadrant (circular pattern) | 2 | |
| Dist. of slab perim. along y ($a_y$) | | (cm) | | | |



Page : 2

**Check results**

| Description | Value | Units | EC2 | Description | Value | Units | EC2 |
|---|---|---|---|---|---|---|---|
| Effective depth of slab (d) | 37.6 | (cm) | (eq6.32) | Basic control perimeter ($u_1$) | 657.0 | (cm) | (fig6.15) |
| Perimeter of the loaded area ($u_0$) | 184.0 | (cm) | (eq6.53) | Design value of the shear stress at $u_1$ ($v_{Ed,1}$) | 0.394 | (MPa) | (eq6.38) |
| Design value of the shear stress at $u_0$ ($v_{Ed,0}$) | 1.406 | (MPa) | (eq6.38) | Punch. shear resistance without shear reinforcement ($v_{Rd,c}$) | 0.356 | (MPa) | (eq6.47) |
| Maximum punching shear resistance ($v_{Rd,max}$) | 3.680 | (MPa) | (eq6.53) | Constant ($v_{min}$) | 0.356 | (MPa) | (eq6.3) |
| 1st check: $v_{Ed,0} \le v_{Rd,max}$ | Sufficiency | | | 2nd check: $v_{Ed,1} \le v_{Rd,c}$ | Reinforcement necessary: - add punching shear reinforcement - increase slab longitudinal reinforcement | | |

**Detailing results**

| Description | Value | Units | EC2 |
|---|---|---|---|
| Perimeter $u_{out,ef}$ | 727.0 | (cm) | (eq6.54) |
| (a) - Distance of 1st perimeter of reinforcement from the loaded area | 18.8 | (cm) | |
| Limit: 0.3·d ≤ a ≤ 0.5·d | 11.3<=a<=18.8 | | (9.4.3) |
| (f) - Distance of last perimeter of reinforcement from $u_{out,ef}$ | 56.6 | (cm) | |
| Limit: k·d = 1.5·d | 56.5 | (cm) | (6.4.5) |
| ($s_r$) - Radial distance of the perimeters of reinforcement | 28.0 | (cm) | |
| Limit: 0.75·d | 28.2 | (cm) | (9.4.3) |
| ($s_{t,1}$) - Tangential distance between link legs on the $u_1$ perimeter | 91.8 | (cm) | |
| Limit: 1.5·d | 56.5 | (cm) | (9.4.3) |
| Distance (p) | 18.1 | (cm) | |
| Distance ($d_1$) | 9.9 | (cm) | |
| Distance ($d_2$) | 9.9 | (cm) | |

| Περιγραφή | Τιμή | Μονάδες | EC2 |
|---|---|---|---|
| Distance ($d_3$) | | (cm) | |
| Distance ($d_4$) | | (cm) | |
| Angle (φ) | 90.0 | ° | |
| ($s_{t,last}$) - Tangential distance between link legs on the last perimeter | 91.8 | (cm) | |
| Limit: 2.0·d | 75.3 | (cm) | |
| ($f_{ywd,ef}$) - Effective design strength of punching shear reinf. | 344.1 | (MPa) | (eq6.52) |
| ($A_{sw,1}$) - Necessary area of a link leg | 1.532 | (cm²) | |
| ($A_{sw,min}$) - Minimum area of a link leg | 1.532 | (cm²) | (eq9.11) |
| Diameter of link leg chosen | 14 | (mm) | |
| Area of link leg chosen | 1.539 | (cm²) | |

**Grouping of punching shear reinforcement**

| Group | Number of lines | Φ (mm) | Number of link legs on line | Height of link leg (cm) | Perimeter where the 1st link leg of the line stands | Distance of the 1st link leg from the loaded area |
|---|---|---|---|---|---|---|
| 1 | 8 | 14 | 2 | 36.6 | 1 | 18.82 |
| 2 | 4 | 14 | 1 | 36.6 | 2 | 46.82 |