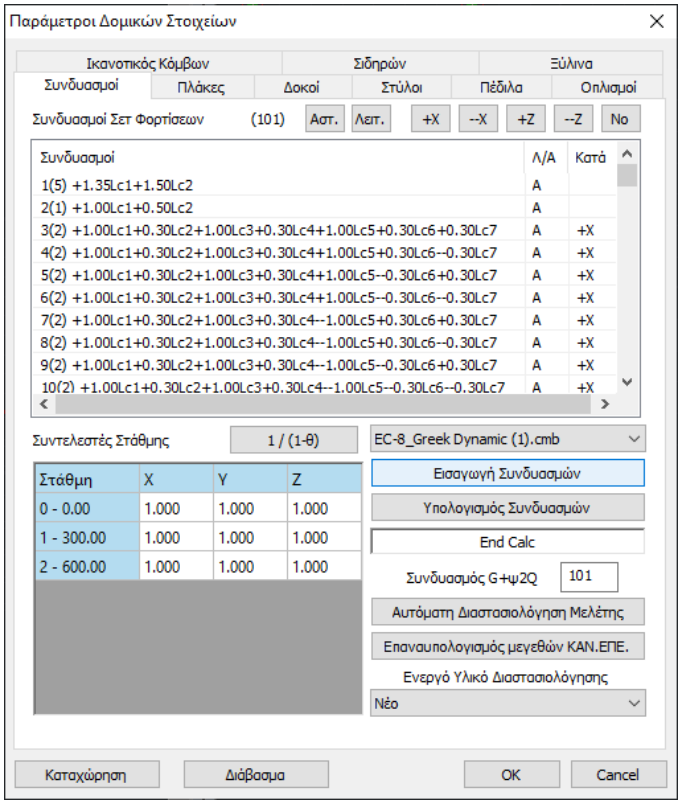
SCADA Pro 25tm
Structural Analysis & Design

User Manual

10A. DIMENSIONING

Part 1/4: Scenarios - Diagrams



ACE HELLAS
INTEGRATED SOLUTIONS

1. Table of contents

Chapter 10: Sizing - Scenarios - Diagrams (part 1/4)

The 10th Module is called "DISCUSSION" and includes the following groups of commands:

- √ Scenarios - GENERAL
- √ Beams
- √ Equitable Control
- √ Pillars
- √ Sandals
- √ Slabs-Mesh

  BETTON
- √ Iron
- √ Wooden
- √ Masonry

- √ Charts GENERAL

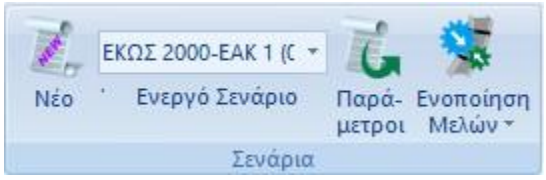
The Sizing manual is divided into 4 parts:

1. **Part 1/4 GENERAL REQUIREMENTS FOR ALL MATERIALS**
2. Part 2/4 COMMANDS FOR BETTING
3. Part 3/4 COMMANDMENTS FOR RAIL AND WOOD
4. Part 4/4 REQUIREMENTS FOR WALLING

⚠ After the completion of the model, the input of the loads, the execution of the analysis and creation of the combinations, the "Dimensioning" of the structural elements of the design follows, where the adequacy check is performed, based on the regulation selected in the "Dimensioning scenario" and the reinforcement of the concrete elements is entered.

With SCADA Pro you can dimension projects made of Concrete, Metal, Wood, Load-bearing Masonry and a combination of these.

Scenarios



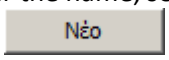
The "Scenarios" field contains the commands to create a new scenario, the list of scenarios and the sizing parameters.

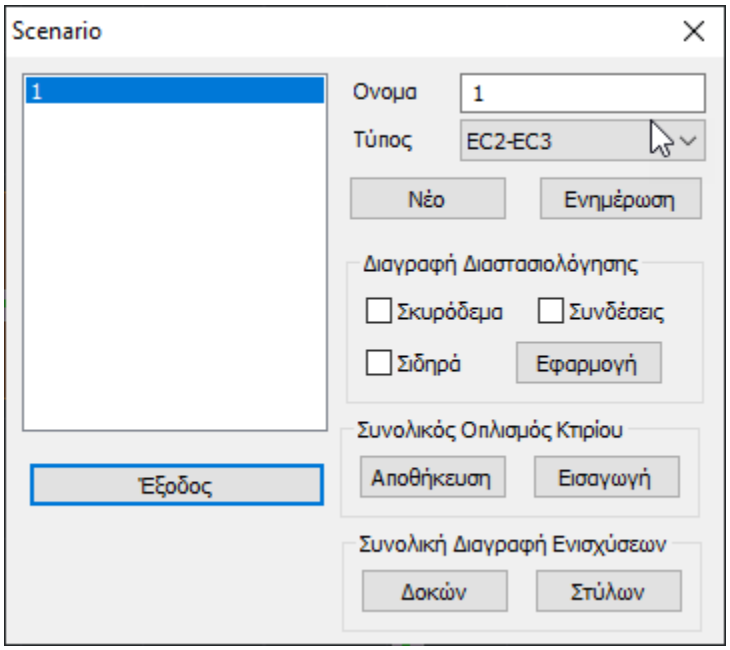
In addition, a new set of mandates has been added concerning the Consolidation of Members:

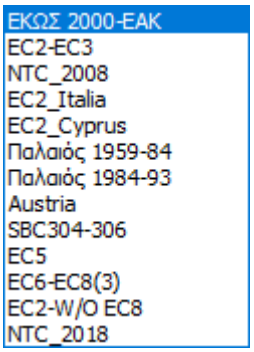
- **poles** and **beams** for Metal and Wooden
- ...for the Betonens.

1.1 New

To create a script for sizing. Enter the name, select the reference legislation and .



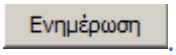




OBSERVATIONS:

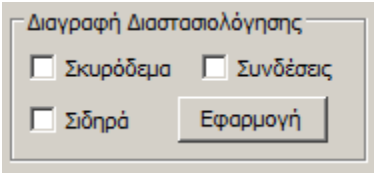
The designation EKOS 2000-EAK, EC2, or Old, refers to the method of analysis as well as the method of dimensioning of concrete sections. It goes without saying that if you have used, for example, a predefined ECK analysis scenario then you will also select ECOS 2000 - ECK dimensioning type.

For metals, EC3 is applied through the programme and is included in all scenarios regardless of the fact that there is no corresponding Greek regulation.

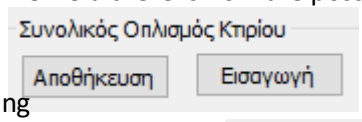
If you are modifying an existing script, select 

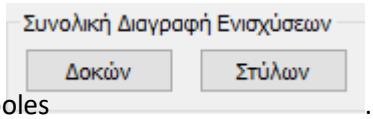
.

In the "Delete Dimensioning" field, activate the corresponding checkbox and "Apply", to delete the results of a previous dimensioning (for concrete elements, steel sections, or connections respectively), to re-dimension using other combinations, or parameters, or scenario, etc.



1.1.1 Storage and import of modified armaments

Within the NEO field there is now the possibility to store the total armament of the building 

as well as to delete in total all the reinforcement of beams and poles 

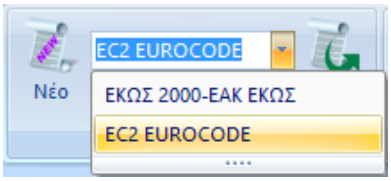
.

With the Total Building Reinforcement field, it is now possible to store the reinforcement of beams and columns with the manual modifications made by the user in order to adjust the reinforcement of an existing design for the purpose of evaluation and redesign, as well as the reinforcements given to these elements. This command is very useful in cases where the need arises to remove, modify or add a new element.

PROCEDURE:
In these cases select Save, then return to the modelling, make the modifications to the vector, run the original EC8_Greek analysis (Static or Dynamic) and come back to the dimensioning. You load the combinations again and re-dimension the entire vector to pick up reinforcements and new or modified elements. By selecting the Insert command, all the rebars and any reinforcements that you manually inserted into the pre-existing elements are brought back in.
It now remains to insert the existing reinforcements into the new or modified elements.

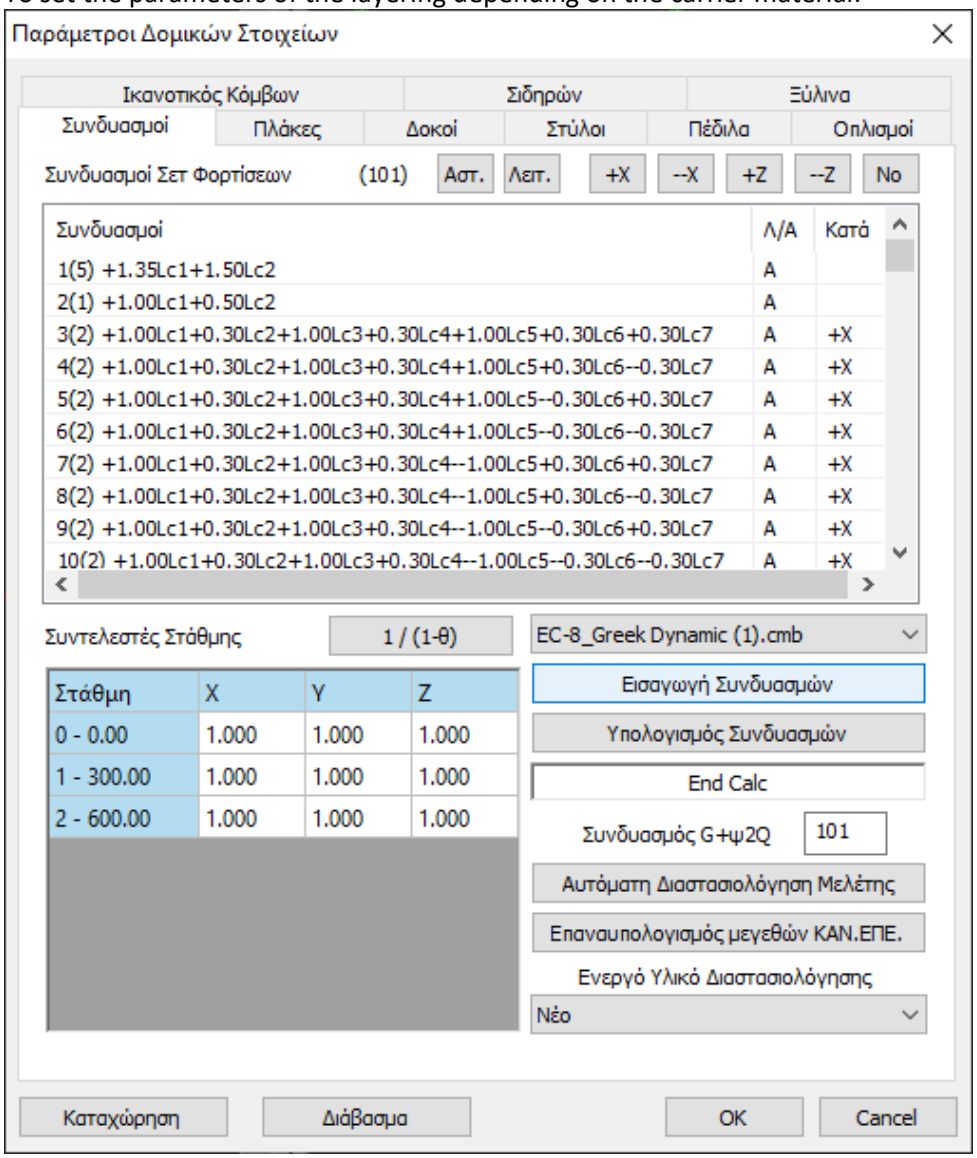
1.2 List

Includes the scripts you have created for the active script option, i.e. the script you will use.



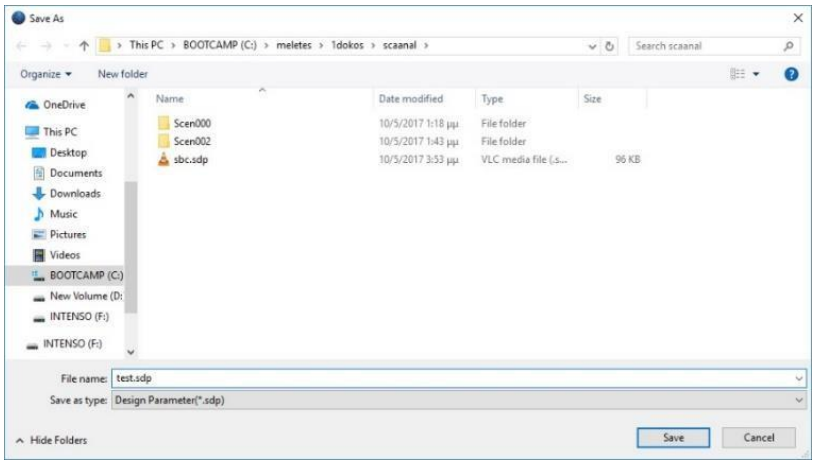
1.3 Parameters

To set the parameters of the layering depending on the carrier material:

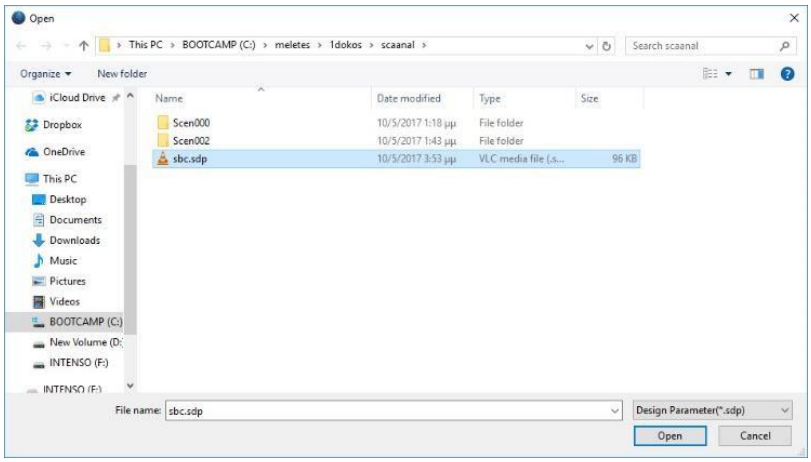


Storage of the sizing parameters of the active scenario:

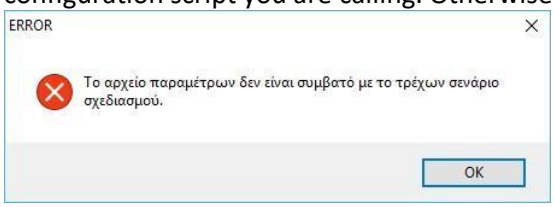
Once you have configured the sizing parameters, you can now save them in a file to use them in your next study.
Pressing the "Register" button opens the storage box



where you type a name (it is good to be relevant to the sizing scenario). The extension of these files is sdp scenery design parameters.
Similarly, with the "Read" option, you can load a previously saved parameter file into a study.



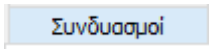
ATTENTION
A prerequisite for calling a configuration file is that the current sizing script is the same as the configuration script you are calling. Otherwise you will see the message



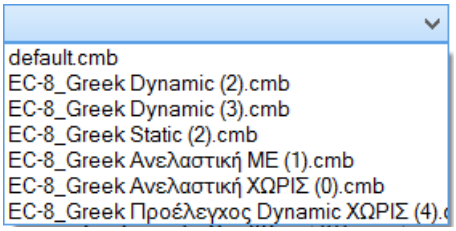
1.3.1 Combinations

OBSERVATION:

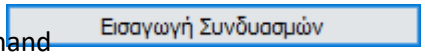
⚠ *Regardless of the material, a prerequisite for sizing is the calculation of combinations.*

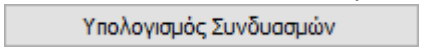


The selection of the .cmb file of the combinations registered by the analysis is either:

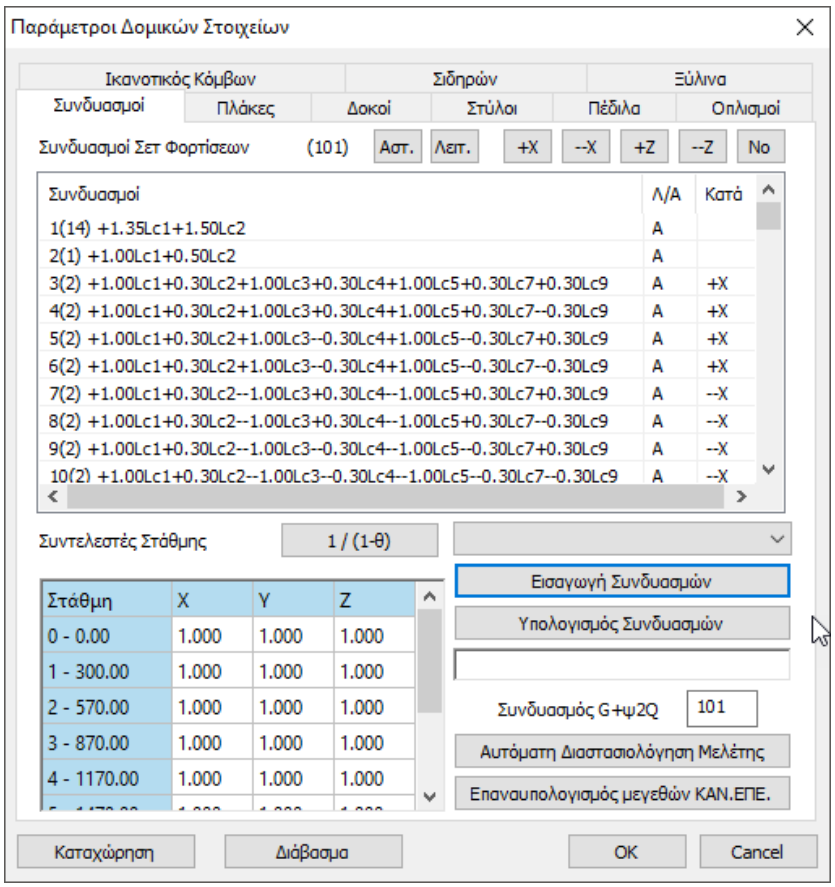
- from the list 

with automatic calculation

- through the command 

where, within study folder, you select from the registered combinations the file of combinations with you will dimension and then via the button 

button to make the calculation.

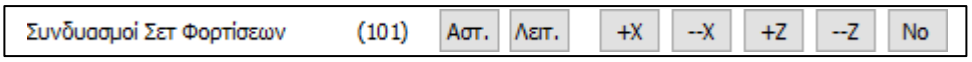


Depending on the case and the conditions that are met, you can use either the combinations of static or dynamic to dimension the superstructure (as long as in the analysis you have "opened" the springs (not the footing)). You can also have run analyses with scenarios of different codes (e.g. EAK and EC8) and by dimensioning with the respective combinations you can see the differences that result.

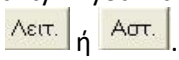
In the "Combinations" field the list of all combinations is displayed.

The first number is the number of the combination and second, in brackets, is the number of the equation from which we obtain
The "L/A" column indicates the marginal state of the combination and the "Against" column indicates which direction is involved for the satisfactory control.

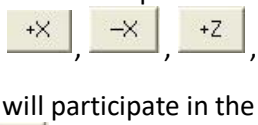
Thanks to the bar:



you can modify both the limit state and the direction.

In the column "L/A" you specify whether the specific combination is Failure or Functionality. If you want to make a change, you select the combination and press the key 

.

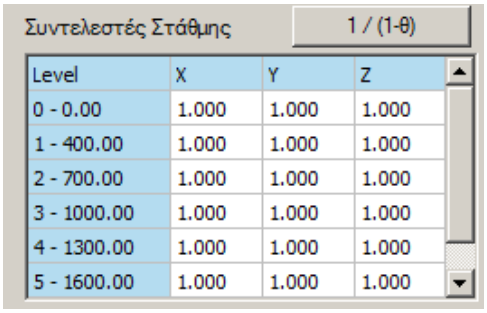
The next column "Against" is about the satisfaction check and in which direction the particular combination is involved. By selecting the corresponding button 

you select the direction in which this combination will participate in the performance of the satisfactory control. Finally, by selecting the 

you specify that the a particular combination will not participate in the satisfaction check. (See below 1.3.7 Capability Node)

1.3.1.1 Level Factors

In the field "**Level coefficients**"



You can increase or decrease, by manually entering coefficients different from 1, the seismic actions per direction and level.

- The 1 / (1-θ) button, if selected, will do the 2nd order influence check, with automatic increase of the intensities when 0.1< i< 0.2, at the levels required.

⚠ **ATTENTION:**
To take into account any modifications to the combinations, select the command again .

The field Υπολογισμός Συνδυασμών / Συνδυασμός G+ψ2Q 99 applies only to the scenarios of the Greek regulation (EKOS).

1.3.1.2 Automatic Study Sizing

The order Αυτόματη Διαστασιολόγηση Μελέτης is an automation related to **Concrete** studies and allows you to dimension the whole study with a simple "click".
Set the parameters in the fields below and select "Automatic Study Sizing". The program will automatically carry out the entire sizing process that you include in the following groups and that is otherwise followed "Step by Step".

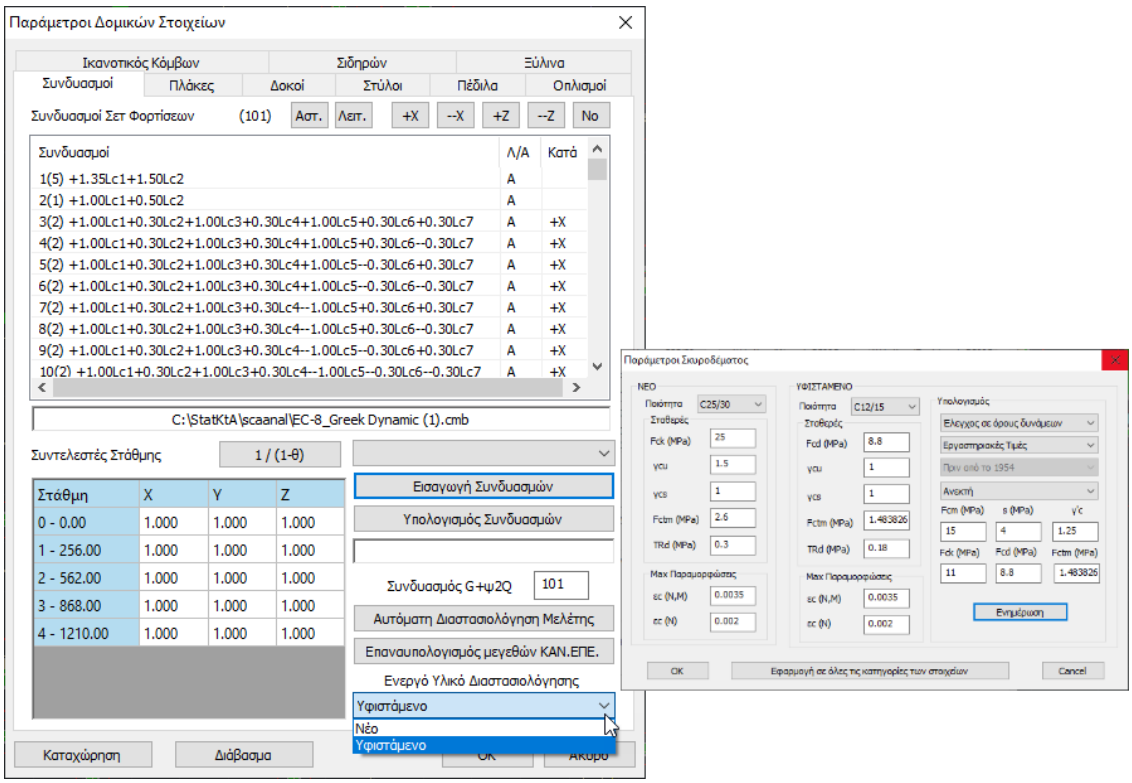
1.3.1.3 Recalculation of the sizes of CAN.Ltd.

The order Επαναυπολογισμός μεγεθών ΚΑΝ.ΕΠΕ. concerns the designs of existing structures controlled in accordance with the CEQS and is an automation that allows the recalculation of the CEQS sizes, and if the modification of the reinforcement according to the existing situation has already been carried out, to keep the existing shaped reinforcement when the characteristics of the material and the reinforcement coating are subsequently changed.

If, for example, you have already defined the existing strengths of your materials, dimensioned your design and already modified the reinforcements of the structural elements and then need to modify the strengths of your materials, you only need to make the change

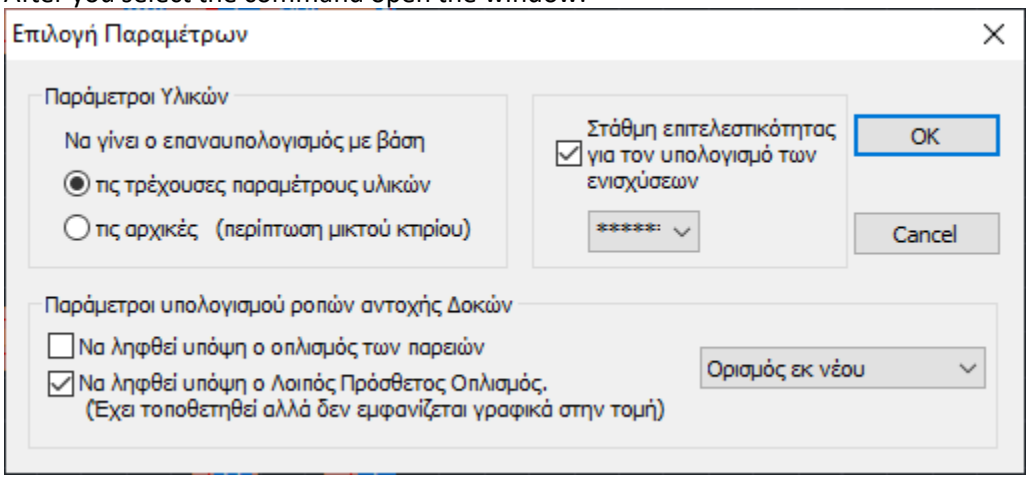
and press the button Επαναυπολογισμός μεγεθών ΚΑΝ.ΕΠΕ. to recalculate all the sizes provided for by the EIA without having to repeat the procedure. The program will automatically calculate the new sizes for all members of the study.

OBSERVATION:
During the recalculation of the .EPE sizes as well as the individual Strength Calculations in beams and columns, the strengths of the materials taken into account are the selected strengths in the parameters of the Dimensioning (either the New or the Existing):



OBSERVATION:

After you select the command open the window:



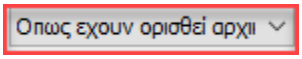
"Material parameters"

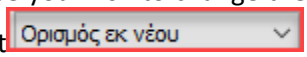
Selecting:

- The current material parameters: the program calculates the strengths according the selected parameters
- The initial ones : the program calculates the strengths according to the initial ones, i.e. with the strengths of disbursement

This command was created to overcome the problem that occurred when in a structure we needed to selectively dimension one part of the structure with some strengths and another with different strengths (mixed building), usually due to the construction of the building in different periods of time (additions, arbitrary etc.) Without this option during the recalculation the program read the materials from the current parameters and not from the dimensioning. Now this problem has been overcome by now having the second option.

"Parameters for calculation of beam strength moments"

If you do not wish to change any parameters, just 

In case you wish to change the parameters for calculating the strength moments of the beams, select 

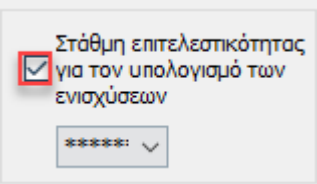
" and you have among the options:

Take into account the reinforcement of the sidewalls. If checked, the side reinforcement will contribute to the calculation of the moment of resistance.

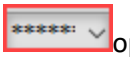
Take into account the Other Additional Armament. If checked, the additional reinforcement, which is not visible in the graphic, will contribute to the calculation of the moment of resistance.

NEW! **"Level of performance for the calculation of aid"**

Added the option to select the performance level, so that either by changing the method (inelastic - elastic), or by changing the level of performance, the reinforcements can be calculated en masse, without the need to change the reinforcement details of each reinforced member.
To enable it you must check the checkbox

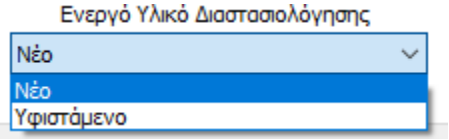


Attention!



option is for the elastic analysis, but in the elastic analysis the performance level is defined by the analysis parameters, in the CANOPE button.
Therefore, in order for the level change to be done correctly in this case, it must first be changed from there, the analysis must be re-run, the new combinations must be saved and finally entered into the sizing. Then the recalculation can be done.

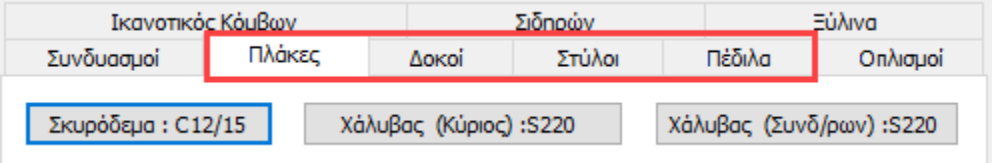
1.3.1.4 Active Sizing Material



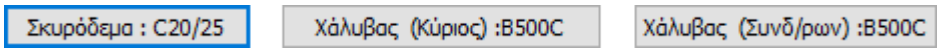
By selecting Active Sizing Material the strengths of the materials taken into account are the selected strengths in the Sizing parameters (either New or Existing).

1.3.1.5 Materials

In the fields Plates, Beams, Columns, Sheets, at the top, there is the choice of the corresponding materials:



Where,



choose the quality of the material to be used for both concrete and reinforcement (main, connectors).
By selecting a different quality for the concrete, the corresponding coefficients are automatically updated.

The new version of SCADA Pro 21 added the possibility of simultaneous definition of two material qualities for the structural elements: new and existing.

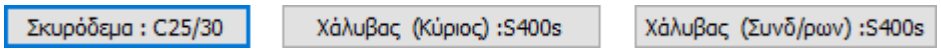
OBSERVATIONS:

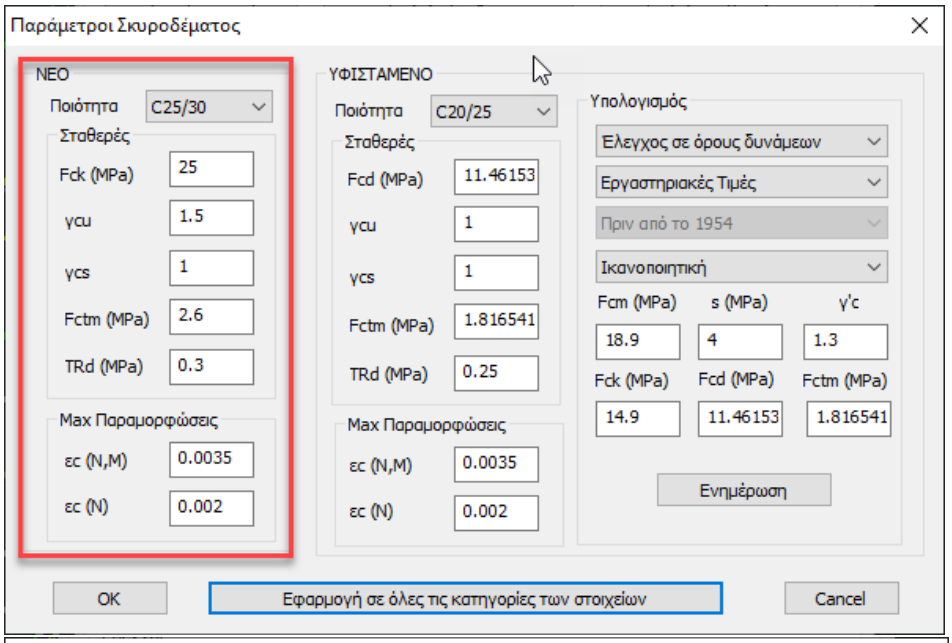
- In the existing material, the calculation of the final compressive strength is now done automatically based on the corresponding provisions of the CEE.
- Then, the attribution of the quality of the material to the elements and consequently their classification (New or Existing) is done automatically with their dimensioning and this information is now stored in each member resulting in the complete separation of new and existing elements, which gives great flexibility to the designer for further processing.
- All three windows (Concrete - Steel main - Connectors) are divided into two sections:
  1. NEW (left) &
  2. BACK (right)

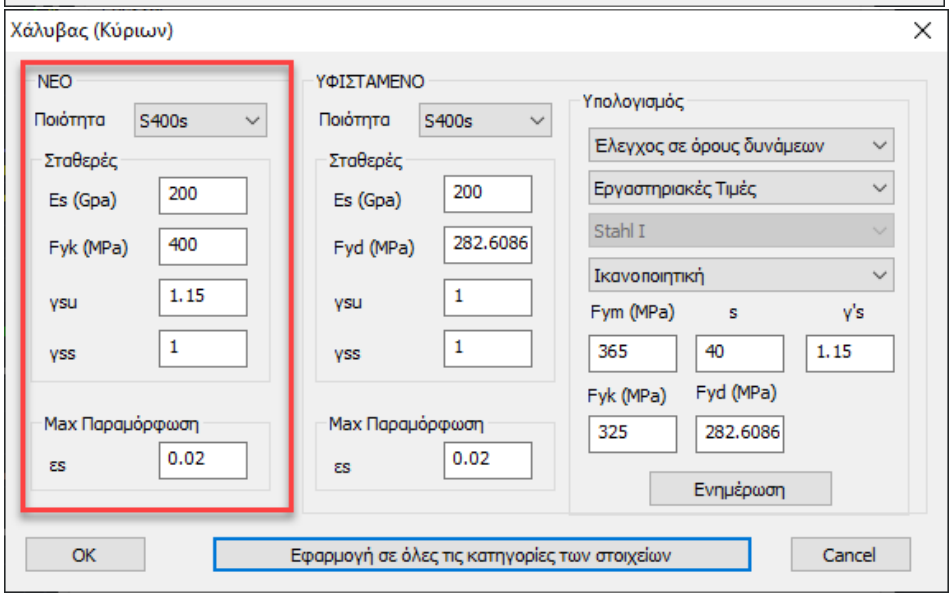
The procedure is the same for all 3 windows in detail:

1. **Purely new construction**

In the case of new construction, the definition of the material is done the same way as , completely ignoring the part of the dialogue box concerning existing material:



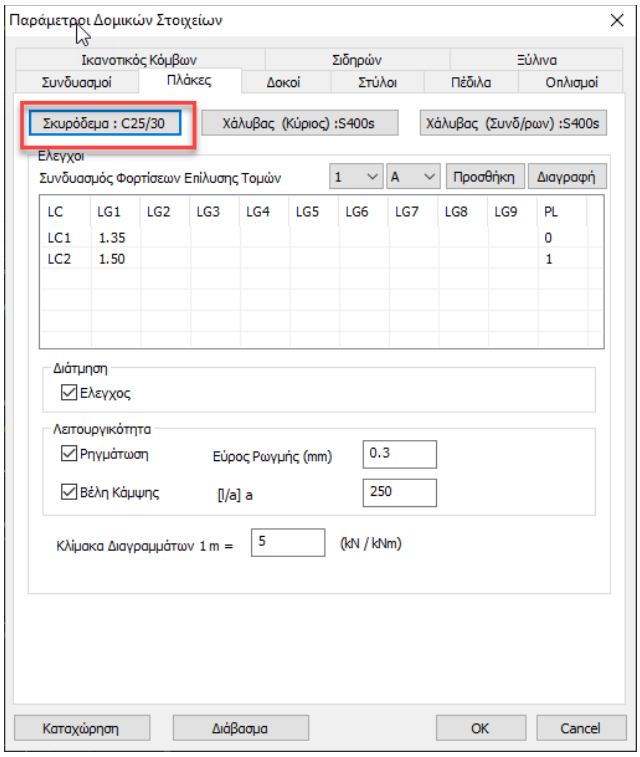




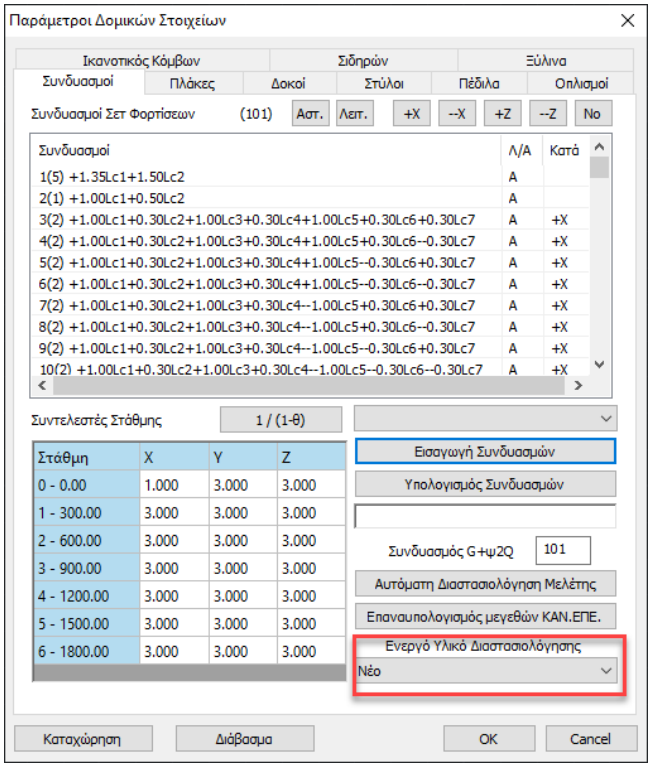
Select the material you want to use from the list.

Note

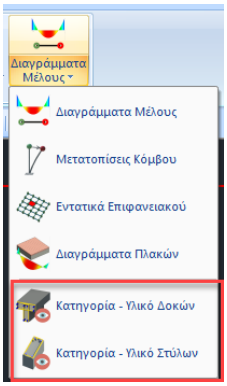
The material listed on the previous screen for each building element respectively, is ALWAYS the new material.



Then and before the initial sizing, the corresponding option in the parameter "Active Sizing Material" to be "New"



With the command "Category - Beam Material" and "Category - Pillar Material"
you can display in your operator the categorisation of the elements as well

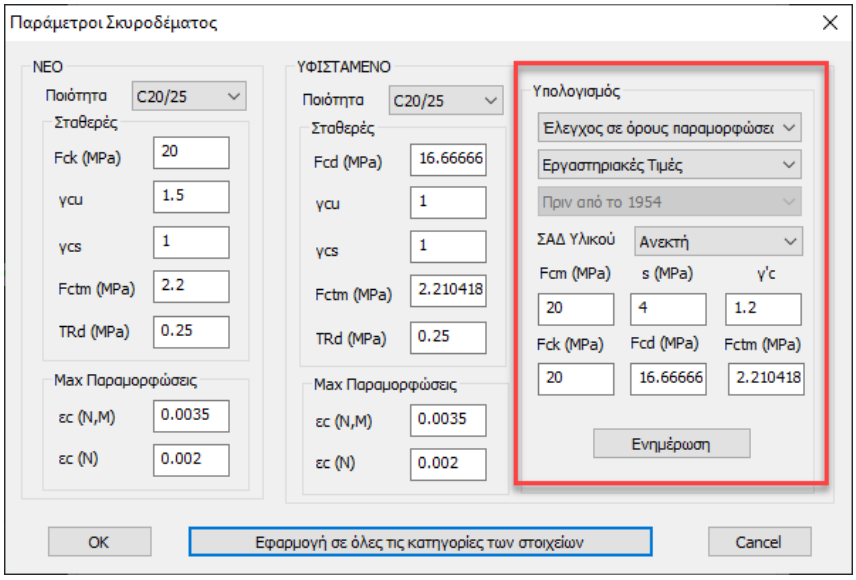


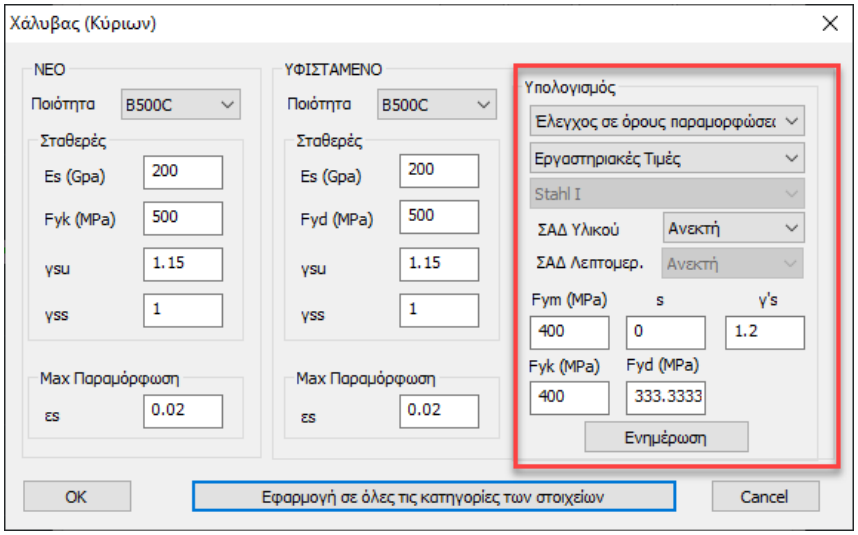
For example, in the following beam spacing

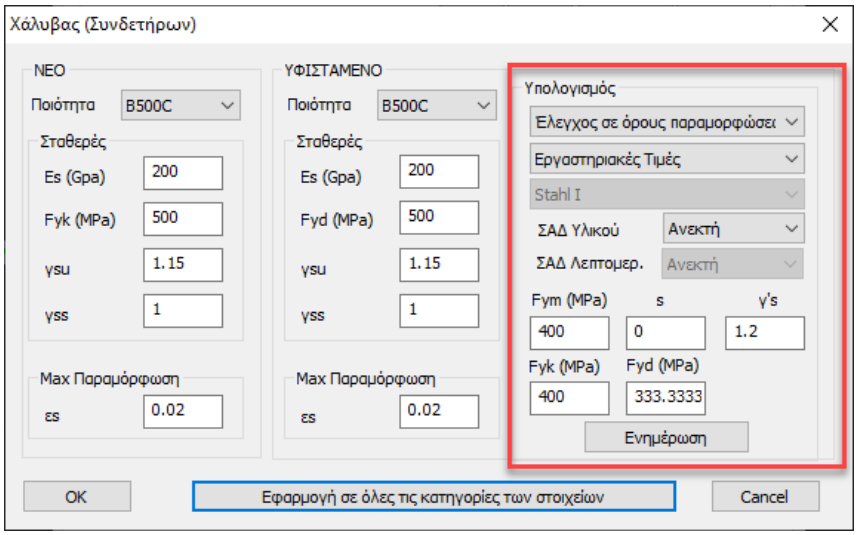
there is the designation (N), New material and the corresponding three characteristic strengths of concrete, main reinforcing steel and steel fasteners

2. Purely existing construction

The logic and course is similar to the new construction. You select a quality from the list and then, based on the provisions of the CEE, you select from the corresponding fields in the "Calculation" section







the corresponding parameters of your study based on the provisions of the relevant paragraph of the CEE. Once the selections are complete, you press the "**Update**" button and the corresponding final strengths are indicated in the fields below the material and are the ones that will be used by the program.

The coefficients γcu and γcs must remain unity.

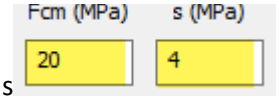
In detail:

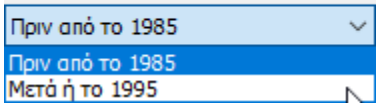
- **SKYRODEMA**

You choose whether a calculation will be made:

- in terms of Forces (Elastic analysis method q)
- in terms of Deformation (Elastic m & Inelastic method)

You choose whether to set:

- Laboratory Values - to be filled in the fields 

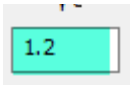
or
- In Absentia Prices (CANPE 2022) which also opens the field of choice of date logging 

and automatically completes the fixed.

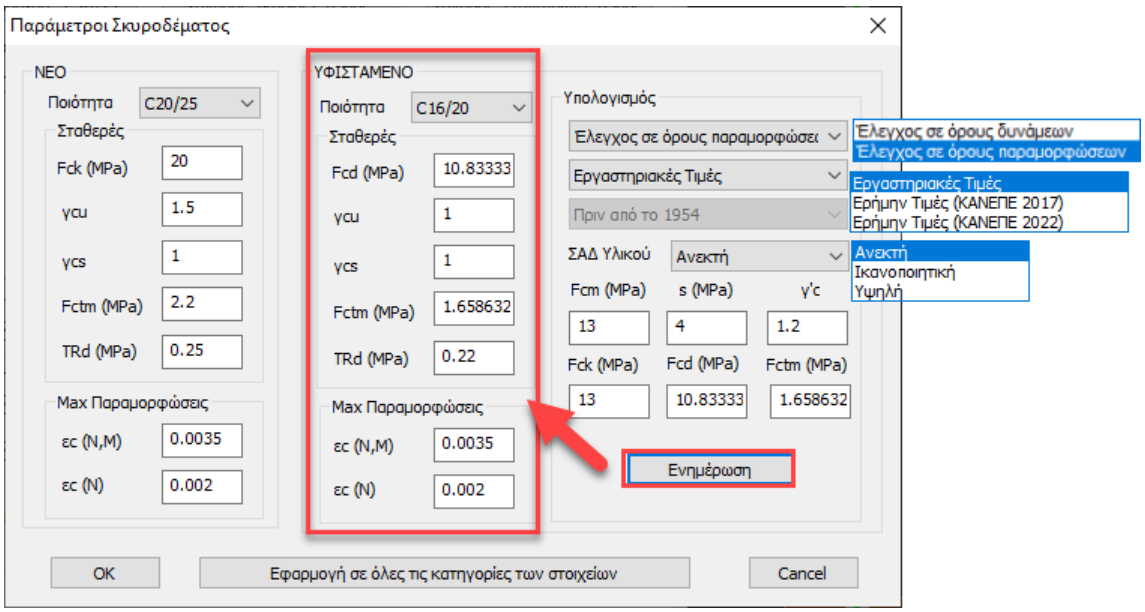
(For compatibility reasons, and the Abandoned Prices of previous revision were retained.)

The last option is the Material SDS:

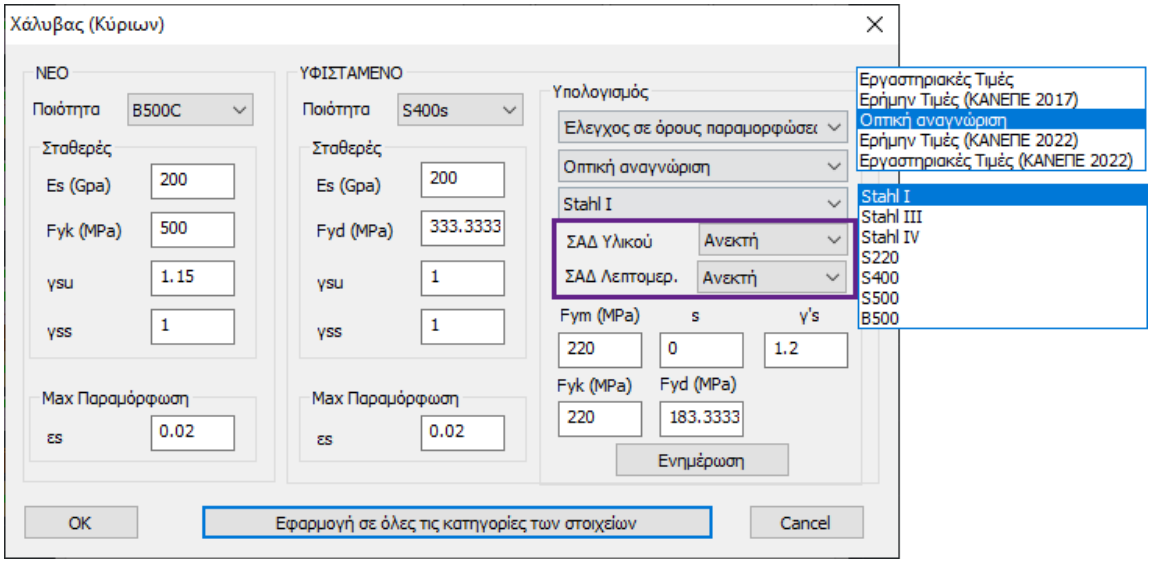
- Windy
- Iconopoeia
- High

And it completes the 

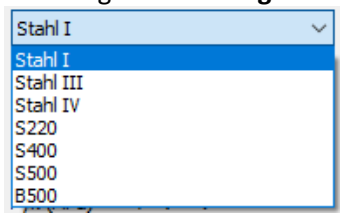
All other values are automatically filled in and the **Update** calculates the Constants for the Existing Concrete.



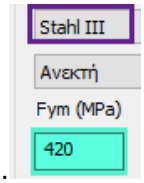
HALYVAS:



When determining the strengths of Steel (main & fasteners) there is the additional presence of **Optical Recognition**.
Selecting **Visual Recognition** opens the list of steel grades that you select



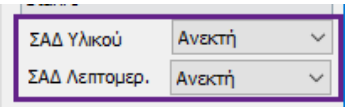
and automatically fills Fym .



All other values are automatically filled in and with the **Update** the Constants for the Existing Steel (main & fasteners) are calculated.

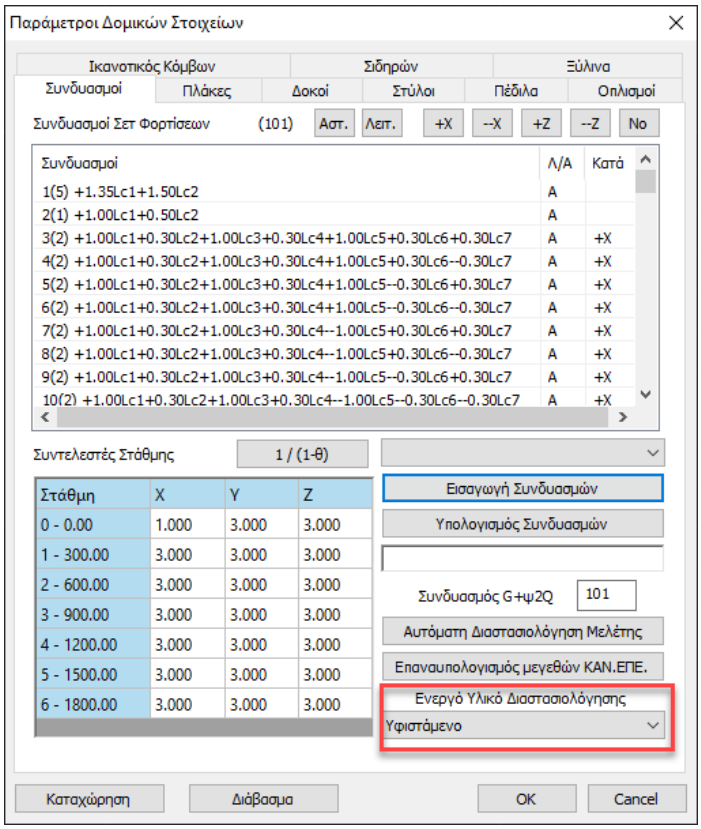
3h revision of the EIA CIP

For steel, the material safety factor γs now depends not only on the material data reliability level but also on the detail data reliability level. These two new options have therefore been introduced:

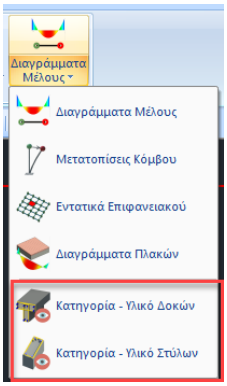


The option Laboratory Values CANEPE 2022 was also introduced, where the cs is derived from a combination of the two SDSs and the option Absent Values CANEPE 2022 was also introduced where the requirement for steel is that the material SDS is satisfactory (rather than tolerable which was in the previous revision).

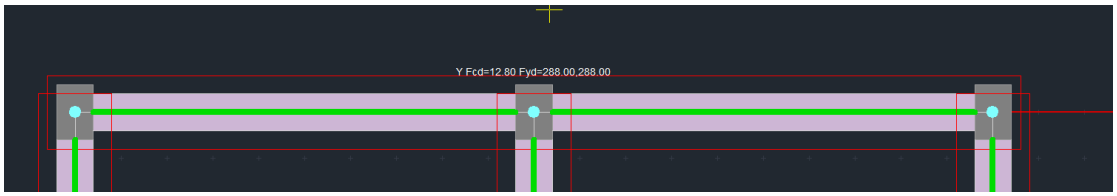
Then and before the initial sizing, as in the new building, the corresponding option in the parameters must be "Existing". Then and as usual you adjust the reinforcements of the structural elements.



With the command "Category - Beam Material" and "Category - Column Material" you can display the categorization of the elements and the strengths of the materials in your organisation.



For example, in the following beam spacing



there is the designation (Y), Existing material and the corresponding three characteristic strengths of concrete, main reinforcing steel and steel fasteners.

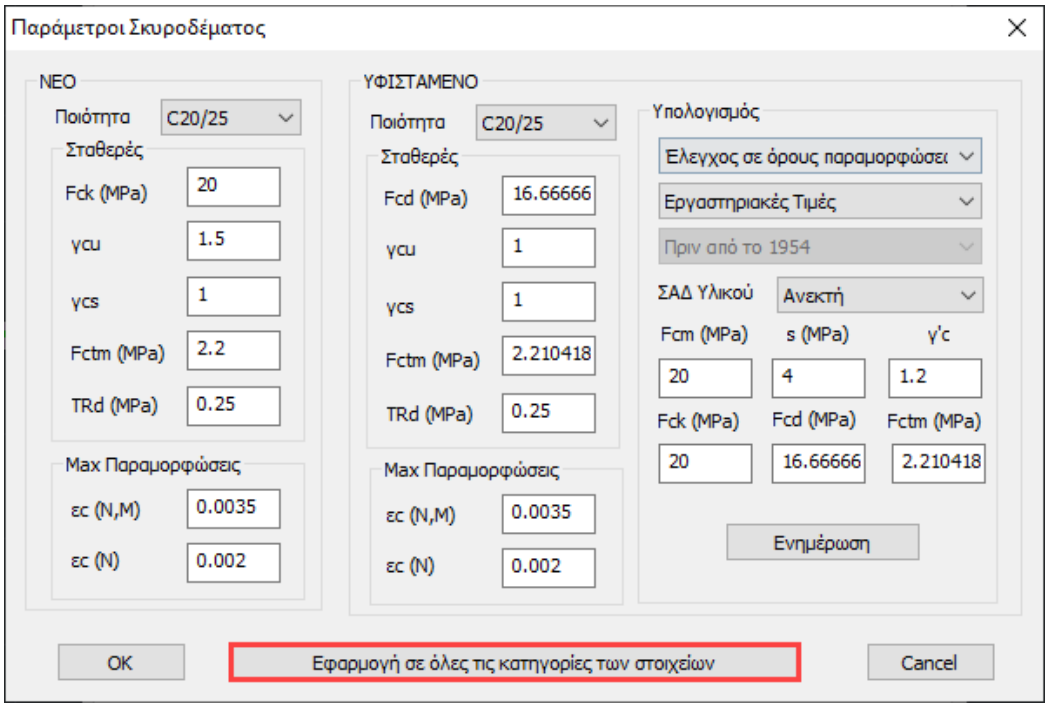
3. Existing construction and addition of new construction

The procedure followed here is a "mixture" of the first two previously mentioned.
You define the two materials (New and Existing) and then do selective sizing in two phases:

- You define the Existing as the active sizing material and selectively size only the Existing elements and then
- You define New as the active dimensioning material and dimension the New.

OBSERVATION:
If you want to change the category of an item from New to Existing, you simply dimension it selectively with the corresponding material active.

Finally, by selecting the "Apply to all categories of items" button,



the materials you specify for one category of building elements (e.g. Pillars) are automatically copied to all other categories of building elements and so you do not need to specify them again, provided of course that the qualities are the same.

This feature applies to both new and existing hardware.

1.3.1.6 Corrosion

It is now possible to calculate the influence of reinforcement corrosion on the mechanical characteristics of structural elements (see EPC Annex 7F) by entering the initial and measured diameter. The program automatically calculates the corrosion rate Xcor, the corresponding degradation factors rcor and all the corresponding degraded quantities (resistances and deformations) according to the type of analysis.

An important modification introduced by the 3rd revision of CEE is the reduction factors for the mechanical characteristics of structural elements with corroded reinforcement rcor

In , Annex 7F proposes indicative values of the softening factor in relation to the corrosion rate Xcor, calculated from the following relationship

$$X_{cor} = \frac{\Delta A_s}{A_s} = \frac{A_s - A_{s,cor}}{A_s} = \frac{D_s^2 - D_{s,cor}^2}{D_s^2}$$

Where
Ds : initial, nominal diameter of the reinforcement
Ds,cor : measured diameter of corroded reinforcement

The deterioration factors shall be calculated as follows:

CAMPSY

- The reduction factor, $r_{cor,Fy}$, of the resistance moment $F_{y,cor}$, of the structural element where corroded reinforcement occurs relative to the resistance moment of the element without corroded reinforcement, F_y, is defined as:

$$r_{cor,Fy} = \frac{F_{y,cor}}{F_y} = 1,00 - 1,30\,X_{cor} \qquad \text{(F.2)}$$

- The reduction factor, $r_{cor,\vartheta y}$, of the yielding strain, $\vartheta_{y,cor}$, of of a structural element where corroded reinforcement occurs with respect to the deflection at yield of the element without reinforcement corrosion, ϑ_y, is defined as:

$$r_{cor,\vartheta y} = \frac{\vartheta_{y,cor}}{\vartheta_y} = 1,00 \qquad \text{(F.3)}$$

In practice, it is assumed that the deformation in the leakage remains constant regardless of the degree of corrosion.

- The reduction factor, $r_{cor,\vartheta u}$, of the strain at failure, $\vartheta_{u,cor}$, of of a structural element where corroded reinforcement occurs relative to the deformation at failure of the element without reinforcement corrosion depends on the magnitude of the axial force, *v*, that stresses it and is defined as:

$$r_{cor,\theta u} = \frac{\theta_{u,cor}}{\vartheta_u} = 1,00 - 2,85X_{cor} \quad \text{for } v \leq 0,20 \qquad \text{(F.4.a)}$$

$$r_{cor,\theta u} = \frac{\theta_{u,cor}}{\vartheta_u} = 1,00 - 3,50X_{cor} \quad \text{for } 0,20 < v \leq 0,40 \qquad \text{(F.4.b)}$$

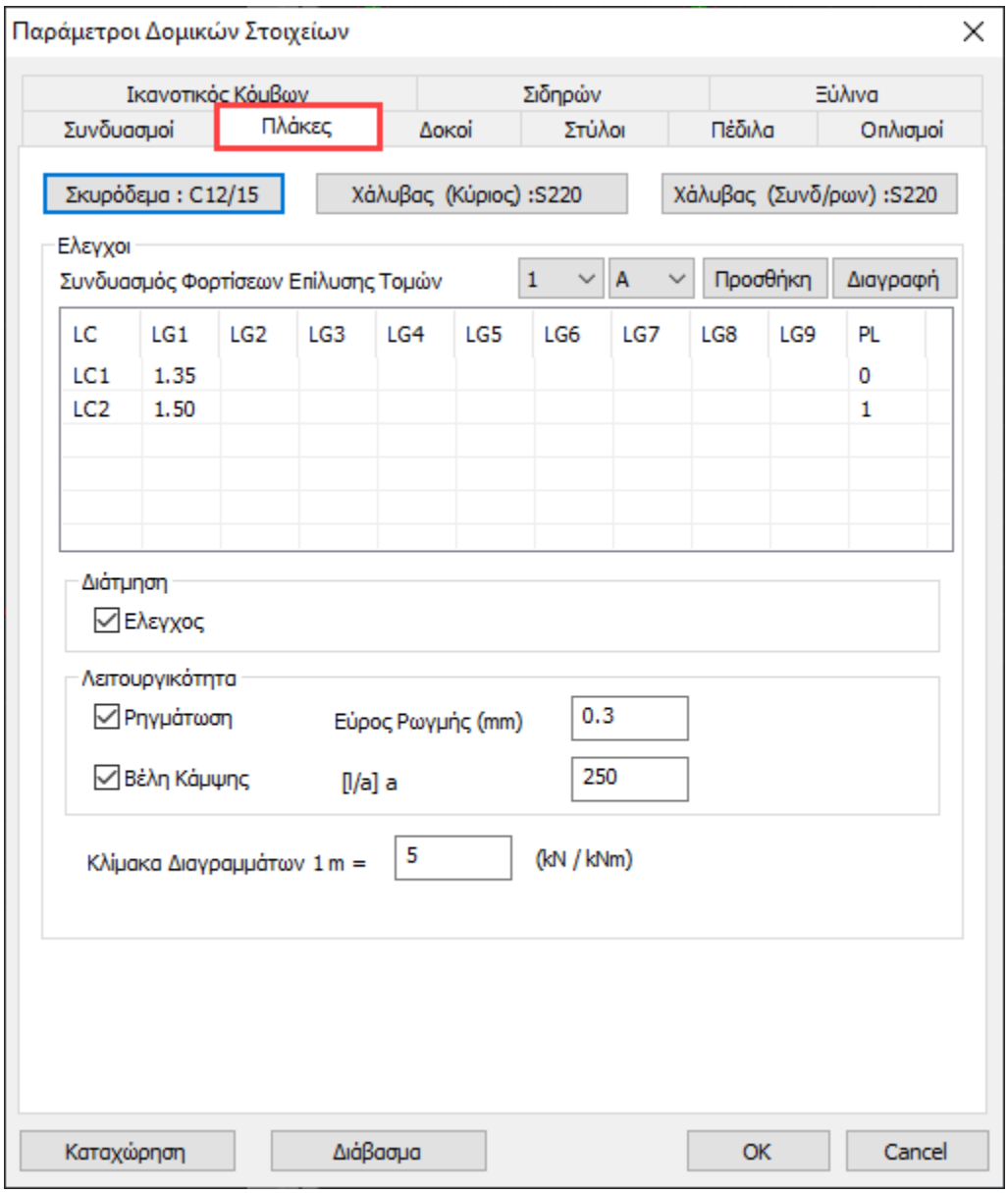
For $v > 0,40$ not enough evidence from the literature has been evaluated. It is reasonable, however, to
it could be assumed that the values of $r_{cor,\vartheta u}$ would be smaller than those obtained from relation F.4.(b) above.

Note that, in each case, $\vartheta_{u,cor}$ will be taken to be greater than or equal to ϑ_y.

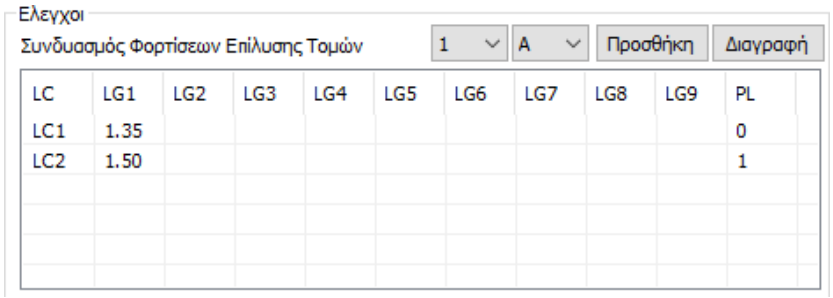
For the shear strength of structural elements, the provisions of Annex 7C of the Regulation apply, where the shear resistance due to the contribution of fasteners (V_w) is calculated on the basis of their corrosion-reduced cross-section divided by a safety factor of 1,50. For degrees of corrosion of the fasteners greater 35 %, the following shall apply
obtain $V_w = 0$.

In the program, the above procedure is carried out automatically by entering the reinforcement details of the beams and columns (see fig. Use CHAPTER A**:**
'DOG ARMS' and Chapter B**:** 'STOCK ARMS')
the initial and the measured diameter and the program will automatically calculate the corrosion rate Xcor, the corresponding degradation factors rcor and all the corresponding take-off values (resistances and deformations) depending on the type of analysis.

1.3.2 Plates



In the field

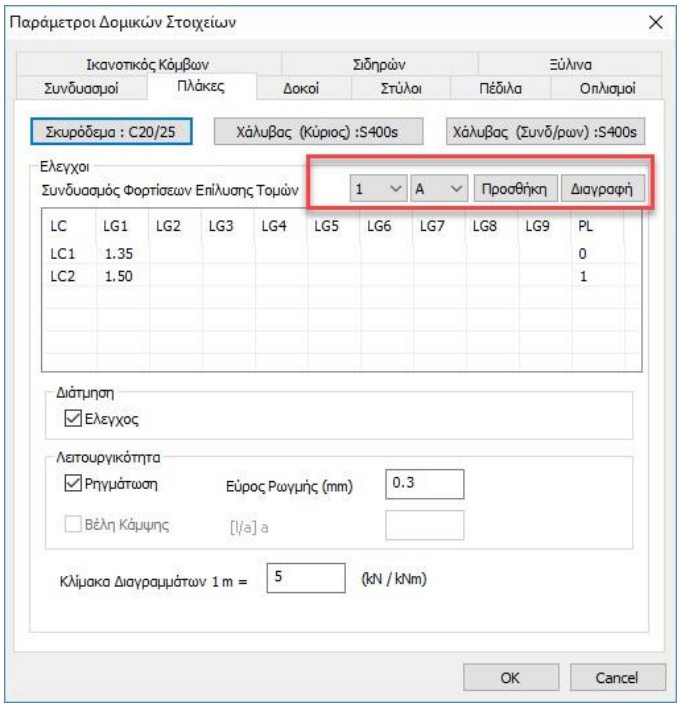


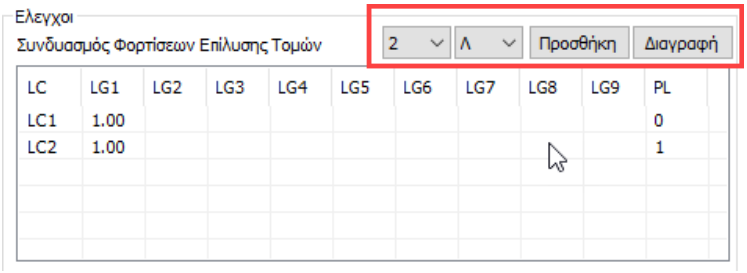
The coefficients for the permanent and mobile loads, which will be taken into account in the solution of the plate sections, are displayed and can be modified by the user.

- The PL column concerns the **Adverse** Plate **Loadings**. A factor of 0 means that the loading loads of the corresponding line will not be included, 1 means that they will be included.
- Select the checks that you wish to be carried out on plates (check the corresponding checkboxes).
- Set the crack width for the cracking test and the scale for displaying the diagrams.

1.3.2.1 Sizing of the plates with input by the designer of more than one combination.

The new version of SCADA Pro has added the possibility of entering more than one combination for the sizing of plates.
In the dimensioning configuration window and specifically in the Plates:

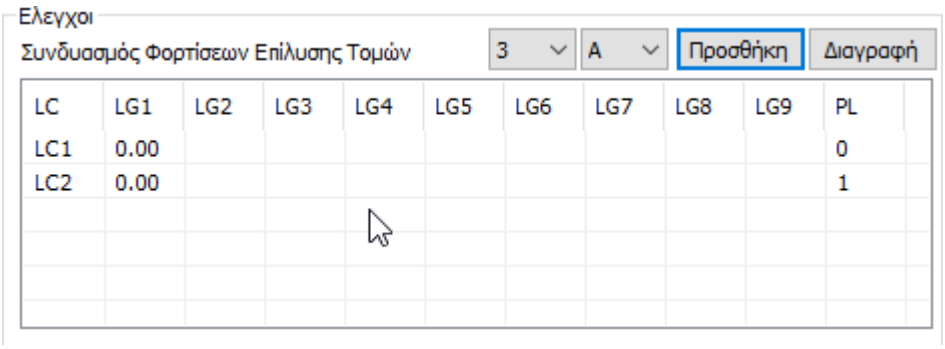




The mask is opened in the same way for all scenarios except that, for the Greek regulations scenario only one combination of failure is predefined, and for the other scenarios one more combination of functionality has been added.

OBSERVATION:
For the Greek regulation (EKOS) the combination of Functionality is not meaningful because the deformation control is based on the dimensions and the type of support of the plates, a process that is done in the pre-estimation of the minimum thickness and has nothing to do with the dimensioning.

To create a new combination, we press the "Add" button. The number becomes 3 and the load factors become 0.

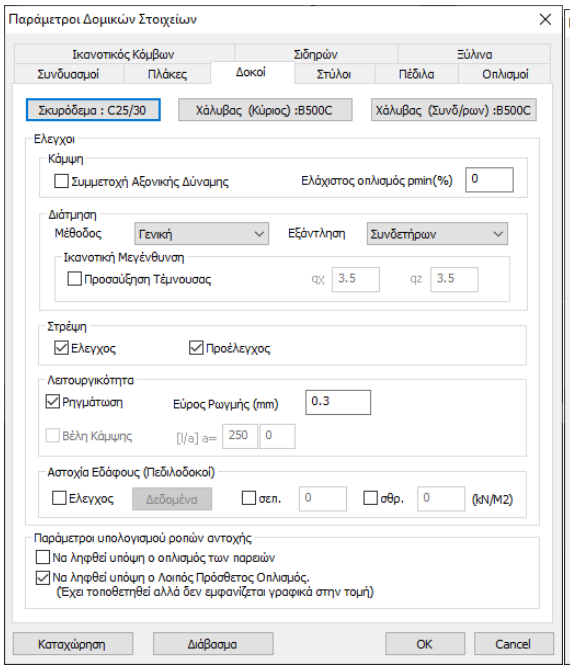


Now we can set the coefficients and whether the combination will be Failure or Functionality.

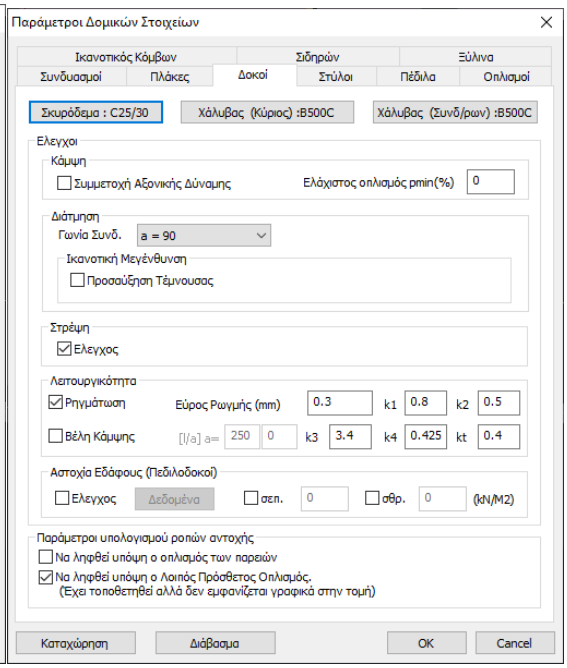
In the same way we can define as many new combinations as we want or modify those already created. The Program will dimension with the worst moment with respect to the failure combinations and with the corresponding sizes will do the deformation checks from the functionality combinations. We can also use the "Delete" button to delete the combination or combinations we have created. Only the 1 & 2 predefined combinations are not deleted.

1.3.3 Beams

For **EKOS-EAK** scenario



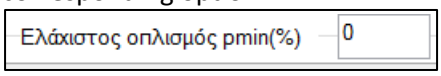
For **EUROCODE** scenario



In the **Beams** field, select the checks you wish to perform on the beams (check the corresponding checkboxes):

1.3.3.1 Bend

For the check in "Bending" decide on "Axial Force Involvement" by checking or unchecking the corresponding option.



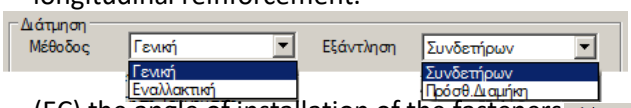
enables the designer to set his own price for the minimum reinforcement percentage.

- If a value is given then the program will take it into account for the minimum reinforcement, and if the field is left blank, the program takes into account the value of Regulation.

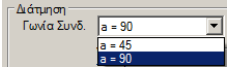
1.3.3.2 Shear

For the control in "Shear" decide:

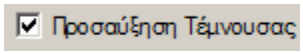
- (EAK) you specify the method by which the Shear will be calculated. It is possible to calculate the shear either by the general method (paragraph 11.2 of E.K.O.S.) or by the alternative method (paragraph 11.2.5 of E.K.O.S.), as well as the possibility to receive the cutting force either by fasteners or by placing additional longitudinal reinforcement.



- (EC) the angle of installation of the fasteners



1.3.3.3 Economic growth

In the field "Iconic Magnification" check the checkbox 

if a satisfactory check is required:

- (EAK) If you select the increase of the intersection, you have the option to set different seismic behaviour coefficient q per direction (qx and qz).
- (EC8 §5.4.2.2.2.) structures with DCM and DCH require "Capacity Design"

The value of $_{\gamma RD}$ for overstrength is automatically set as a function of ductility: $_{\gamma RD=1}$**,0** for **DCM /** $_{\gamma RD=1}$**,2** for **DCH**.

1.3.3.4 Cranking

For the control in "Rotate" check the checkbox:

- (EAK) "Control" and "Pre-Control".
  When you have only the "Control" option enabled, the contribution of the concrete, i.e. $_{Vcd}$ **= 0** (i.e., it is assumed that the shear stress received by the concrete is zero) and then the calculation of the connectors is performed.

ATTENTION:
If both the "Check" and "Pre-check" options are enabled, the program checks if a "Torsion Check" is required.
- If a check is required, set $_{Vcd}$= 0 when calculating the shear connectors.
- If no check is required, the program determines the value of $_{Vcd}$ and then calculates the connectors.

- Activating only the "Pre-check" option without simultaneously activating of the "Rotate" option is meaningless.
- (EC) "Check" The program considers $_{Vcd}$= 0 and calculates the clips.

The next section concerns the checks in the limit state of functionality, OKL, you specify whether you want to check for cracking by ticking or not the corresponding option, as well as the minimum crack width by entering its value in the box next to the indication "Crack width (mm)". The program shall take as default value a crack width equal to 0,3 mm, as specified in the relevant chapter of the Eurocode.

1.3.3.5 Functionality

For the OKL, activate the controls "Expansion" and

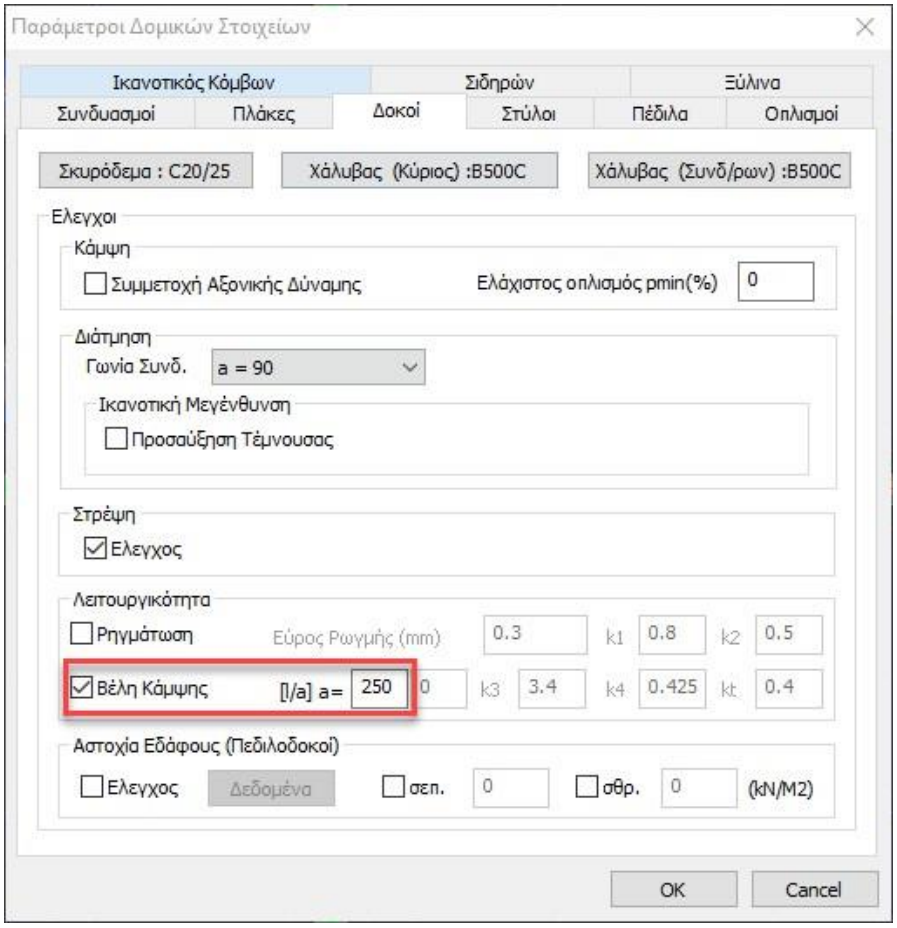
- (EAK) possibly modify the crack width (EC)

- (EC2) modify, if necessary, the constants K*

*EC2§7.3.4
K1 : is a coefficient to take into account the congruence properties of the reinforcement with congruence:
K1=0,8 for high affinity bars
K1=1,6 for bars with a practically smooth surface
K2 : is a coefficient to take account of the distribution of deformations K2=0,5 for bending
K2=1,0 for net tensile strength
K3=3,4
K4=0,425

The control of "Bending Needles" in the beams according to EC2 is done by selecting the corresponding control in the beam sizing parameters.

This is where the upper limit (l/a) of the deformations is determined.



The results of the check are displayed at the end of the investigation file

```
ΕΛΕΓΧΟΣ ΠΑΡΑΜΟΡΦΩΣΕΩΝ
ΔΟΚΟΣ 1 5 b=0.25 h=0.50 c=0.03
Συνδ.      M          N          Du1          Du2
100     -21.356     -0.000     0.00443     0.00011     0.00454     0.01500
101     -20.315     -0.000     0.00432     0.00012     0.00443     0.01500
102     -23.958     -0.000     0.00471     0.00010     0.00481     0.01500
```

EXAMPLE:

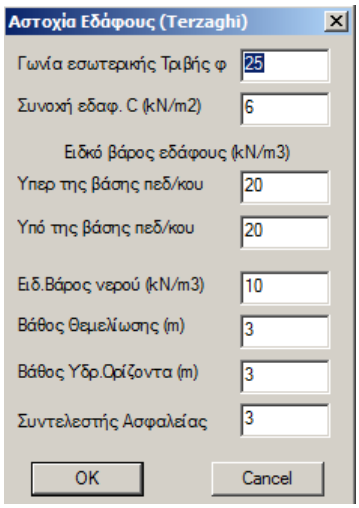
The above beam is checked for the 3 combinations of functionality (100, 101, 102) and the value Du1 is the maximum deformation of the element as obtained from the calculation of its elastic line.

The magnitude Du2 is the deformation as calculated based on relation 7.18 of EC2. The sum (Du1+Du2) is then calculated and entered and in the fourth column is the upper limit l/a.

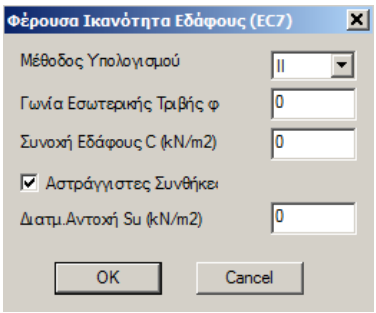
It must be (Du1+Du2)< l/a , so that there is no problem.

1.3.3.6 Pedestals

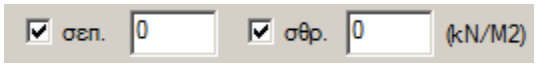
Activate "Check" , which concerns the calculation of the limit load of the footings of a foundation, to activate the "Data" command, which is only filled in if you have a soil engineering study, and not arbitrarily.



The calculation of the limiting load of the Pedestal is performed according to the general Terzaghi formula divided by the safety factor of the Pedestal parameter table which is recommended to be not less than 3.



The calculation of the limiting load of the Pedestal is performed according to the selected Calculation Method according to EC7.



two options concerning the designer's determination of the allowable soil stress. Select with the mouse the corresponding option next to "Shep (KN/m2)" and in the adjacent box type the value of the allowable stress that we want the program to take into account. Then select with the mouse the corresponding option next to "shtr (KN/m2)" and in the adjacent box type the value of the fracture stress that we want the program to take into account.

OBSERVATIONS:

The above parameters in the "Soil failure" section refer to the strength of the soil for footing with footings and the calculation is done either with Terzaghi/EC method, or by entering the allowable stress and the fracture stress in the corresponding fields.

The soil bearing capacity check based on EC7 for the footings is shown in the results in the following format:

```
|-------------ΕΛΕΓΧΟΣ ΦΕΡΟΥΣΑΣ ΙΚΑΝΟΤΗΤΑΣ ΕΔΑΦΟΥΣ (EC7)-------------------------|
|Μέθ.Υπολογισμού : ΙΙ   Γωνια εσωτερ.τριβης φ= 4.00   Αστράγγιστες Συνθήκες:Ναι|
|Συνοχή Εδάφους C(kN/m2)=12.0   Διατμητική Αντοχή Su(KN/m2) =   20             |
|Λόγος Επάρκειας λgo =   0.69                                                  |
--------------------------------------------------------------------------------
```

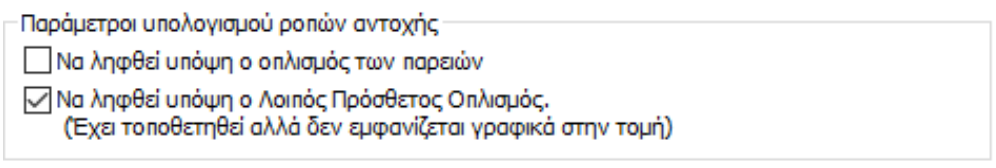
Also shown in the exploration are the detailed test results for each combination

```
ΕΛΕΓΧΟΣ ΦΕΡΟΥΣΑΣ ΙΚΑΝΟΤΗΤΑΣ ΕΔΑΦΟΥΣ (EC7)
ΔΟΚΟΣ 1 41
  1 N= -74.76 My=   0.00 Mz=  44.88 Vy=   0.00 Vz=   0.00 qEd= -77.82 qRd= 111.99 HEd=   0.00 Rd=  10.49 l=0.69
  2 N= -48.13 My=   0.00 Mz=  28.79 Vy=   0.00 Vz=   0.00 qEd= -53.57 qRd= 111.73 HEd=   0.00 Rd=  10.57 l=0.48
  3 N=  -9.09 My=   0.00 Mz=  44.52 Vy=   0.00 Vz=   0.00 qEd= -47.52 qRd= 104.20 HEd=   0.00 Rd=  34.50 l=0.46
  4 N= -13.54 My=   0.00 Mz=  42.33 Vy=   0.00 Vz=   0.00 qEd= -47.79 qRd= 104.20 HEd=   0.00 Rd=  34.50 l=0.46
  5 N=  -8.66 My=   0.00 Mz=  45.02 Vy=   0.00 Vz=   0.00 qEd= -47.40 qRd= 104.20 HEd=   0.00 Rd=  34.50 l=0.45
  6 N= -13.12 My=   0.00 Mz=  42.84 Vy=   0.00 Vz=   0.00 qEd= -47.67 qRd= 104.20 HEd=   0.00 Rd=  34.50 l=0.46
```

The results show the worst ratio.

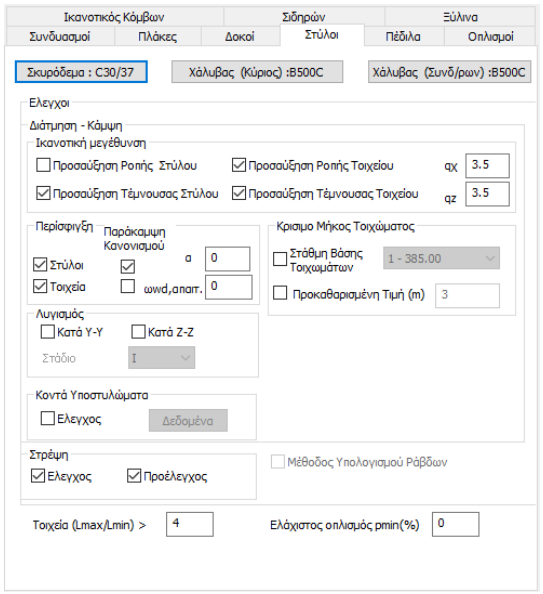
1.3.3.7 Parameters for calculating the strength moments of the beams

You can now specify <u>for existing buildings</u>, whether in the calculation of the beam strength moments based on the interaction diagrams, you want to take into account the reinforcement bars of the side walls, as well as any additional reinforcement that exists and which does not appear in the beam cross-section.

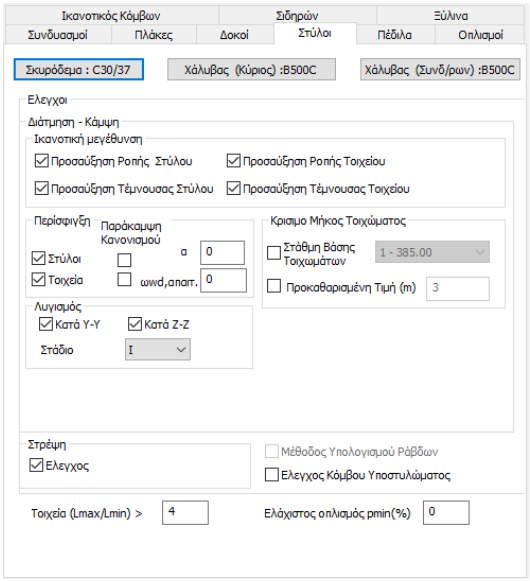


1.3.4 Pillars

For **EKOS-EAK** scenario



For **EUROCODE** scenario

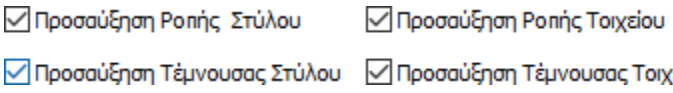


The **Columns** field refers to columns and walls. Select the checks you wish to perform (check the corresponding checkboxes).

1.3.4.1 For EKOS-EAK scenario

1.3.4.1.1 Economic Growth

In the field "Capacitive Magnification" the possibility of increasing shear and/or torque during the capacitive check of the nodes is given by selecting next to the corresponding



case .

In the box next to 'qx' and 'qz' the value of the coefficient q is given, which may be varied in each direction and which is the upper limit of the increase in shear or moment.

1.3.4.1.2 Cranking

For the control in "Rotate" check the checkbox:

When you have only the "Check" option enabled, the contribution of the concrete is ignored, i.e. V_{cd} = 0 (i.e., it is assumed that the shear stress received by the concrete is zero) and then the calculation of the connectors is performed.

⚠ **ATTENTION:**

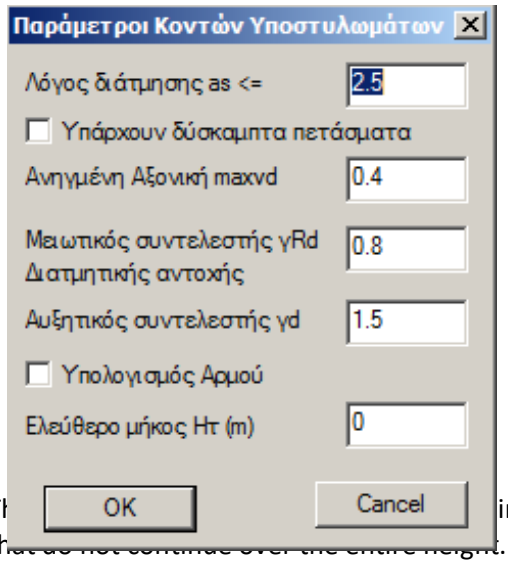
If both the "Check" and "Pre-check" options are enabled, the program checks if a "Torsion Check" is required.

- If a check is required, set V_{cd}= 0 when calculating the shear connectors.
- If no check is required, the program determines the value of V_{cd} and then calculates the connectors.

- Activating only the "Pre-check" option without simultaneously activating of the "Rotate" option is meaningless.

1.3.4.1.3 Near Pillars

There is also a check for "Close Pillars". Activate the checkbox in "Check" and enter the parameters for short columns in the data:



The first option relates to the limit of the shear ratio below which a column is considered short. The default value under the Regulation is 2,5.

The next option is whether or not stiff pedestals are present (their existence requires the second condition of paragraph 18.4.9.1 to be satisfied)

The next option is the limit of the reduced axial load. For short poles the default by regulation is 0.40. The next option is the shear strength reduction factor and its default value by regulation is 0.8.

Th in the calculation of the joint in the case of wall fillings th .
If you want to calculate a joint, select the check box "Joint calculation"
Finally you set the height of the wall Ht in meters. This height is taken into account in the calculation of the joint.

After you have sized the poles, in the results display the last section is for the previously mentioned checks.

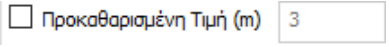
```
|-------------------------------------------------------------------------------|
|           Ε Λ Ε Γ Χ Ο Σ   Κ Ο Ν Τ Ο Υ    Υ Π Ο Σ Τ Υ Λ Ω Μ Α Τ Ο Σ           |
|        Λόγοι διάτμησης ΒΑΣΗΣ          ||       Λόγοι διάτμησης ΚΟΡΥΦΗΣ        |
|Msd(kNm)|Vsd(kN)-|h(cm)|---αs---|Συνδ.||Msd(kNm)|Vsd(kN)-|h(cm)|---αs---|Συνδ.|
|   -1.77|    3.98| 0.35|y= 1.27 | 86  ||   -4.18|   -5.90| 0.35|y= 2.02 | 24  |
| -235.89| -100.24| 0.75|z= 3.14 | 42  ||   -0.26|  -30.42| 0.75|z= 0.01 | 25  |
|Εάν αs <=  2.50 Απαιτείται ικανοτικός έλεγχος στην αντίστοιχη διεύθυνση       |
|----------------Υ Π Α Ρ Χ Ο Υ Ν   Δ Υ Σ Κ Α Μ Π Τ Α   Π Ε Τ Α Σ Μ Α Τ Α-------|
|                ΙΚΑΝΟΠΟΙΗΣΗ ΣΥΝΘΗΚΗΣ Mv + q/1.5 * (Med) <= MRd                  |
|----ΘΕΣΗ----|-Mv(kNm) + q/1.5  *(Med)(kNm)------------?-MRd(kNm)--|Συνδ.|-----|
|ΒΑΣΗ   Y    |   18.61 |  2.33  |  -23.92 =  -37.22    < -461.43   |  36 | Ναι |
|ΚΟΡΥΦΗ Y    |   -5.13 |  2.33  |   -0.96 =   -7.37    < -464.00   |  20 | Ναι |
|------------|---------|--------|-----------------------------------|-----|-----|
|ΒΑΣΗ   Z    |         |        |                                   |     |     |
|ΚΟΡΥΦΗ Z    |   23.18 |  2.33  |   37.76 =  111.29    <  215.53   |   2 | Ναι |
|--ΥΠΟΛΟΓΙΣΜΟΣ ΔΙΣΔΙΑΓΩΝΙΟΥ ΟΠΛΙΣΜΟΥ Asd (30% * Συνολικού Διαμήκους Οπλισμού)--|
|Κατά Y : Asdy =  9.59 cm2 ( 4Φ18) --------- Κατά Z : Asdz =  9.59 cm2 ( 4Φ18) |
|                ΕΛΕΓΧΟΣ ΥΠΕΡΒΑΣΗΣ ΤΗΣ ΜΕΤΑΚΙΝΗΣΗΣ ΑΣΤΟΧΙΑΣ δu                  |
|Σταθερές Υλικού-Διατομής:E(GPa)=28.00 G(GPa)=11.67 Ay(m2)= 0.22 Az(m2)= 0.22  |
|Συντελεστές: γθd=1.3 θψ=0.008 q=3.5                                            |
|       ΣΧΕΤΙΚΗ ΣΕΙΣΜΙΚΗ ΜΕΤΑΚΙΝΗΣΗ            ΜΕΤΑΚΙΝΗΣΗ ΑΣΤΟΧΙΑΣ             |
|---ΘΕΣΗ--|Msd(kNm)-|--Vsd(kN)-|δελ(mm)| δ(mm)|?| αs   | θpe |δu(mm)|Συνδ.|-----|
|ΒΑΣΗ   Y |   -1.00 |  -116.76 |  0.00 |  0.00|<| 1.14|0.009|  0.14|  13 | Ναι |
|ΚΟΡΥΦΗ Y |    4.54 |   -85.41 |  0.00 |  0.01|<| 2.01|0.015|  1.25|   5 | Ναι |
|---------|---------|----------|-------|------|-|-----|-----|------|-----|-----|
|ΒΑΣΗ   Z |   18.84 |    16.39 |  0.03 |  0.13|<| 1.64|0.013| 23.69|   2 | Ναι |
|ΚΟΡΥΦΗ Z |  -27.12 |    16.39 |  0.05 |  0.25|<| 2.36|0.018| 43.28|   2 | Ναι |
|ΥΠΟΛΟΓΙΣΜΟΣ ΔΕΙΚΤΗ ΣΥΜΠΕΡΙΦΟΡΑΣ q'= max{1.5,αs+1}<=q -- (q'y=1.5 -- q'z=1.5)   |
|ΠΡΟΤΕΙΝΟΜΕΝΟΣ ΑΡΜΟΣ (mm) δτy =  10.01   δτz =  10.16                         |
|-------------------------------------------------------------------------------|
```

- First, the y and z shear ratios (local axes) at the top and base of the column are calculated. If they are greater than 2,5, no further check is required. If less than 2.5 and greater than 1.5 are obtained, no further testing is required.
  diagonal reinforcement but either a satisfactory check must be made or the following condition must be satisfied in the results. The existence of rigid castings requires the satisfaction of this condition.
- The calculation of the diagonal reinforcement follows since the shear ratio less than 1.5..
- This is followed by the calculation of the relative seismic displacement δ and the failure displacement δu and the check δ<δu. If the inequality is not met, a reduction of the behaviour index q is required.
- Finally, calculate the seismic coefficient of lift-off behaviour q' by y and by z and the proposed joint δτy if selected in the parameters.

1.3.4.1.4 Critical wall length

The critical wall length is defined as follows:

- The level at which the earthquake has been imposed by the analysis is identified (identify it you must select the "checks" command from the analysis).
- With the checkbox 

unchecked, the program will automatically take the earthquake force level as the basis of the walls. With checked the user manually sets this level. Any change made by
  analysis, if the user has selected the checkbox, the option will remain unaffected.
- With the second checkbox 

unchecked, the program will set the critical length as the length calculated by the regulation, otherwise the user manually selects the critical length (in m).

1.3.4.2 For EUROCODE scenario

1.3.4.2.1 Economic Growth

In the field "Iconic Zoom" check the checkboxes

☑ Προσαύξηση Ροπής Στύλου ☑ Προσαύξηση Ροπής Τοιχείου

☑ Προσαύξηση Τέμνουσας Στύλου ☑ Προσαύξηση Τέμνουσας Τοιχείου

The value of γ_{RD} for overstrength is automatically set as a function of ductility:
$\gamma_{RD=1}$,0 for **DCM / $\gamma_{RD=1}$,2** for **DCH.**

A distinction is made between the poles and the walls. FACTS
The wall moment increment shall always be in accordance with EC8 (**§5**.4.2.4), as shall the wall shear increment.
PENS
The post shear increment (per **§5**.4.2.3) always applies, while the torque increment depends on the static system - required in framed & double equivalent to framed. Specifically, you can see if it is required from the table below from the "checks" in the analysis.

Σεισμική Τέμνουσα Τοιχωμάτων										§ 5.1.2.
Σεισμική Τέμνουσα Τοιχωμάτων							Στάθμη Αναφοράς		0	0.000(m)
α/α Στάθμης	Συνδ/μος	Τέμνουσα Τοιχ./Συνολική Τέμν. = nvx			Ικαν. Έλεγχος	Συνδ/μος	Τέμνουσα Τοιχ./Συνολική Τέμν. = nvz			Ικαν. Έλεγχος
		Τέμνουσα Τοιχωμάτων	Συνολική Τέμνουσα	nvx			Τέμνουσα Τοιχωμάτων	Συνολική Τέμνουσα	nvz	
1 ***	3	0.000	358.834	0.00	ΝΑΙ	35	0.000	365.217	0.00	ΝΑΙ
2	3	0.000	175.323	0.00		35	0.000	170.654	0.00	
ΣΗΜΕΙΩΣΕΙΣ:	*** = Στάθμη ελέγχου πν από κανονισμό									

Καθορισμός Συστήματος Κτιρίου		Προσαύξηση Ροπής (§4.4.2.3)	
Διεύθυνση X:	Σύστημα Πλαισίων	Διεύθυνση X:	ΝΑΙ
Διεύθυνση Z:	Σύστημα Πλαισίων	Διεύθυνση Z:	ΝΑΙ

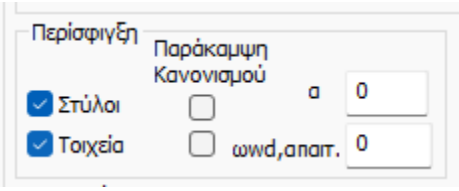
1.3.4.2.2 Critical wall length

The critical wall length is defined as follows:

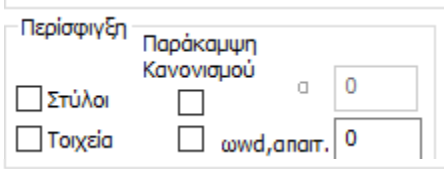
- The level at which the earthquake has been imposed by the analysis is identified (identify it you must select the "checks" command from the analysis).
- With the checkbox ☐ Στάθμη Βάσης Τοιχωμάτων 1 - 300.00 ΟΡ.ΥΠ unchecked, the program will automatically take the earthquake force level as the basis of the walls. With the the user manually sets this level. Any change made by analysis, if the user has selected the checkbox, the option will remain unaffected.
- With the second checkbox ☐ Προκαθαρισμένη Τιμή (m) 3 unchecked, the program will set the critical length as the length calculated by the regulation, otherwise the user manually selects the critical length (in m).

1.3.4.2.3 Circumference

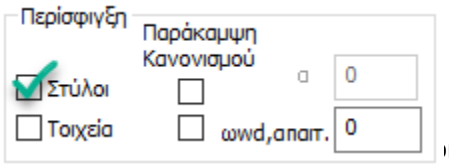
For the control in "Circumference"
In the new version the part of the tightening parameters has been changed in order to have a uniform logic in these parameters in both regulations (ECE-CEEC and EC).
More specifically, a separation of selection of poles, walls for the control in tensioning was made.



New logic is as follows:

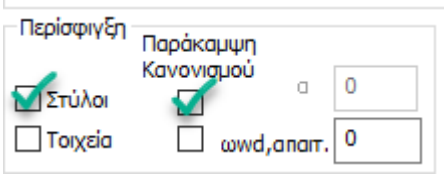


When the option is unchecked on either the poles or the walls, it is not checked in both regulations.



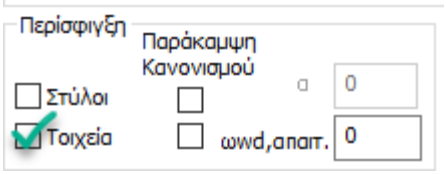
When the selection of poles is checked, for EC8 the check is made only at the level required (for DCM only in the critical area at the foundation foot (ground floor foot)). For DCH it is required and done everywhere at all levels.

ne anywhere because there is nv criterion from analysis and it is up to the user to decide where to do the check.

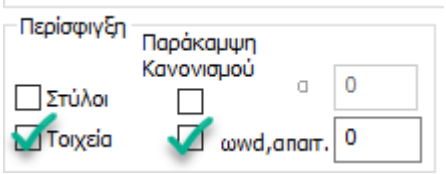


When the poles have the option AND activated "Bypass Regulation" then the check is done wherever the user chooses regardless of the regulation requirement.

Specifically for EAK-EKOS it is the option that must be activated when want to selectively tighten the items I want.

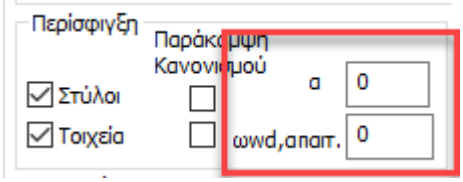


When the selection of walls is checked, then the check is made where required which for both regulations are those walls that are included in the critical length of the building.



When in in files AND the option is activated "Bypass Regulation" then the check is done wherever the user chooses regardless of the regulation requirement.

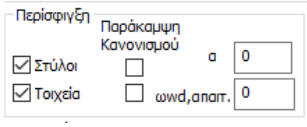
The options for pinned values in a and ωwd,require apply as before and are obviously uniform for poles and walls.



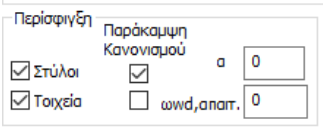
If the side fields of the efficiency factor 'a' and/or the tightening factor 'ωwd,required' remain zero, the program shall take into account the value of the factor 'a' depending on the arrangement of the fasteners and for 'ωwd,required'. the limit of the regulation. Otherwise, the check shall take into account the values you type.

Finally, it is recalled that for the columns the bracing is done their critical areas (at their ends) and for the walls along their entire length (within the critical height of the building).

By default in the parameters you will find:

For EC scenario: 

Containment in Pillars and Vessels where required by regulation

For EAK-EKOS scenario: 

Clamping on Pillars everywhere (user checks the necessity) and Vessels where required by the regulation.

IMPORTANT CLARIFICATIONS ON THE CIRCUMVENTION:

With regard to the calculation of the wall edge restraint fasteners at their critical height, the procedure based on EC8-1 was incorporated in the latest internal version.

The following applies to the initial dimensioning of the wall:

- *It is mandatory to check for tightening at its critical length except in case vd is less than 0.15 (for DCM only) and except in case the result is negative.*
- *The calculation of the required curvature flexibility mf is done with torque design and a corresponding moment of resistance at the foot of the wall (i.e. where is founded, where it ends). The calculation is done for the maximum ratio of all combinations (as required by the Eurocode). Obviously this ratio corresponds to a combination, but this does not mean that, in order to calculate the final Wwdreq, you should take into account in the calculation formula, the vd corresponding to this axial. So in the program we find the mf based on the maximum ratio and then for the calculation of Wwdreq, we take into account this mf and the worst vd (which may possibly result from another combination).*
- *The number of connector segments taken into account by the program in the initial test in shear and tension is always n=2 (worst case) for both addresses. In the recheck (see. B.DETAILS OF REINFORCEMENT*

STYLES_19) it is possible to take into account the actual segments per direction.

- *A key change compared to the calculation of nd in ECOS and EC8-1 is that in EKOS the nd is calculated and relates to the column (axonal reduced to column and area of the column) while in EC8-1 the nd is calculated with the whole axial and the whole cross-section. This also applies to the reduced axial check. It should be noted that with the "whole" axial and area sizes the results are generally more favourable.*

EXAMPLE:
Let's look at an example from which we quote from the Investigation in detail the resulting figures.

```
Κολωνάκι 0 (60.000-40.000) - 200.000
N=753.13 Ac=0.80 Ao=0.18 vd=0.28 bo0=0.55 bo1=0.32 Σbi=0.4087 μφ=4.520 pv=0.002 ωv=0.128
Wwdreq = 0.24 Wwdcalc=0.26
Τελικοί Συνδετήρες Φ8/0.10
```

In the above text, 200 cm is the total dimension of the wall. Then the axial N which is the total, Ac=2x0.4=0.8 m2 (the total cross-section for the calculation of vd), vd which as we said refers to the whole cross-section and the sizes Ao, bo0, bo1, Σbi which refer to the column. The quantities ρv and ωv refer to the mechanical percentage of vertical truss reinforcement.

Wwdreq is the size required by equation 5.20 of EC8, while Wwcalc is the realisable size and corresponds to F8/10 (dimensional) and the corresponding volume of the column based on the following formula

Wwdcalc $= (V_s / V_o) \cdot (f_{yd} / f_{cd})$

As far as the size Vs for the connectors is concerned, the program, always in the initial dimensioning and always for the walls, takes into account the direction perpendicular to the small side of the wall. This is to compensate for the unfavourable effect of the two-part connector, which is taken into account in both directions, whereas in at least one direction a connector with more than two parts is generally obtained. Finally, I emphasize that the segments and Vs only concern Wwdcalc.

About the recheck on restraint within the column details and when the same sizes are calculated. (see B.Pillar fittings_19).

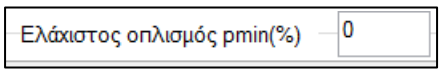
1.3.4.2.4 Twilight

For control in "Bending", activate  the direction in the bend test is to be performed (along the local Y and/or Z axis).

- To display the local axes: "Show">>"Switches">>"Local Axes"

1.3.4.2.5 Cranking

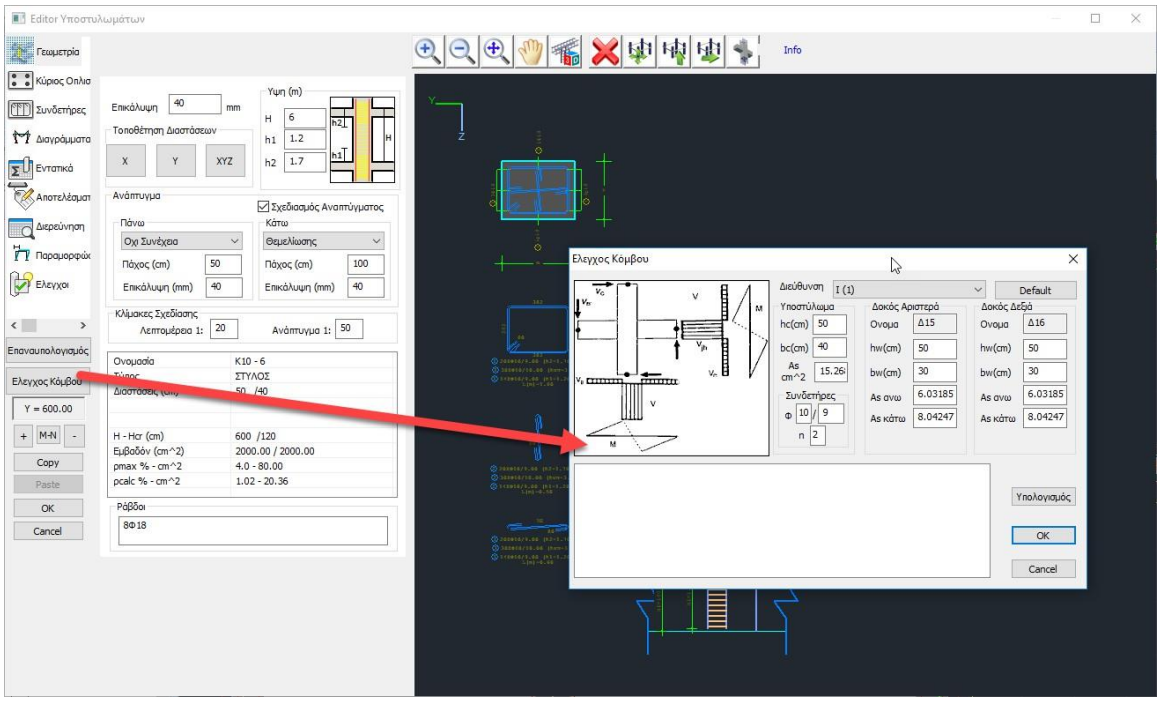
For the control in "Rotate" check the checkbox. The program assumes Vcd = 0 and calculates the clips.



enables the designer to set his own price for the minimum reinforcement percentage.
If a value is given then the program will take it into account as the minimum reinforcement, and if the field is left blank, the program will take the regulation value into account.

1.3.4.2.6 Node Control L/S Control

Activate the checkbox in the "Node Control" to perform the check required in the CIP cases *. §5.5.2.3 & §5.5.2.3 §5.5.3.3 located within the Column Arming Details.



The way to carry out the check is described in detail in the corresponding chapter of the user manual B. PENSION FITTING LEVELS.

5.5.2.3 Κόμβοι δοκών-υποστυλωμάτων

(1)P Η οριζόντια τέμνουσα που δρα στον πυρήνα ενός κόμβου μεταξύ κύριων σεισμικών δοκών και υποστυλωμάτων θα καθορίζεται λαμβάνοντας υπόψη τις δυσμενέστερες συνθήκες σεισμικής δράσης, δηλ. συνθήκες ικανοτικού σχεδιασμού για τις δοκούς που συνδέονται στον κόμβο και συμβατές τιμές των τεμνουσών δυνάμεων στα άλλα συνδεόμενα στοιχεία.

(2) Επιτρέπεται να χρησιμοποιούνται απλουστευμένες εκφράσεις για την οριζόντια τέμνουσα δύναμη που δρα στον πυρήνα σκυροδέματος των κόμβων ως εξής:

α) σε εσωτερικούς κόμβους δοκών-υποστυλωμάτων:

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

β) σε εξωτερικούς κόμβους δοκών-υποστυλωμάτων:

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

όπου

A_{s1} είναι η διατομή του άνω οπλισμού της δοκού

A_{s2} είναι η διατομή του κάτω οπλισμού της δοκού

V_C είναι η τέμνουσα δύναμη του υποστυλώματος πάνω από τον κόμβο, από την ανάλυση σε σεισμική κατάσταση σχεδιασμού

γ_{Rd} είναι συντελεστής υπεραντοχής λόγω σκλήρυνσης από παραμόρφωση του χάλυβα και δεν πρέπει να είναι μικρότερος από 1,2.

(3) Οι τέμνουσες δυνάμεις που δρουν σε κόμβους θα αντιστοιχούν στην δυσμενέστερη φορά της σεισμικής δράσης που επηρεάζει τις τιμές A_{s1}, A_{s2} και V_C που χρησιμοποιούνται στις εκφράσεις (5.22) και (5.23).

5.5.3.3 Beam-subcolumn nodes

α) Σε εσωτερικούς κόμβους δοκών-υποστυλωμάτων πρέπει να ικανοποιείται η ακόλουθη έκφραση:

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

όπου

$\eta = 0{,}6(1\text{-}f_{ck}/250)$;

h_{jc} είναι η απόσταση μεταξύ των ακραίων στρώσεων οπλισμού του υποστυλώματος

b_j είναι όπως ορίζεται στην έκφραση (5.34);

ν_d είναι η ανηγμένη αξονική δύναμη του υπερκείμενου υποστυλώματος, και

f_{ck} δίνεται σε MPa.

β) Σε εξωτερικούς κόμβους δοκών-υποστυλωμάτων:

V_{jhd} πρέπει να είναι μικρότερη από το 80% της τιμής που δίνεται από το δεξιό μέρος της έκφρασης (5.33) όπου:

V_{jhd} δίνεται από τις εκφράσεις (5.22) ή (5.23) αντίστοιχα

και το δρων πλάτος κόμβου b_j είναι:

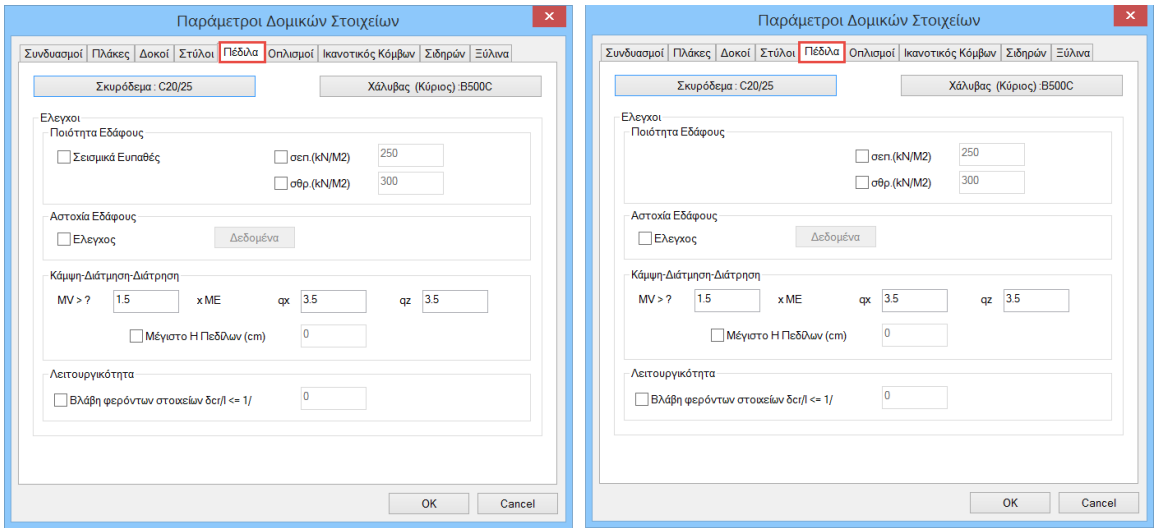
α) εάν $b_c > b_w$: $b_j = \min\{b_c ; (b_w + 0{,}5 \cdot h_c)\}$; (5.34a)

β) εάν $b_c < b_w$: $b_j = \min\{b_w ; (b_c + 0{,}5 \cdot h_c)\}$ (5.34b)

1.3.5 Sandals

For **EKOS-EAK** scenario

For **EUROCODE** scenario



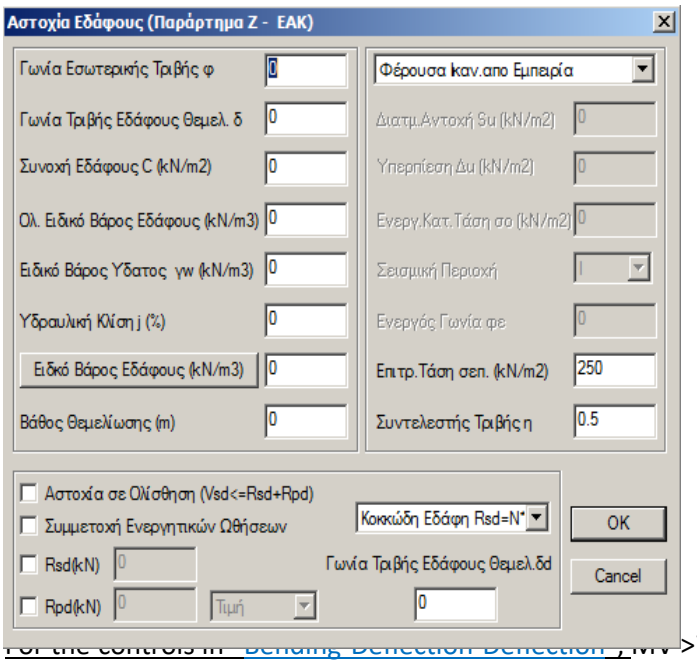
In the **Fields** field select the checks you wish to carry out (check the corresponding checkboxes) and enter the details of a possible soil engineering study.

1.3.5.1 For EKOS-EAK scenario

The following controls are:

The first section deals with soil quality. Check "Soil Quality" where by activating the checkboxes of the trends you can set your own limit values. The option "Seismically Vulnerable" soil applies if you want to take into account the provisions of par. 5.2.3.2. of the E.A.C. 2000 for seismically sensitive soils.
In the "stop" options. and "shtr." enter the values of permissible stress and breaking stress respectively.

"Soil Failure Check" where by activating the checkboxes of the controls you activate the "Data" button and parameters for checking the bearing capacity of the soil. If you have a soil engineering study, enter the soil data.



The upper part of the dialogue box refers to the "Limit Load Failure" check in accordance with Annex G of the ECE 2000.

First, from the list on the top left, select the method of calculating the limit load, depending on the type of foundation soil.

The key

is a key for calculating the buoyant (active) specific gravity of the soil if pressed with the left mouse button (Annex G G.1[2]).

For the controls in "Bending Deflection Deflection", MV >? ***1,5*** * ME

This parameter refers to the NRC.

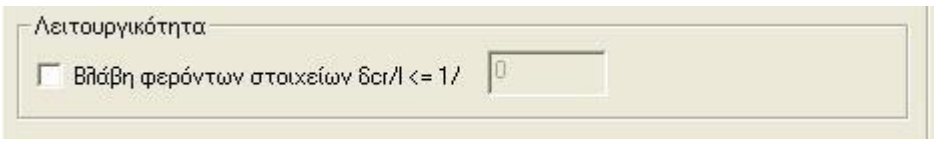
When determining the coefficient $_{acd}$, the N.E.A.C. in paragraph 5.2.2 gives the margin for its calculation according to ex.5.2a in case the moment $_{MV}$ (moment resulting from the non-seismic loads of the combination of ex.4.1) is significant compared to $_{ME}$ (seismic moment at the nearest location of possible plastic joint in the superstructure element bearing on the foundation element under consideration). Since the Regulation does not specify the magnitude of the moments beyond which ex.5.2a will apply, this determination is left to the designer's discretion.

If you do not want to take into account, when calculating $_{\alpha cd}$, the most favourable case of eq.5.2a but eq.5.2, we give a sufficiently large value (e.g. 100) as the MV/ME coefficient.

In the box next to the indication "qx" and "qz" the designer enters the value of the seismic behaviour coefficient of the structure per direction as defined in p.2.6 of E.A.K. 2000. The default value of the program is 3,5.

In the "Maximum Pedestal Height (cm)" option and in the adjacent box type the maximum pedestal height. When dimensioning the program performs a perforation check. If the initial pedestal height is not sufficient, by continuous testing the program reaches a height at which all checks are satisfied. In case a pedestal height greater than the maximum set is required, a warning message is issued to the user.

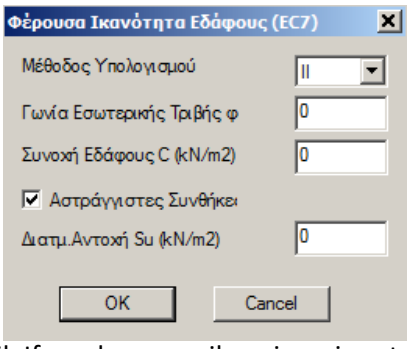
The last section of the pedestals concerns the control of differential settlements.



Check the corresponding option and specify the upper limit of the δcr/l ratio that the program will take into account for the differential settling check.

1.3.5.2 For EUROCODE scenario

The following controls are:
The first section deals with soil quality.



"Soil Quality Control" where by activating the checkboxes of the trends you can set your own limit values. In the options "stop." and "shtr." you enter the values of the allowable stress and fracture stress respectively.

"Soil Failure" check where by activating the checkboxes of the controls you activate the "Data" button
and the parameters for checking the bearing capacity of the soil. If you have a soil engineering study, enter the soil data.

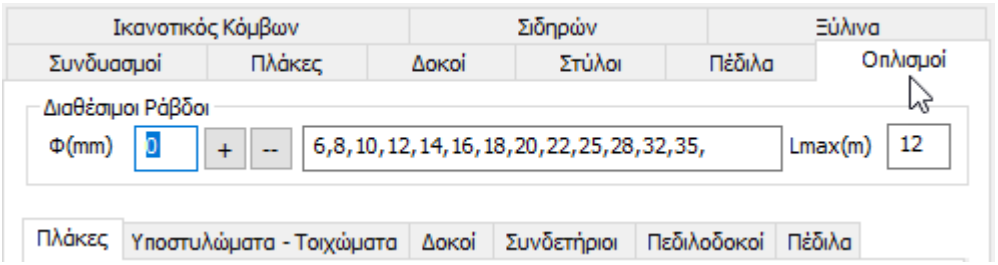
For the checks in "Bending-Shear-Deflection-Deflection", check the checkbox and enter the limit for the maximum height of the pedestal. During the dimensioning process the program carries out the check without shear. If the initial height of the pedestal does not satisfy the check then the program continues testing until it reaches the height that satisfies the check. If this is greater than the set limit a permanent message appears informing that a finer perimeter is required.

At Fields 

type the prices of coefficients that you used in the analysis.

For the "Functionality" checksactivating the checkbox allows you to enter the limit of the δcr/l ratio to be taken into account in the differential settling check.

1.3.6 Armaments



In the first field, "common" for all structural elements, you specify which bar diameters are to be used. From the list 

of diameters you add or remove the one you enter in 

with the help of the keys 

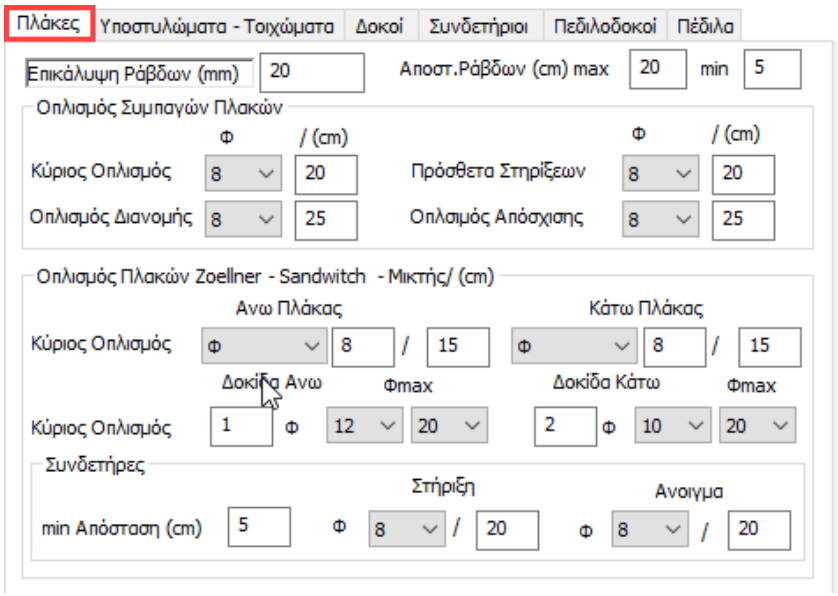
and 

respectively.

In the **Lmax** (cm) field enter the maximum length of the rod to be used in the reinforcement of the structural elements.

1.3.6.1 Armaments - Plates

Enter the overlap in mm according to the climatic conditions.



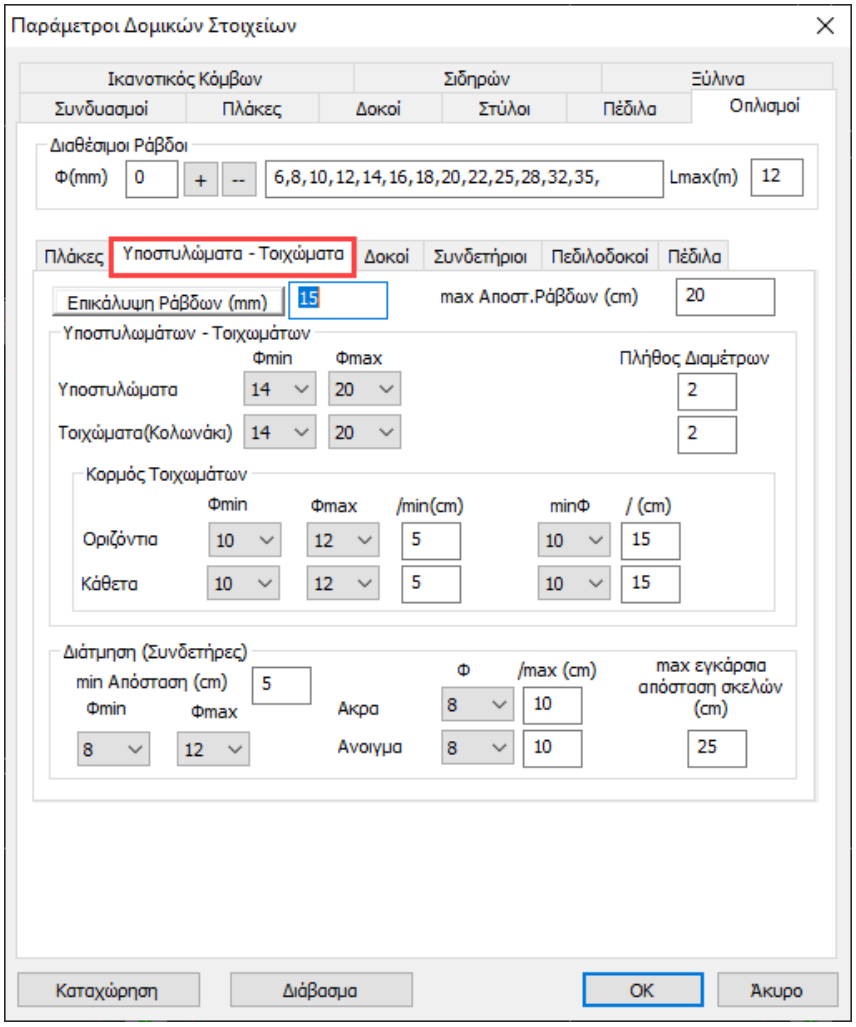
- In the field Αποστ.Ράβδων : type in cm the maximum and minimum desired distance of the bars inside the plate.
- In the field "Solid plate armouring": define the main armouring, the additional armouring and the
  the separation reinforcement and the distances between them.
- In the field "Zoellner-Sandwitch-Mixed Plate Armouring": you define the main armouring above and below. Similarly for the beams you define the number of bars and the min and max diameter top and bottom.
- In the "Connectors" field: set the minimum distance of the connectors in cm and the diameter/distance to support and opening.

1.3.6.2 Reinforcements - Supports-Walls

Enter the overlap in mm according to the climatic conditions.

In the max field Αποστ.Ράβδων : enter in cm the maximum desired distance of the bars inside the poles.

In the "Columns-Walls" field : define the maximum and minimum diameter of the main reinforcement in bending to be used in the columns and wall columns.



With the "Number of Configurations" option you specify the number of successive different diameters to be used for the reinforcement of the columns and walls respectively.

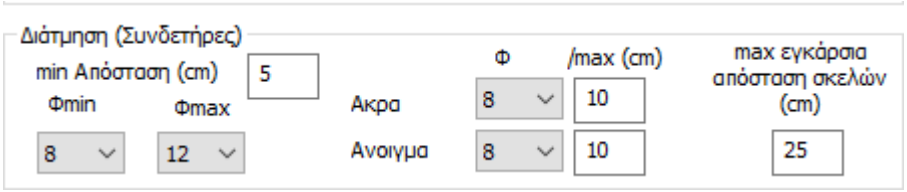
- For example, if you set number of diameters 2, the program will use two more consecutive diameters for the reinforcement of the pole,

i.e. a total of 3 (e.g. F16-F18-F20). If you enter the value 0 the program will use only one diameter.

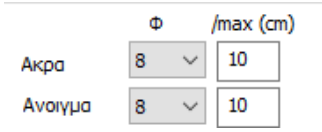
The parameters relating to the reinforcement of the wall frame are then specified.

For horizontal and vertical bars respectively, you specify the minimum and maximum diameter to be used, as well as the minimum distance below which the diameter will be increased.
While minF/cm is a limitation at the square centimetre level on the surface of the trunk. By default you set the ratio to F10/15 horizontally and vertically.

In the "**Shear**" field

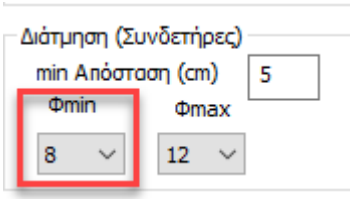


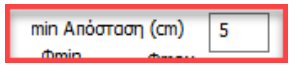
First we start by defining a LOWER BORDER of reinforcement:



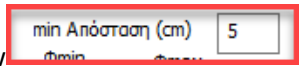
is the minimum for the connector reinforcement desired by the designer. If this reinforcement is sufficient, it shall be installed.

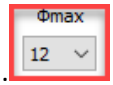
If the minimum reinforcement we have defined is now not sufficient, then the program searches for the reinforcement required based on intensity and starts with Φmin:



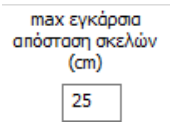
and with a distance greater than 

.

If the need for a shorter distance below 

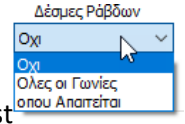
arises, the diameter is changed up to . 

If eventually and with Φmax a distance shorter than the min distance is required, the program will implement it but a failure with symbol "S" will be displayed.

With the option 

you enter the maximum desired distance between the legs of the fastener above which the program automatically inserts an additional fastener. Thus the designer can increase or decrease this limit according to the requirements of his or her institution. Please note that the programme will also take into account the limits of the Regulation.
5.4.3.2:

β) Η απόσταση μεταξύ διαδοχικών διαμηκων ράβδων συγκρατούμενων με συνδετήρες δεν υπερβαίνει τα 200 mm, λαμβάνοντας υπόψη το EN 1992-1-1:2004, **9.5.3(6)**.

In the scenarios of the EKOS-EAK: With the "Bonds of Bars" list 

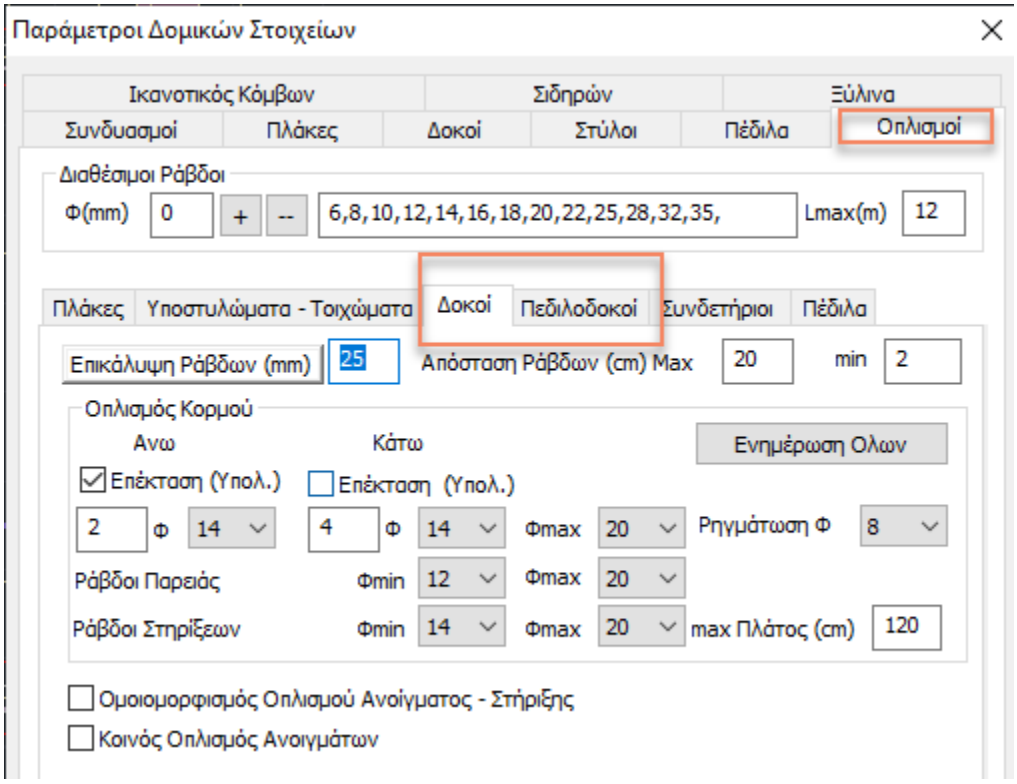
select whether or not to place bundles of bars at the tops of the columns. By selecting the corresponding option you determine whether to place beam bundles at all corners or only where required or not at all

1.3.6.3 Armaments - Beams & Brackets

For the "**Beams**" and "**Connectors**":
Enter the overlap in mm according to the climatic conditions.

In the field Απόσταση Ράβδων enter in cm the maximum and minimum desired spacing of the bars inside the beams.

Common for scenarios of EKOS-EAK and EUROCODE



In the "**Body Armament**" field:

The options "**Extend Upper**" and "**Lower**" mean that the reinforcement of the supports will also pass through the opening and will be included in it.
Note: In the new version the option of extension is only of a calculative nature, as in any case when there is continuity, the reinforcement will pass to the next opening for anchoring purposes.

Then you select the minimum number and the maximum and minimum diameter of the bars of the main reinforcement of the span, supports and stringers.

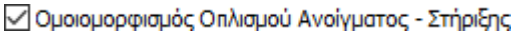
First you determine the minimum main reinforcement of the upper opening. You enter the number and select the diameter of the bar and do the same for the reinforcement of the lower opening.

You then determine the maximum diameter of the bars.

Next, you select the minimum and maximum diameter of the main reinforcement bars to be used in the supports and struts.

Finally, you specify the minimum diameter to be used for the crack reinforcement.

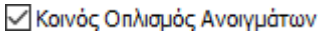
In the max Width option, you specify the maximum distance above which two support rods are placed, and below which one common rod is placed.

Activate the checkbox 

for the program to homogenize the reinforcement in the opening and supports. This means that it puts the same reinforcement in the span and the supports and is usually used in beams with short lengths.

- **IMPORTANT NOTE:**

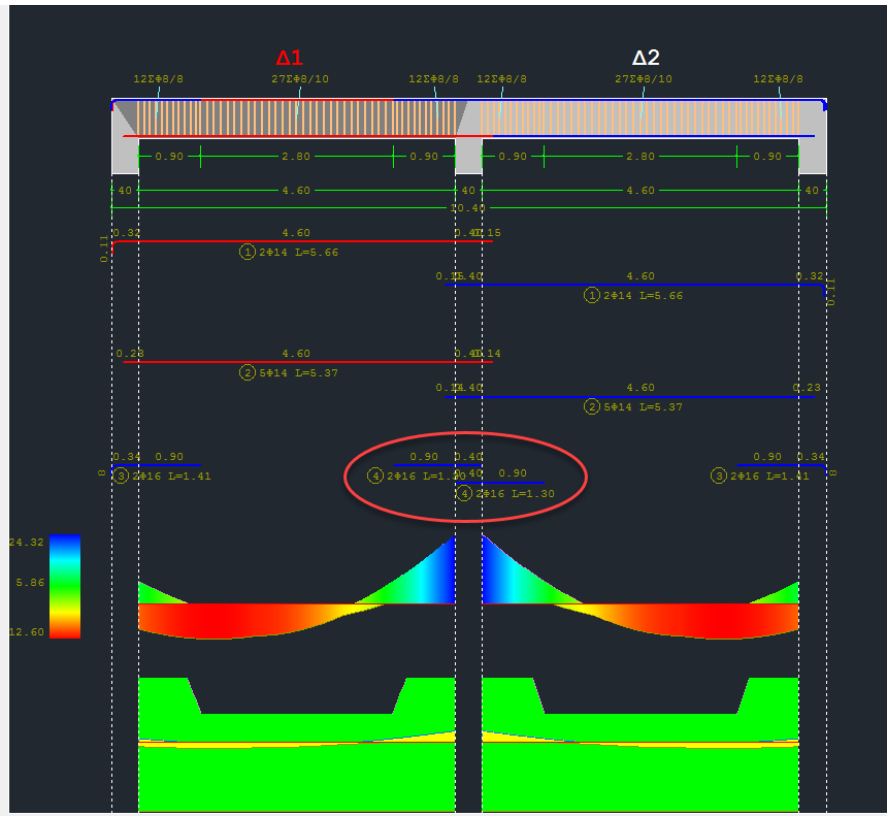
COMMON APERTURE ARMAMENT:

Activate the checkbox 

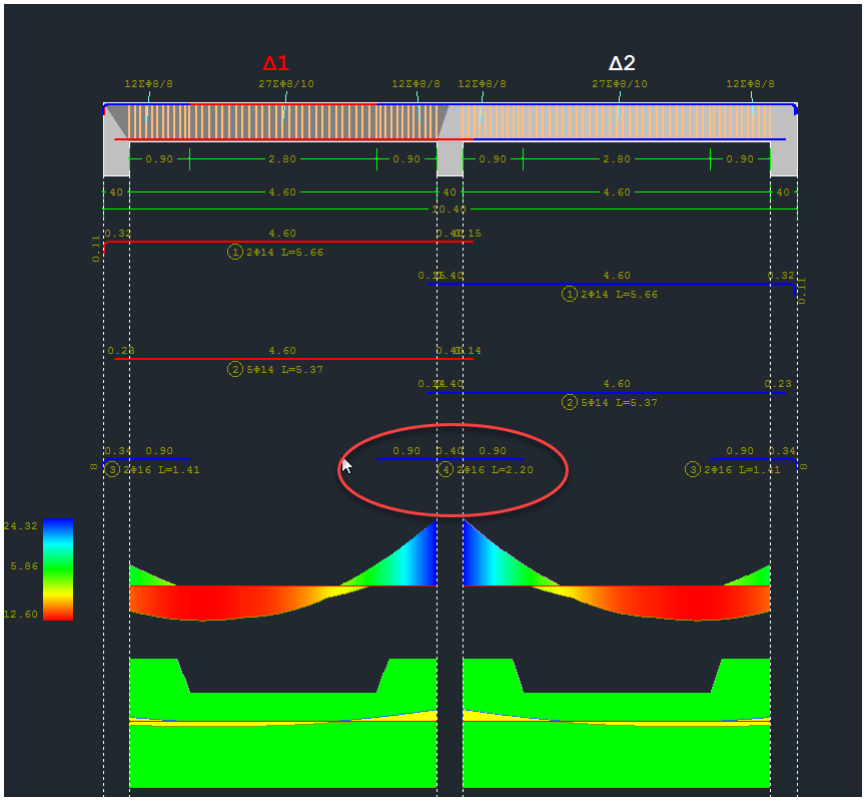
so that the reinforcement in the spans is common for the whole beam.

In the course of the implementation of the single reinforcement in the beams with common reinforcement of a certain length, the program places the additional bars of the supports according to criteria. There are two ways of placing the additional reinforcement of supports.

- The first way is that the additional bars come from each opening on either side and are placed on the corresponding side of the opening.
- The second way is to place a common support iron.

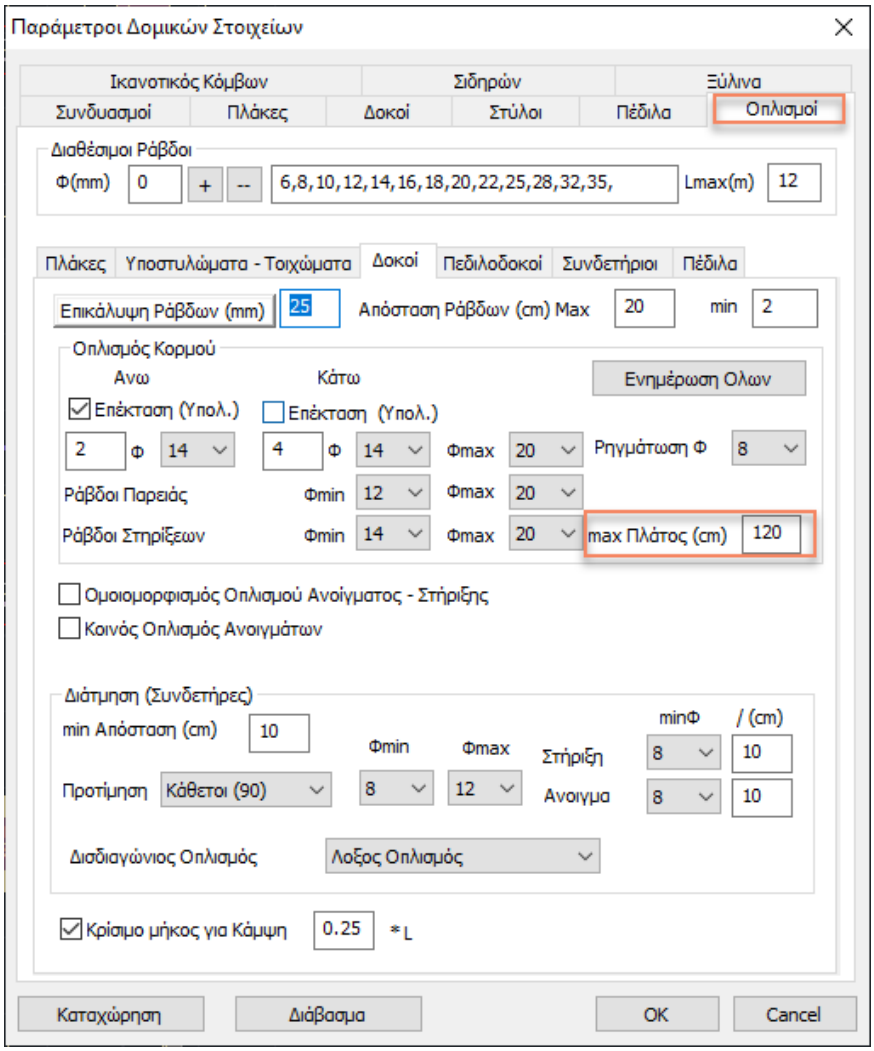


picture 1



picture 2

A. The first criterion is the <u>width of the support</u>, as defined by the reinforcement parameters of the beams.



If this exceeds the value of the max support width (see figure 1), then separate support bars are installed per side.
If the width of the support is less than the max width, then a common iron is placed for the whole support (see figure 2).

- **NOTE:**

The change of this parameter after the creation of the beam spans requires deleting and recreating them.

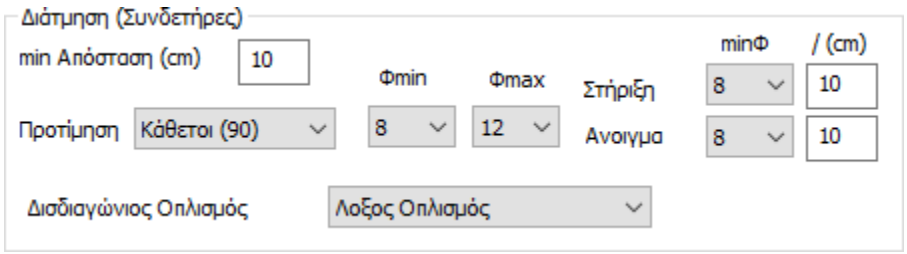
B. The second criterion has to do with the width of the beams in the support. If this width is different for the two beams, then separate additional support bars are installed. If not, a common bar shall be fitted

- **CONCLUSIONS:**

A joint is only placed if both of the above criteria are met:

- Support width< max width in parameters
- Same beam width

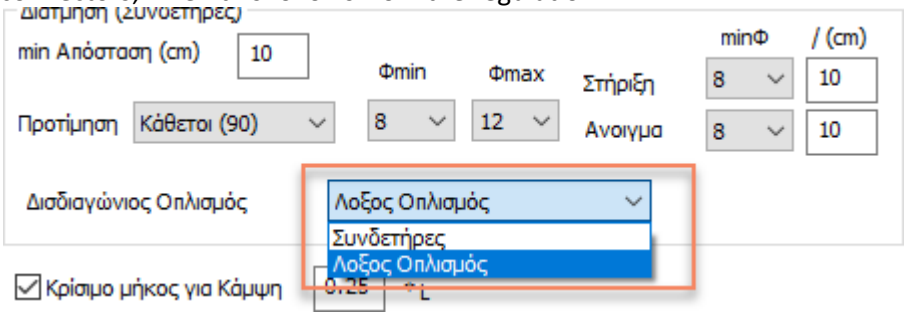
The next section deals with the selection of parameters for the **shear reinforcement of the beams**.

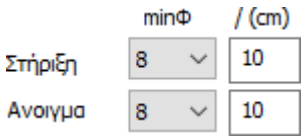


First you define the minimum distance in centimetres. Then you determine whether you want the intersection to be received from vertically positioned fasteners, or from obliquely positioned fasteners (45$^{o)}$. You then specify the minimum and maximum diameter of the fasteners to be used.

NEO!

Added the user option to receive the shear either with oblique reinforcement or with biaxial connectors, when this follows from the regulation.



Finally, in the options 

you specify in the support and in the opening respectively the diameter and the distance you wish. If this reinforcement is sufficient, it is installed.

If now the minimum reinforcement we have defined is not sufficient, then the program searches for the reinforcement required based on intensity and starts with Φmin:

and with a distance greater than 

.

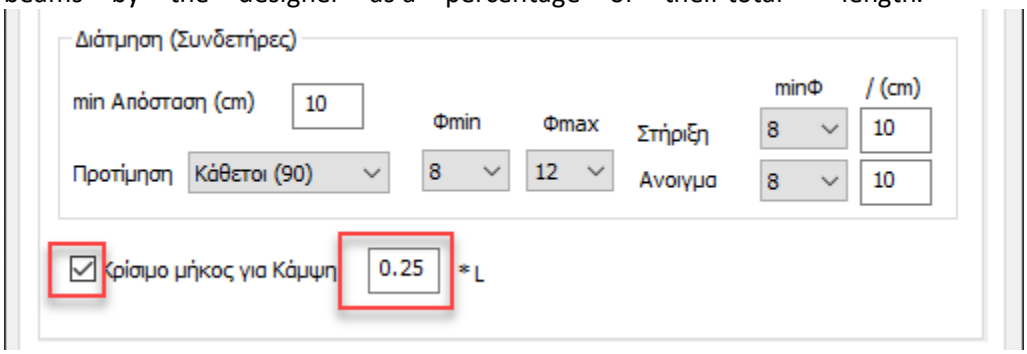
If the need arises for a shorter distance under the 

the diameter is changed up to . 

If eventually and with Φmax a distance shorter than the min distance is required, the program will implement it but a failure with symbol "S" will appear.

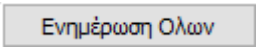
NEO!

The new version of the program added the possibility to define the critical bending length of beams by the designer as a percentage of their total length.



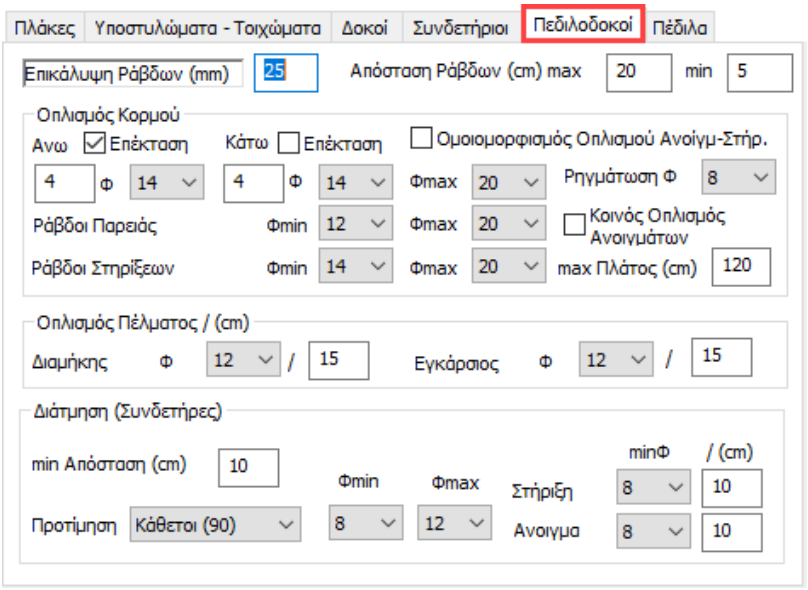
The default value is 0.25 of the length. To be taken into account the parameter must be checked. If it is not checked, the program will take into account as the critical length the one received so far, i.e. the shear length.

- **OBSERVATION:**

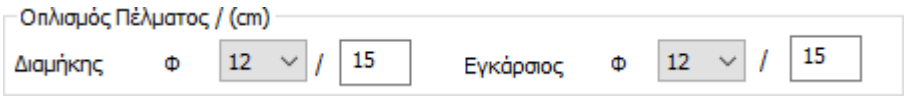


: after completing the parameter selections and before closing the window, press the Update All button to update the parameters.

1.3.6.4 Pedestals

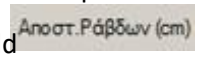


For the frame reinforcement as well as for the shear reinforcement the same applies as for the beams and the connecting beams. The additional element for foot beams is the definition of the minimum longitudinal and transverse reinforcement.

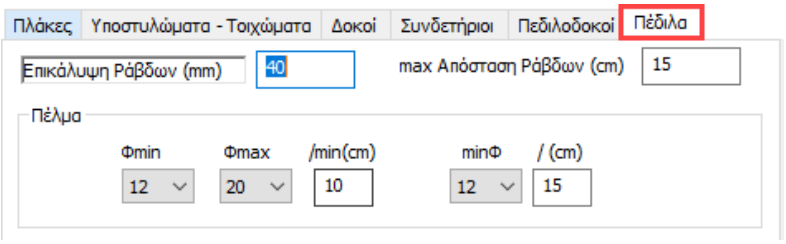


1.3.6.5 Armaments - Sandals

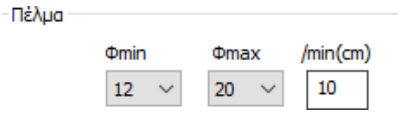
Enter the overlap in mm according to the climatic conditions.

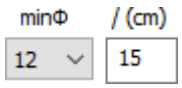
In the field 

enter in cm the maximum and minimum desired distance of the bars inside the pedestals.



Finally, for the tread of the pedestals you specify the minimum and maximum diameter of the reinforcement and the minimum distance of the bars below which the program changes

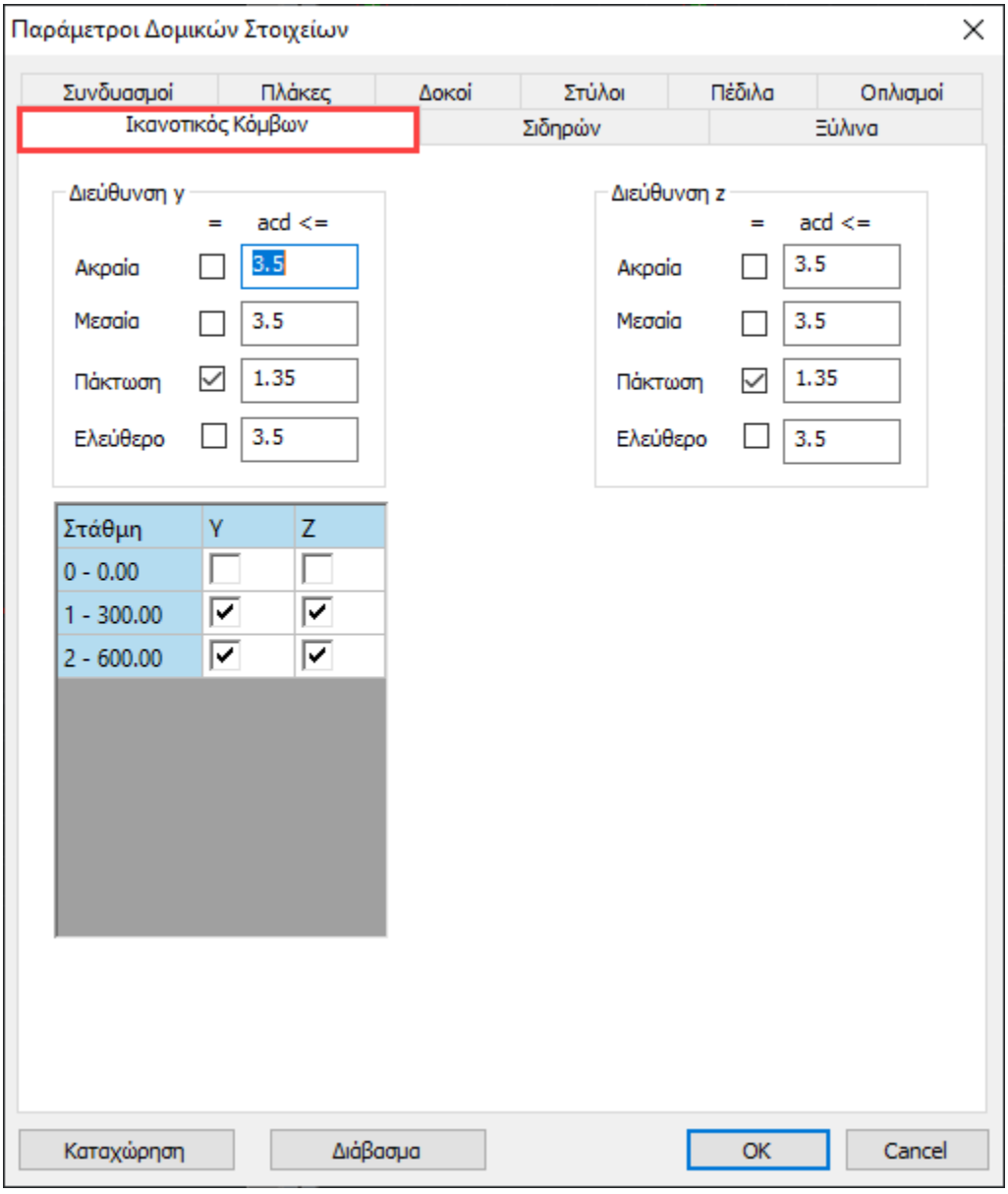


diameter . In the option 

you specify the diameter and the distance of the reinforcement you wish to place. If this reinforcement is sufficient, it is placed. Otherwise, the reinforcement resulting from the program check is placed.

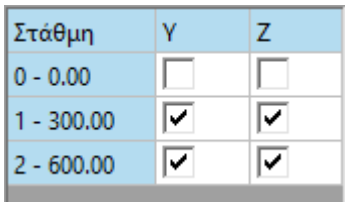
1.3.7 Iconic Node

In the last section "**Icannotic Node**"



specify by x and z the parameters to be used in the satisfaction check.

At the bottom



Στάθμη	Y	Z
0 - 0.00	☐	☐
1 - 300.00	☑	☑
2 - 600.00	☑	☑

you select the level or levels and the direction where you wish to perform the level check.

Specify the upper bound on the satisfactory node enlargement factor $_{acd}$.

In general, the value of $_{acd}$ is defined to be less than or equal to the value of the seismic behaviour coefficient q.
For the footing positions of the columns, $_{acd}$ shall be taken as 1,35.
Check the corresponding option and enter the value you want.
If you do not check any option, the program will take into account the value of acd it will calculate.

OBSERVATION:
The definition of the node type will then be done with the "Node Characterization" option.
Not "Node Characterization" by the user means that all nodes are taken as free in both directions, except for the packed nodes.

Finally, in the investigation of the pole

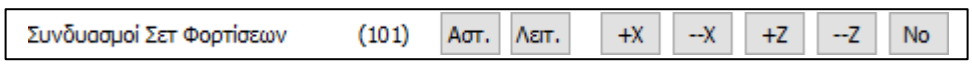
Προσαύξηση	Ροπής (Ικανοτικός)			
Συνδ.	My	Mz	My	Mz
ΤΕΛΟΣ 3	33.604	-40.115	33.604	-60.463
ΤΕΛΟΣ 4	33.604	-40.115	33.604	-60.463
ΤΕΛΟΣ 5	25.082	-39.773	25.082	-59.266
ΤΕΛΟΣ 6	25.082	-39.773	25.082	-59.266
ΤΕΛΟΣ 7	35.577	-5.907	35.577	59.266
ΤΕΛΟΣ 8	35.577	-5.907	35.577	59.266
ΤΕΛΟΣ 9	27.054	-5.565	27.054	60.463
ΤΕΛΟΣ 10	27.054	-5.565	27.054	60.463
ΤΕΛΟΣ 11	34.196	-39.773	34.196	-59.266
ΤΕΛΟΣ 12	34.196	-39.773	34.196	-59.266
ΤΕΛΟΣ 13	24.490	-40.115	24.490	-60.463
ΤΕΛΟΣ 14	24.490	-40.115	24.490	-60.463
ΤΕΛΟΣ 15	36.168	-5.565	36.168	60.463
ΤΕΛΟΣ 16	36.168	-5.565	36.168	60.463
ΤΕΛΟΣ 17	26.462	-5.907	26.462	59.266
ΤΕΛΟΣ 18	26.462	-5.907	26.462	59.266
ΤΕΛΟΣ 19	35.577	-38.975	35.577	-56.472
ΤΕΛΟΣ 20	35.577	-38.975	35.577	-56.472
ΤΕΛΟΣ 21	27.054	-38.633	27.054	-55.274
ΤΕΛΟΣ 22	27.054	-38.633	27.054	-55.274
ΤΕΛΟΣ 23	33.604	-7.047	33.604	55.274
ΤΕΛΟΣ 24	33.604	-7.047	33.604	55.274
ΤΕΛΟΣ 25	25.082	-6.705	25.082	56.472
ΤΕΛΟΣ 26	25.082	-6.705	25.082	56.472
ΤΕΛΟΣ 27	36.168	-38.633	36.168	-55.274
ΤΕΛΟΣ 28	36.168	-38.633	36.168	-55.274
ΤΕΛΟΣ 29	26.462	-38.975	26.462	-56.472
ΤΕΛΟΣ 30	26.462	-38.975	26.462	-56.472
ΤΕΛΟΣ 31	34.196	-6.705	34.196	56.472
ΤΕΛΟΣ 32	34.196	-6.705	34.196	56.472

there are, for each combination, the results of the satisfactory (initial and incremental moments). Of course, for the direction not taken into account by the satisficer, the initial and incremental moments are the same.

1.3.7.1 Other ways of exempting an address from the control of the addressee

In the "Combinations" field the list of all combinations is displayed.
The first number is the number of the combination and second, in brackets, is the number of the equation from which we obtain
The "L/A" column indicates the marginal state of the combination and the "Against" column indicates which direction is involved for the satisfactory control.

Thanks to the bar:



you can modify both the limit state and the direction.

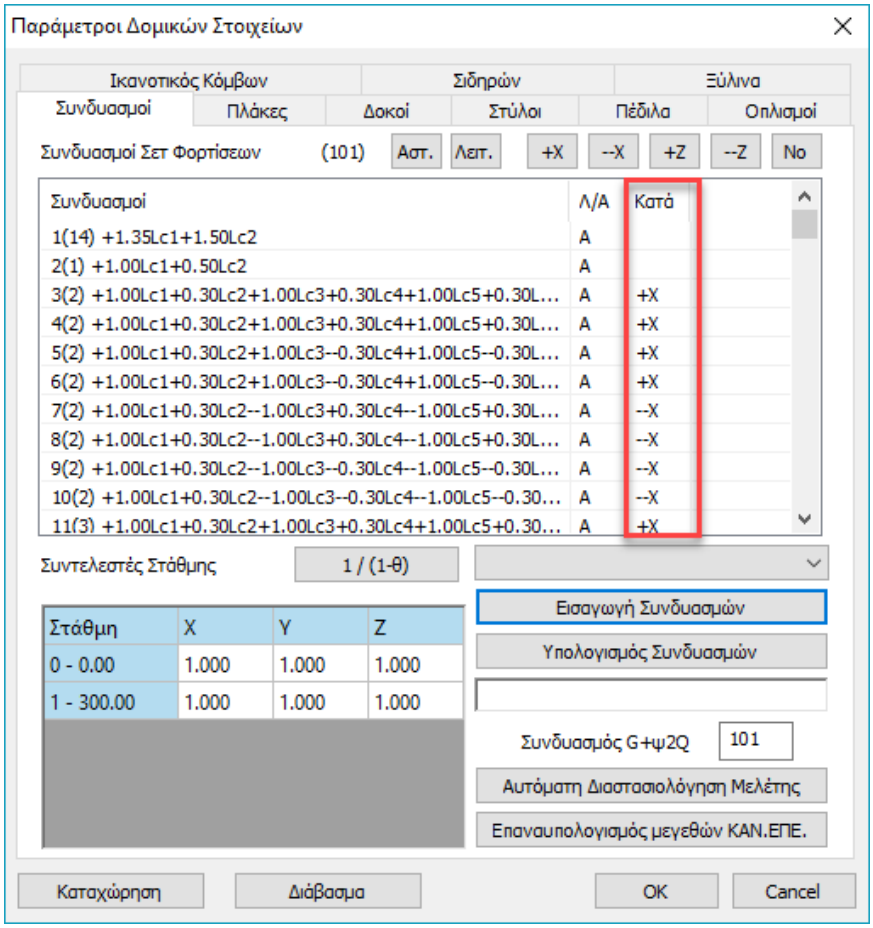
In the column "L/A" you specify whether the specific combination is Failure or Functionality. If you want to make a change, you select the combination and press the key Λειτ. ή Αστ..

The next column "Against" is about the satisfaction check and in which direction the particular combination is involved. By selecting the corresponding button +X, −X, +Z, −Z you select the direction in which the particular combination will participate performing the satisfaction check.

Finally, by selecting the button No button to specify that this combination will not participate in the satisfaction check.

The regulation mentions the capacity check and the necessity to perform it per direction of the earthquake and not per direction of the column.

The exclusion of a direction from the control of the capacity for one or more columns is implemented in SCADA by the zeroing of the incremental factor acd for those seismic combinations where the seismic force, for the specific direction, participates with unity. This is the meaning of the characterization of the combinations that appears in the sizing parameters.



a combination is marked by x or by z when the corresponding seismic force has a factor of one.

So in conclusion, we would say that when I want to exclude one direction from a pole so that it does not become satisfactory, I go to the characterization and choose the direction of the local axis that is parallel to the direction of the earthquake I want to exclude.

Now if the pole is twisted or the pseudo axes are twisted as in the EAK, I choose the local one with the smaller angle than the corresponding seismic one I want exclude. In this way the program will calculate acd for me only for the specific seismic direction (but obviously for both local directions of the pole) and will not calculate any acd for the seismic combinations of the other direction.

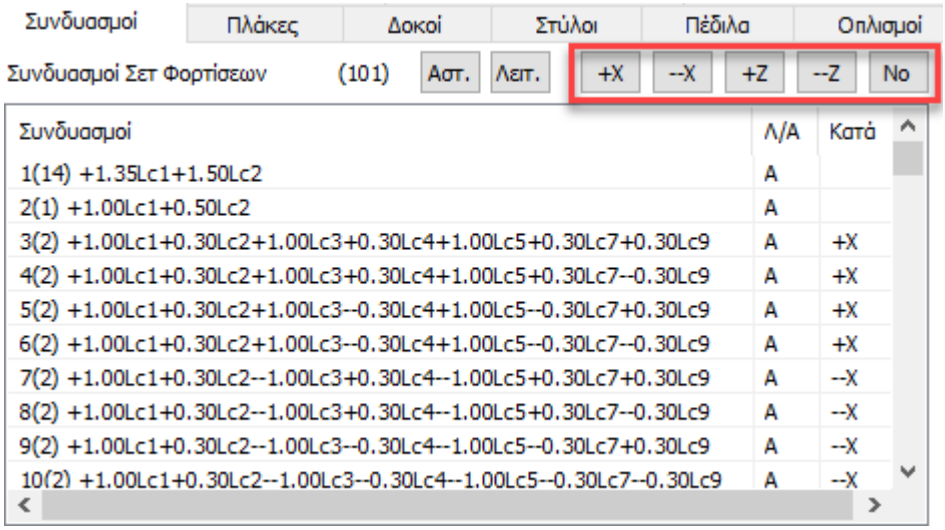
Indicatively in the following printout:

Κόμβος = 8
Στύλος Κάτω = 4

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

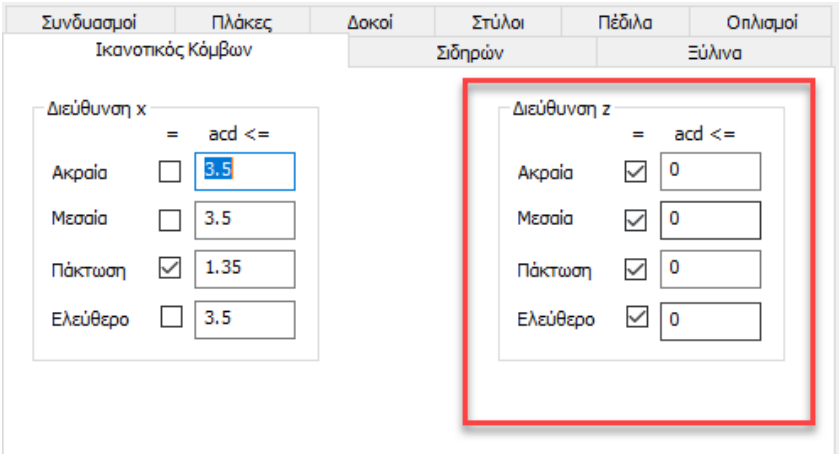
you can see that acd has been calculated for combinations up to 34 (it is the combinations +x and -x) while from 35 onwards no acd has been calculated at all (it is the combinations +z and -z)

Still, another way avoid content in one direction is tweak the combination designations in the dimensioning with the tools above.



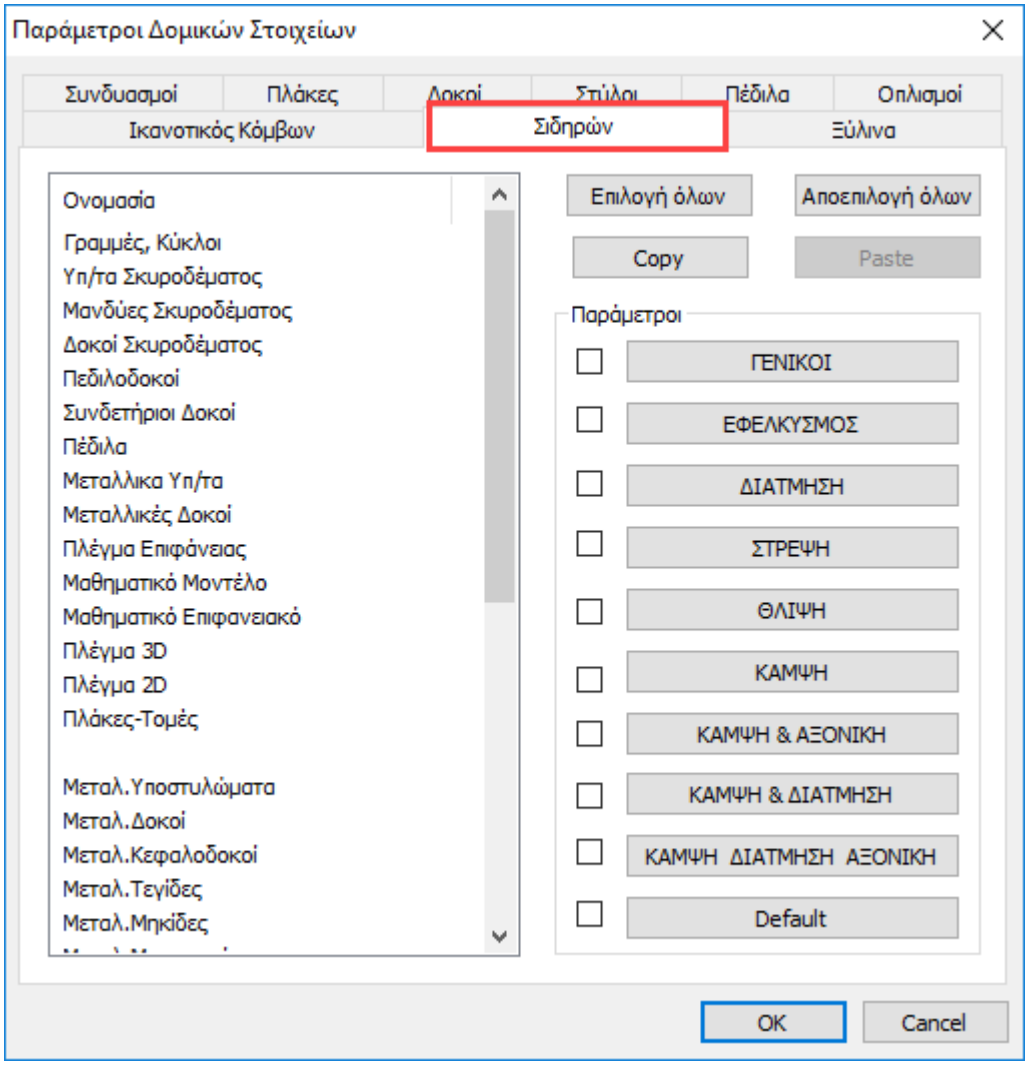
You can select one or more combinations, depending on the seismic direction, and mark it with "No". In this way they will not be taken into account at all in the satisfactory.

Finally, a final way to avoid checking on one of two addresses is to set the acd =0 limit in the general parameters.
You will achieve the same result again.



1.3.8 Iron

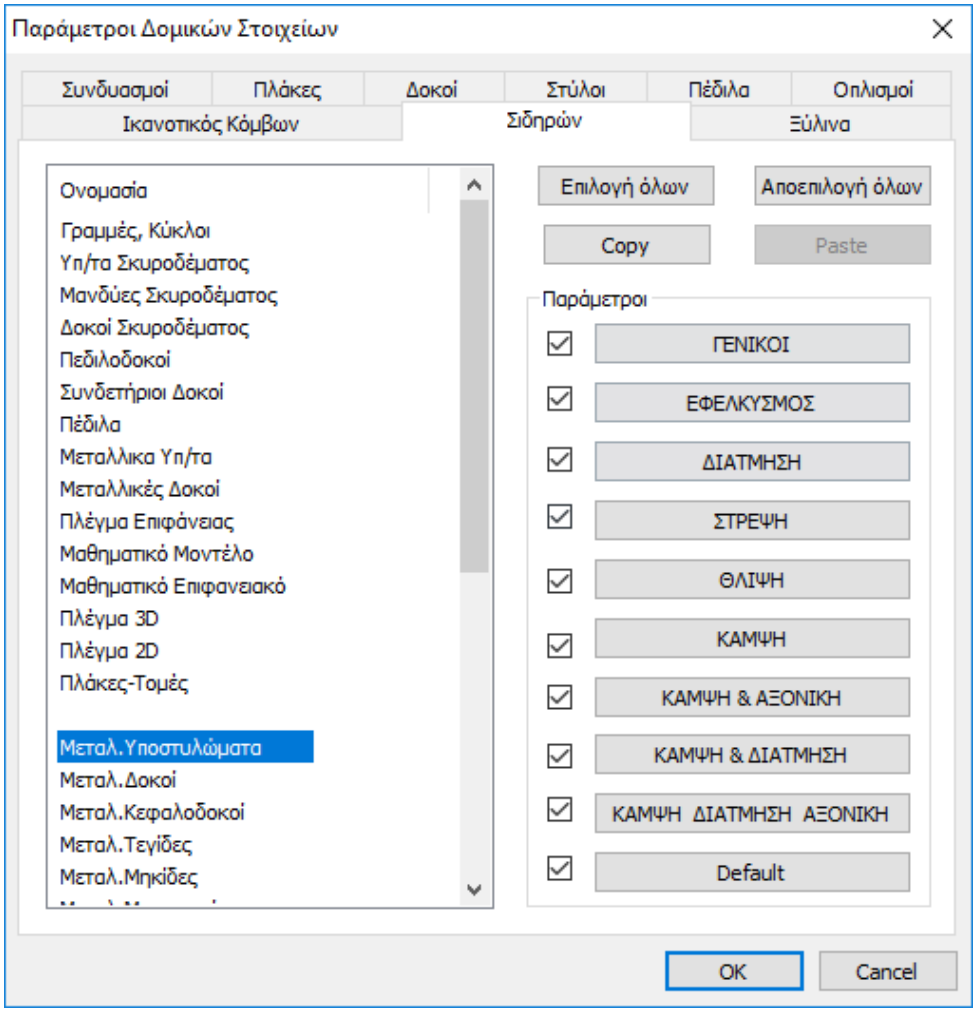
The next section deals with the parameters of the control of the cross-sections of **steel structures**. Selecting the section displays the following dialog box



For metal structures, to set the parameters related to the sizing of metal elements, select the "Iron" field. The box that appears is divided into two parts: on the left there is a list of all layers and on the right a list of controls, each containing the corresponding parameters of that control. First you select one layer, or more with the help of "ctrl" , or all of them with the "Select all" button. Then you activate the checkbox of a control and select the corresponding key to enter the parameters.

The "Deselect all" button cancels the previous selection of layers.

Once you have set the parameters of one layer you can copy them to other layers using the "Copy" command. Select a layer and "Copy", then select another layer and "Paste" and the parameters of the first layer are copied to the second one.



The definition of the sizing parameters of the metallic sections is done layer by layer. You select the layer for which you want to define the parameters (e.g. Metallic Sub/Underground) and per control category (General, Tensile, Shear, etc.), you define the corresponding parameters. Once you have set the parameters for a layer, the program allows you to copy these parameters to another layer using the Copy And Paste logic.

EXAMPLE:

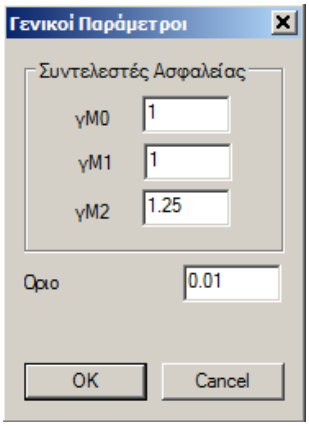
Suppose you have set all the parameters for the Metallic Bases layer and you want to pass these parameters to the Metallic Beams layer. Select the check box next "Default" and all the parameter categories are automatically selected.
Then select the "Copy" button and select the Metal Beams layer and press the "Paste" button that has already been activated. Now all the parameters of the Metallic Lumber layer have been passed to the Metallic Beams layer as well.

An alternative method to set the same parameters for all layers that include metallic cross-sections is to select all layers with the "Select all" button and set the parameters for each control category once.
It should also be noted that to set parameters at least one (or more) layer must be selected.

The parameters for each control category are explained in detail below. By selecting

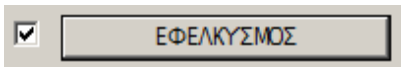
the "GENERAL" section, the following dialog box appears:

to set the GM safety factors:

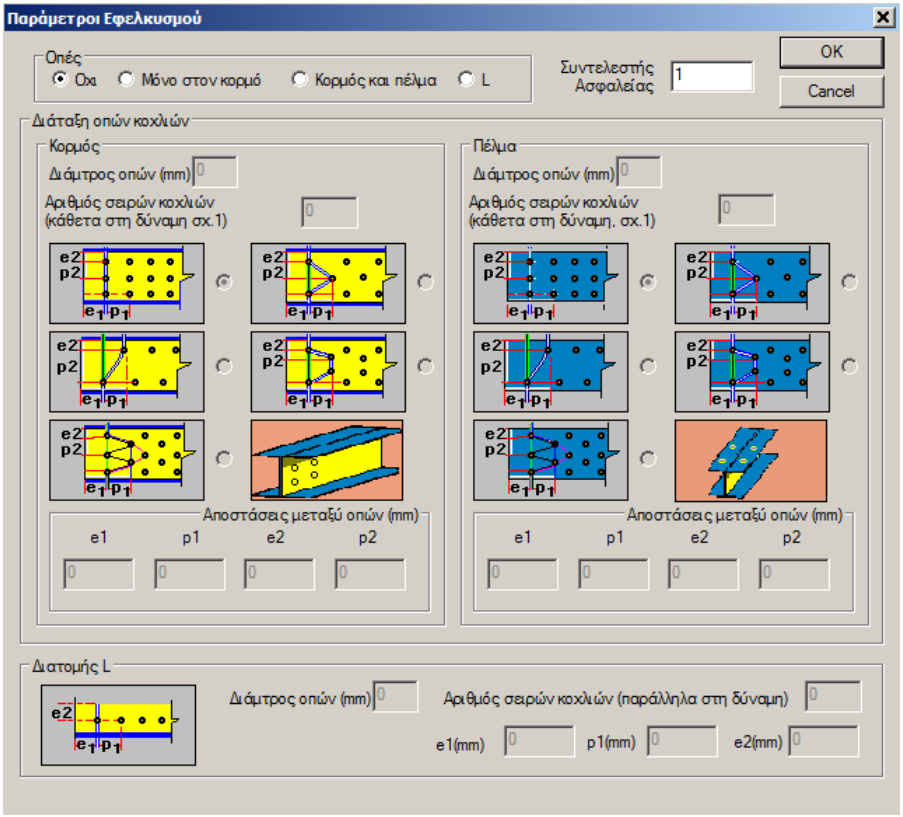


γM0= transverse stress resistance for each category of members γM1 = resistance to buckling based on tests
cM2= tensile fracture strength of cross sections

Here you can set the individual safety factors and a minimum threshold for the intensive sizes below which the intensive sizes are not taken into account. The above values are those proposed by the Eurocode.

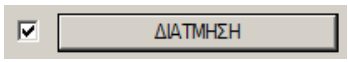


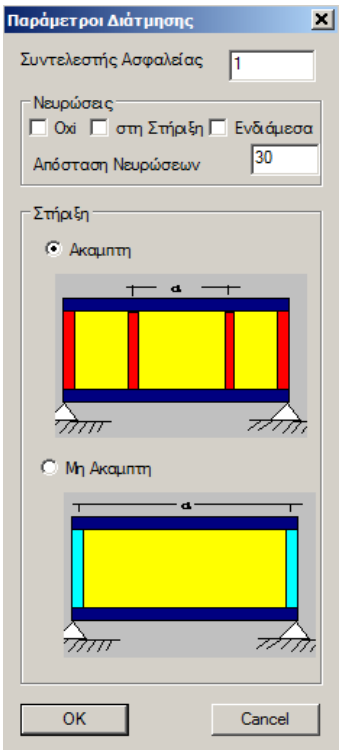
To set the "Tensile" parameters and check the position of the holes (EC3 chapter 1.8 §3.5):



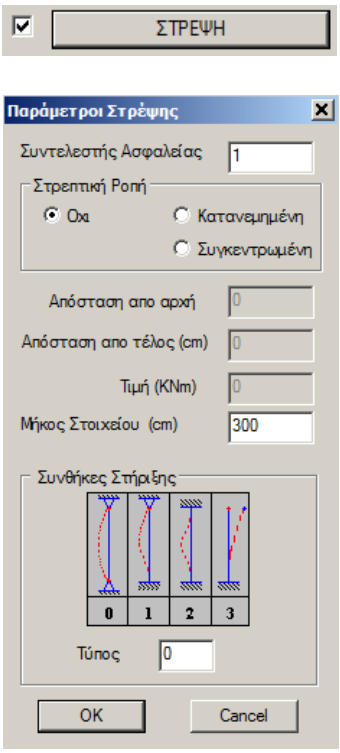
For the holes define the distances from the ends, the diameter and the number of rows on the torso and tread.
In the case of an L section, set the parameters at the bottom of the box.

The rationale here is to specify whether the application during the tensile test take into account the bolt holes of the connections in order to account for reduced tensile strength of the cross-section. If you decide to provide data you will derive it, for the specific layer (e.g. Metallic Supports) from the corresponding connection checks you should have already performed. You must therefore have previously checked the connections before you can give data here.
The safety factor for all checks is predefined and equal to unity, which means that the program calculates the ratio of the corresponding stress to strength and if this ratio is greater than unity, a failure occurs.



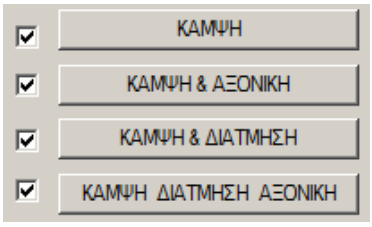


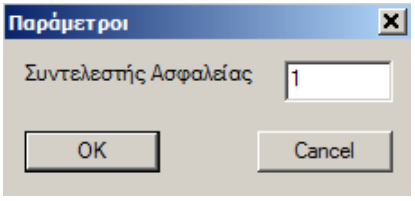
Here you define whether the elements of this Layer have ribs or not and if they do, where they are present (on the support and/or the trunk). You also define the spacing of the ribs as well as whether the connection of an element is rigid or not.



Here you specify whether the members of the layer are loaded by torsional moment (distributed or concentrated). If they are loaded, you define the elements of the loading. You also specify the support conditions of the members based on the support type shown in the graph.

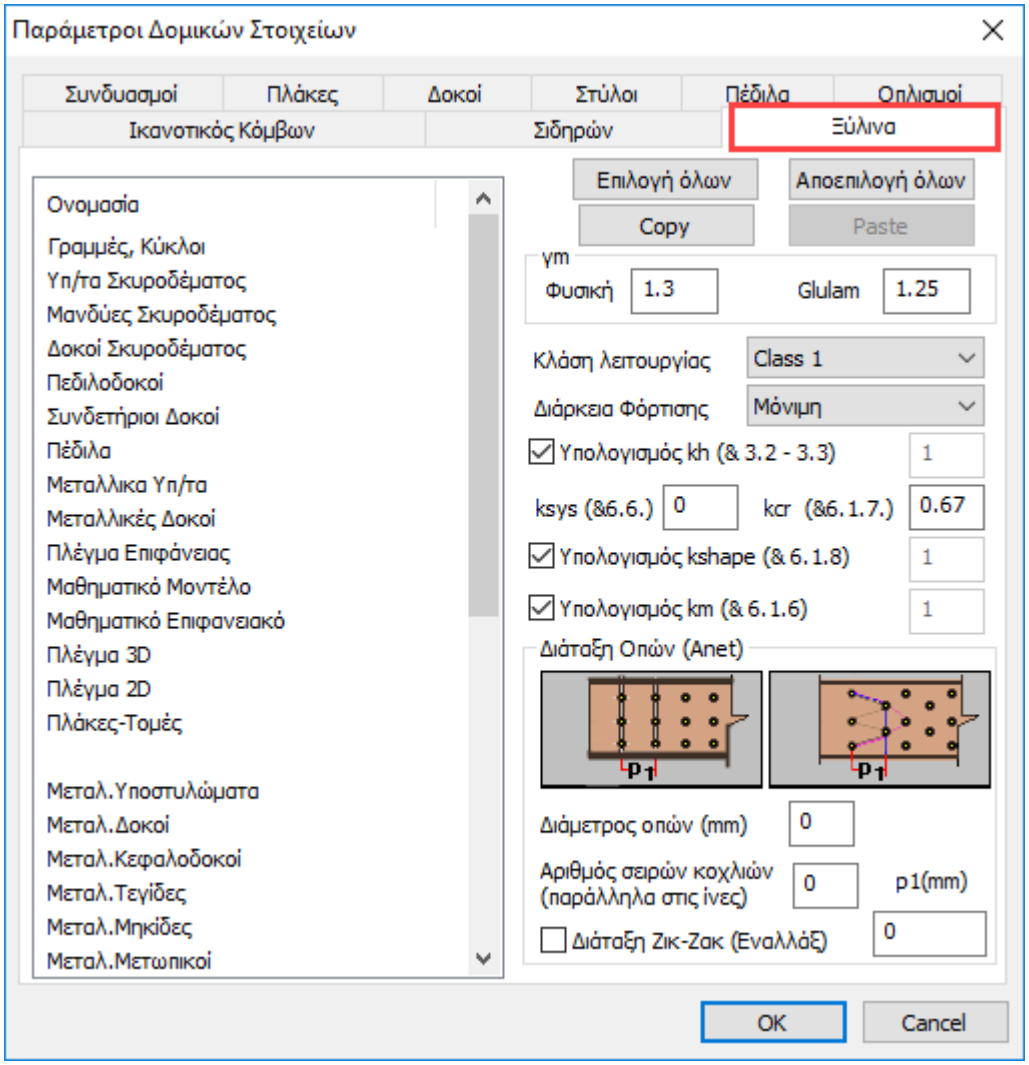
For all checks set the "Safety Factor", i.e. the ratio between a design value and the corresponding resistance value, which is 1 by default.





1.3.9 Wooden

The next section deals with the parameters of the cross-section control of **wooden structures**. Selecting the section displays the following dialog box



The definition of the dimensioning parameters of the wooden sections is done layer by layer. You select the layer whose parameters you want to set (e.g. Wooden Lumber Yards) and set them. Then the program gives you the possibility to copy these parameters to another layer with the logic of Copy And Paste

EXAMPLE

Suppose you have set all the parameters for the Wooden Lodges layer and you want to pass them to the Wooden Beams layer. Select the Wooden Lumber layer, press the "Copy" button and select the Wooden Beams layer and press the "Paste" button which is already activated. Now all the parameters of the Wooden Lumber Yards layer have been passed to the Wooden Beams layer. The parameters that the designer is asked to define are then explained in detail on a case-by-case basis:

Safety factors gM

The values of the safety factors γM for materials shall be used for cases of normal or accidental loading (accidental loading does not include the case of earthquake, to which paragraph 2.9.4 of this document applies).

Fundamental combinations of actions	γ_M	
Natural timber (Solid)	1.3	Members
Glued laminated timber (Glulam)	1.25	

Indicative values (EC5 - Table 2.3) are presented in the table below:

Class of operation of structures - influence of moisture content percentage (MoC)

Humidity is related the environmental conditions in which the structure (or member) operates, i.e. temperature and relative humidity. The Regulation defines three categories of operation (EC5 - 2.3.1.3):

Category of operation	**Temperature and relative humidity environment**	**P.P.Y.**	**Examples**
1	Temperature 20° C The relative humidity exceeds 65% only for a few weeks a year	(9±3) %	Closed structures that heated (hot roofs, floors and interior walls)
2	Temperature 20° C The relative humidity exceeds 85% only for a few weeks a year	(12±3) % (15±3) %	Enclosed structures not heated or periodically heated (e.g. holiday homes) Open roofed constructions, cold roofs, external walls and more generally structures that are not directly exposed to the weather phenomena
3	Climate conditions leading to a rate of moisture content	> 19 %	Structures in damp places or structures exposed to

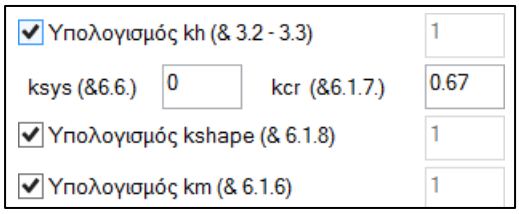
	greater category 2		weather conditions (e.g. direct wetting)

Charging duration categories

The strength of a member decreases with the duration of the load, which is why the Regulation divides actions into categories according to their duration (EC5 - 2.3.1.2):

Duration category	Period of time	Load example
Permanent	over 10 years old	same weight
Long term	6 months to 10 years	storage loads
Midyears	1 week to 6 months	imposed floor loads
Short-term	up to 1 week	snow
Instant		wind, accidental loads

The coefficients **Kh**, **Kshape**, **Km** are automatically calculated by the program according to the respective chapters of EC5. The designer can define his own values by unchecking the checkbox and entering the value in the corresponding field.



- **Kh** is an incremental factor and depends on the type of wood, the size of the member and the type of loading.
- **Ksys** is an incremental factor relating to continuous cross-sectional systems of cargo
- **Kcr** is a decreasing coefficient with a constant value of 0,67 for shear
- **Kshape** is incremental coefficient is incremental torsion
- **Km** involves biaxial bending, alternately reducing one of the 2 torques

6.6 Αντοχή συστήματος

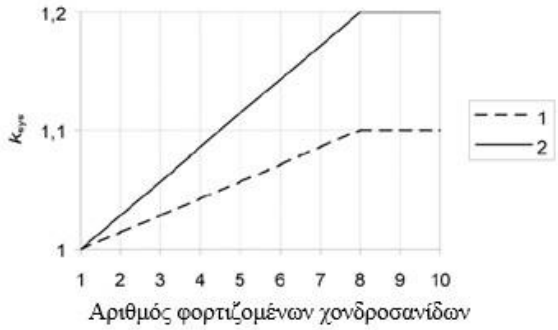
(1) Τα μεγέθη αντοχής ισαπεχόντων παρομοίων απλών μελών, μελών σύνθετης διατομής ή δομικών συστημάτων, που συνδέονται εγκαρσίως μεταξύ τους με ένα συνεχές σύστημα διανομής των φορτίων, μπορούν να πολλαπλασιάζονται επί τον συντελεστή αντοχής συστήματος k_{sys} .

(2) Με την προϋπόθεση ότι το συνεχές σύστημα κατανομής των φορτίων είναι επαρκές για να μεταφέρει φορτία από ένα μέλος στα γειτονικά του, ο συντελεστής k_{sys} θα πρέπει να λαμβάνεται ίσος προς 1,1.

(3) Ο έλεγχος αντοχής του συστήματος κατανομής των φορτίων θα πρέπει να γίνεται με την παραδοχή ότι τα φορτία αποτελούν βραχυχρόνια φόρτιση.

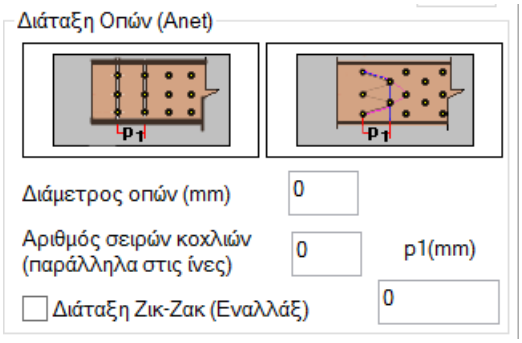
ΣΗΜΕΙΩΣΗ: Για δικτυώματα στεγών τοποθετημένων σε αξονικές αποστάσεις όχι μεγαλύτερες από 1,2 m, μπορεί να θεωρηθεί ότι τα καδρόνια επικεραμώσεως, οι τεγίδες, ή επιφανειακά στοιχεία που εδράζονται στα δικτυώματα μπορούν να διανείμουν το φορτίο στα γειτνιάζοντα δικτυώματα, με την προϋπόθεση ότι όλα τα παραπάνω δομικά μέλη κατανομής φορτίου καλύπτουν τουλάχιστον δύο ανοίγματα, και είναι κλιμακωτά διατεταγμένα.

(4) Για ξύλινα ορθόστρωτα καταστρώματα ή πατώματα θα πρέπει να χρησιμοποιούνται οι τιμές του k_{sys} που δίδονται στο Σχήμα 6.12.



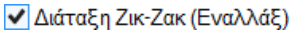
Υπόμνημα:
1 Ηλωμένες ή κοχλιωμένες χονδροσανίδες
2 Προεντεταμένες ή συγκολλημένες χονδροσανίδες

Σχήμα 6.12 Συντελεστής αντοχής συστήματος, k_{sys}, για ορθόστρωτες πλάκες καταστρώματος από φυσική ξυλεία ή μέλη από συγκολλητή ξυλεία



Finally, in the *Hole Layout* field you define an initial approximate layout of the holes in the wooden member that is used for the specification of the members and is then checked when solving the connections.

Set the diameter of the holes and the number of rows of bolts, as well as the distance p1 according to the two drawings. To define a Zig-Zag layout you select the corresponding checkbox



1.4 Consolidation of Members

In the new version of the program a new group of commands has been added which concerns the consolidation of metallic members for the calculation and display of bending and deformation checks based on EC3.

Important notes:

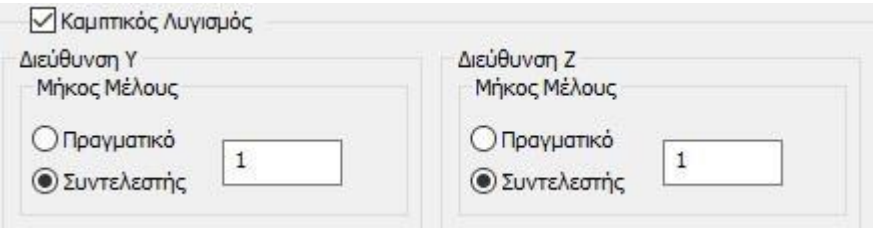
- With the use of this tool, it is now possible for the designer to correctly define the initial length of the member per direction to be taken into account in the

  controls of bending. 

- This determination has so far been made using the known coefficients (see next section

)



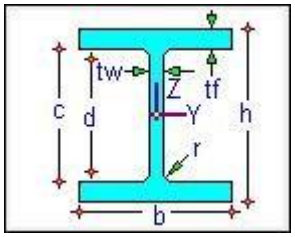
- Now with the use of the consolidation by direction, the process of the rates will not be needed, but the consolidation will be done, in most cases automatically.
- It should also be noted that with the process of consolidation it is correctly taken calculating the bending length and in printing the results a consolidated member is now printed once with an indication of the individual members it includes.

Basic concepts of bending about strong and weak axis.

What does the corresponding bending length ly and lz mean:
It is recalled that, in general, in double-tau metallic cross-sections, the local axis

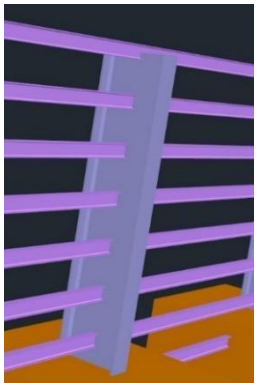
- **y-y** is the **strong**, and the
- **z-z** the **patient**,

as shown below:

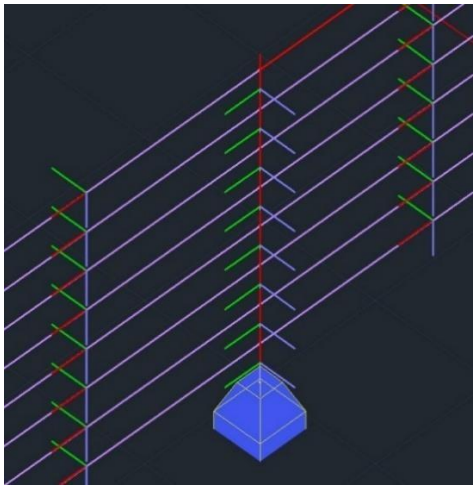


EXAMPLE:

For the example, we will look at the bending lengths of the following post connected laterally to the thwarts. We will first look at the initial bending lengths Ly and Lz for the post.



The times of the local axes of the pole and the thimbles are as follows:



Bending of the **pole** about its local **strong y-y** axis (green) means:

- the buckling caused by the moment My (rotation about the y-y axis), i.e. out-of-plane buckling, which in the particular case, the unified, i.e. the total length of the post shall be taken as the buckling length.

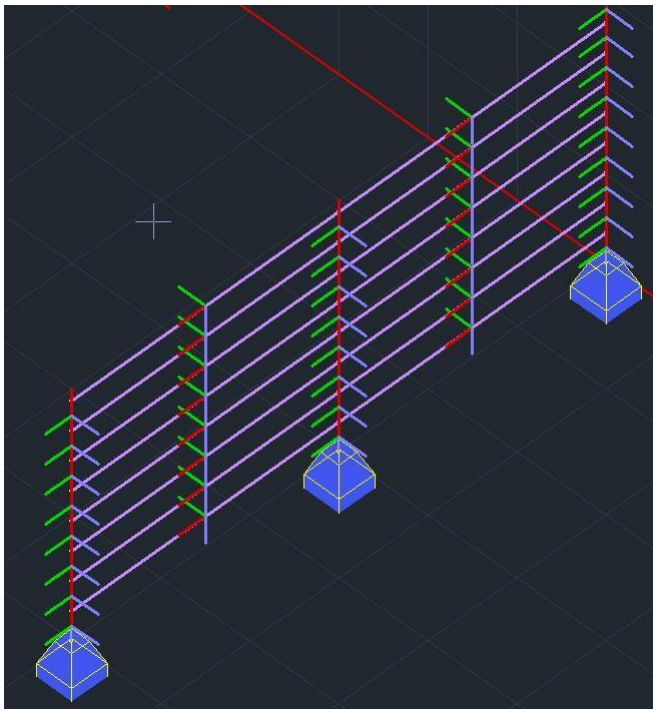
In the other direction, bending the pole about its local **weak z-z** axis (blue) means:

- the bending caused by the torque Mz (rotation about the z-z axis), i.e. bending in-plane, the post is considered to be supported laterally by the thwarts and thus the bending length Lz will be taken as the length of each member.

- **NOTE**

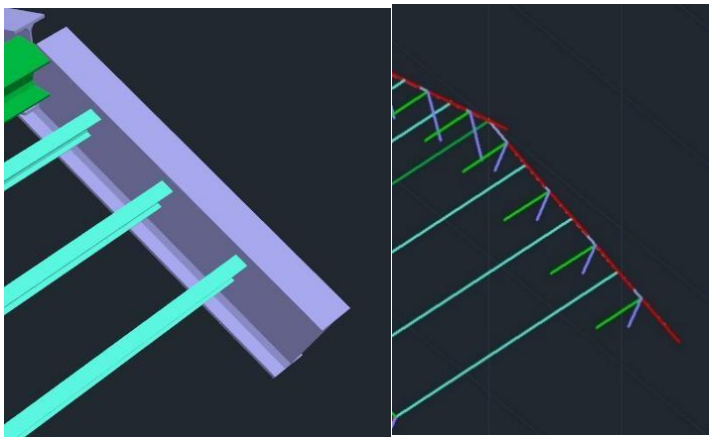
As a general rule, we could say that, we obtain **unified Ly length** in direction where the local y-y axis is parallel to the elements supporting - securing the member, while in the other direction, if there are no elements, **the individual lengths** are taken as **Lz**.

In the same example and with regard to the **thymus:**



The supports from the poles are parallel to the local z-z axis (blue, out of plane) of the thimbles. So, the consolidation will be done in Lz (full length), while for the y-y direction (green, in-plane) the length of each member will be taken as Ly.

Similarly for the following **inclined beam**:



The local axis of the beam parallel to the tesserae is y-y. Therefore, Ly will be taken as the unified of the whole beam, while Lz will be taken as the individual lengths.

The consolidation command group includes the following commands:

The logic of the consolidation methodology is that, either automatically or manually, the individual members of an element are consolidated by bending direction.

The buckling length taken for calculation purposes is not the actual length of the member, but the unified length from the beginning to the end of the column or beam respectively.

In addition, in the presentation of the results, for these consolidated members the worst controls are shown only once and not for each one as was the case until now.

Finally, in automatic consolidation, there is the definition of stop levels.

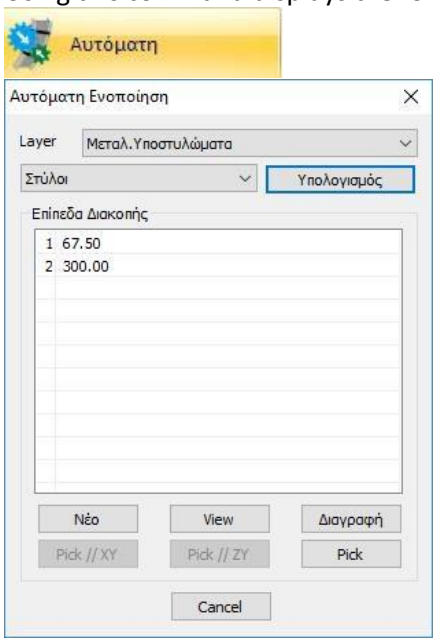
Break planes are horizontal or vertical planes that are used as breakpoints in the consolidation of a continuous element.
For example, for vertical elements (Pillars) the stop levels are horizontal planes which are defined, like the levels, with an altitude.

- **OBSERVATION**

It is good to work with the commands in the 3d mathematical model and have the commands displayed in the
local axes.

1.4.1. Automatic Consolidation

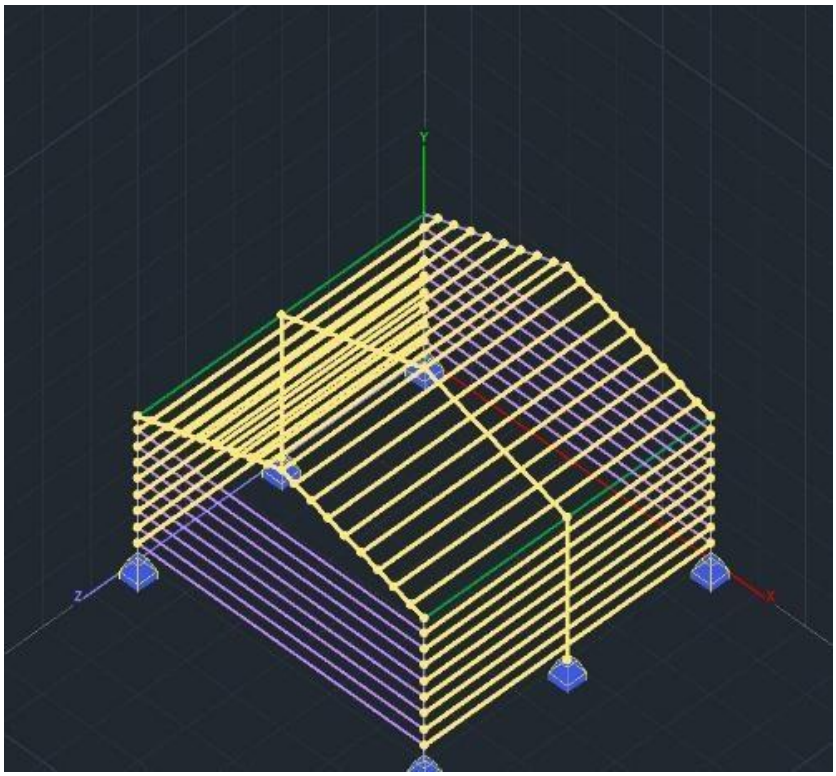
Using this command displays the following dialog box



In the upper field you select the layer of the elements you want to consolidate. The

Consolidated members are displayed in colors on the screen.

- The y-y consolidated local data are shown in yellow.
- In cyan colour the z-z consolidated local
- In pink are the consolidated along both axes



Right below you specify the type of element contained in the selected layer. The program automatically understands the type of element if it is vertical (Pillars) and all other elements are Beams.

With the "**Calculate**" command the program consolidates the data of the specific layer based on the above mentioned.

The next section deals with the definition and processing of stop levels.
Interruption levels are levels that are boundaries of the beams and poles where you want the integration for either one or the other direction to be interrupted.

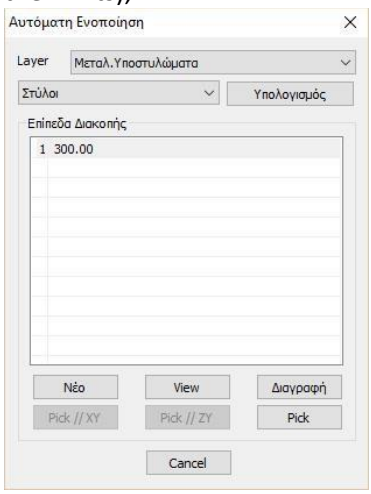
- For poles, the stopping levels are horizontal levels defined by the altitude.
- For beams, the stop planes are always vertical planes defined by two points.

Predefined limits:

- for the horizontal planes the horizontal planes are the foundation level and the upper last level (the last level).
- and for the beams are the vertical limits of the girder.
- The default limits are not shown in the table of cut-off levels.

EXAMPLE:

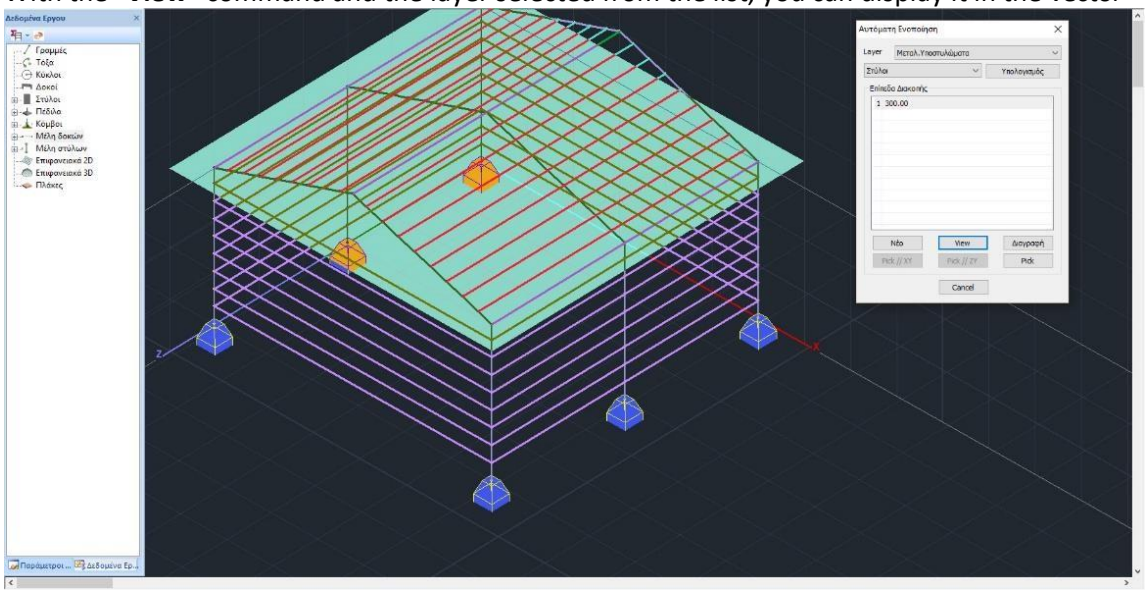
For example, in a building with levels at 0.00, 300.00 and 600.00, the table with the cut-off levels for the poles will by default show only the 300.00 level (i.e. only the intermediate level without the limits),



on the grounds that if the poles are consolidated, this will be interrupted at 300.00 cm, i.e. the pole from 0.00 to 300.00 cm and the pole of the other floor from 300.00 to
600.00 cm.

- To set your own cut-off level for **PENS**:

you select the "**New**" button and then with the "**Pick**" command you point to a point.
The horizontal plane defining the altitude of this point is a stopping level.
With the "**View**" command and the layer selected from the list, you can display it in the vector

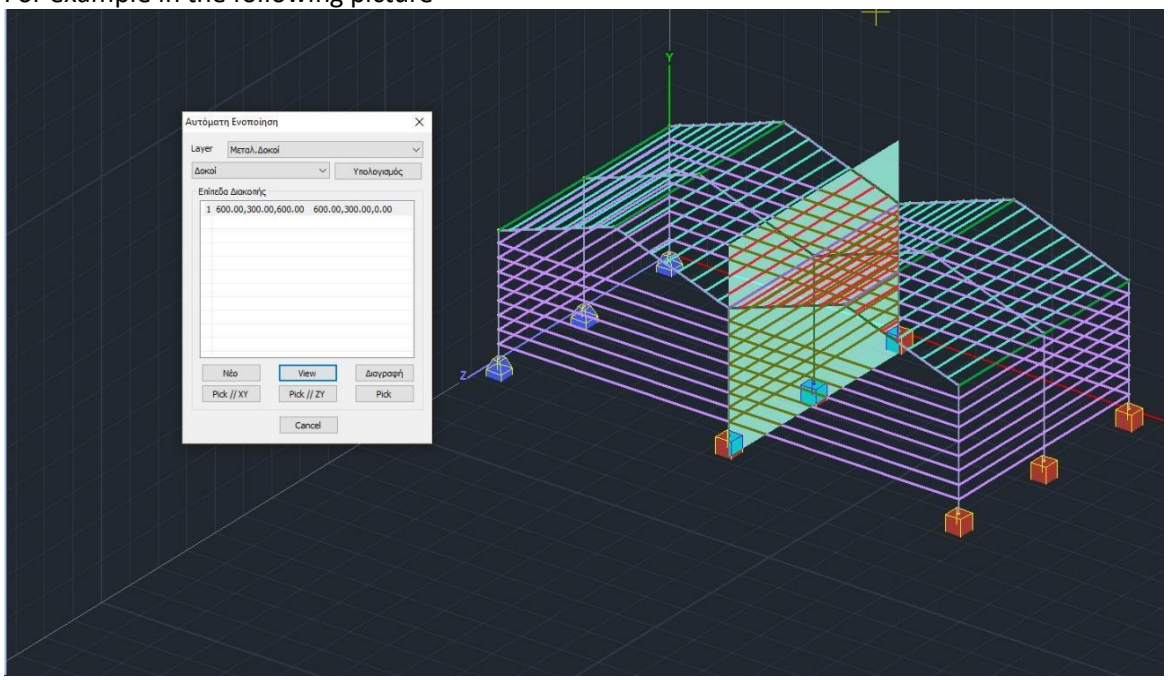


Finally with the "**Delete**" command you can delete an interrupt level.

- Regarding **the DOKUS**:

the definition of vertical stop levels is done in the same way, but here with "**Pick**" you define two points, i.e. a line defining a vertical stopping plane.

For example in the following picture



the stopping level of the front and rear plinths is defined as the boundary of the two buildings.

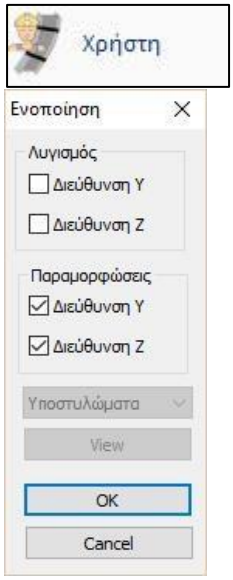
- Especially for the beams, and when the stop levels you want to set are parallel to Global XY or ZY, you select the corresponding button and you now set only one point.

For beam and post break levels the *correction* can be done in two ways.

- Either by deletion and appointment of a new one,
- either by selecting the corresponding level and redefining with a pick point or points.

1.4.2. User Consolidation

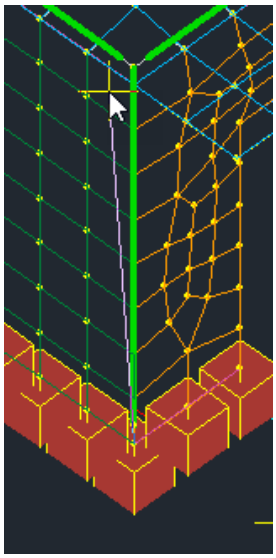
Select the command and then point to the start and end points of the members you want to consolidate.
Selecting the second point (end point) displays the following dialog box:



where you set the direction of unification for Bending and Deformation.

1.4.5. Consolidation of concrete columns

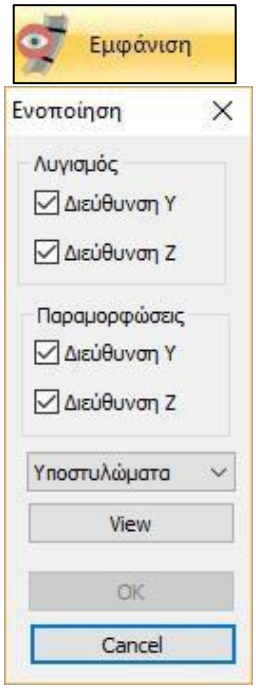
This command is mainly used in masonry buildings with vertical reinforced concrete elements that connect the nodes of the surface nodes and that need to be consolidated in order to be dimensioned.



Select the command and then point to the start and end points of the members you want to consolidate.

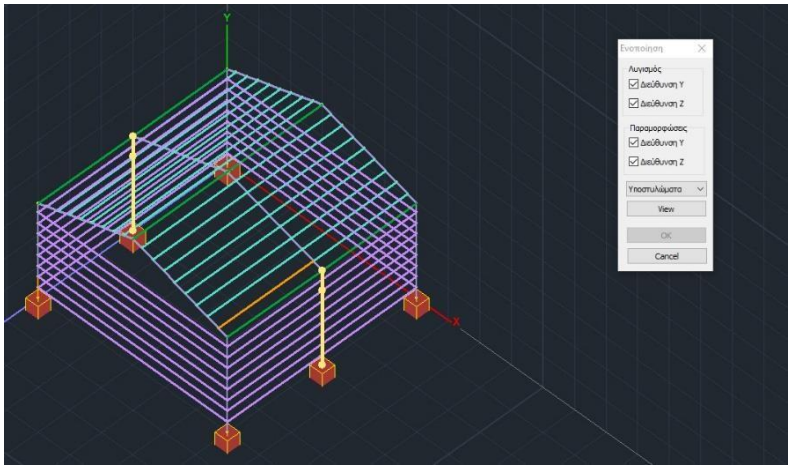
1.4.5. Show

By selecting the Show command, you can see which elements have been consolidated based on the colours mentioned above.
The following dialog box appears:



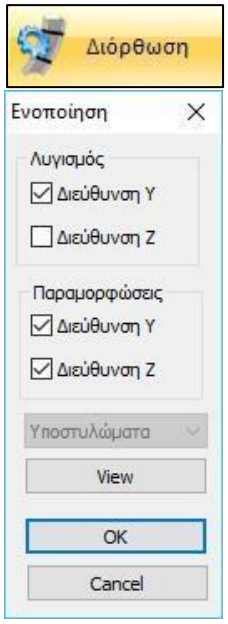
The use of the mask is simple:

- Select the type of item you want to view from the list, and then
- click type (Bending - Deformations) and direction you want to see which data are consolidated.



1.4.6. Correction

The "Correct" command allows you to correct items that have already been consolidated.
By using the command and selecting any member that belongs to consolidated member, the following dialog box appears:



where the corresponding fields that you have consolidated appear checked.
Here you can modify the consolidation options based on direction and type. With the View option you can view the member with the corresponding color of the consolidation.

- **ATTENTION**

With this command you must select a member that has already been consolidated. If you select a else the dialogue box will not open.

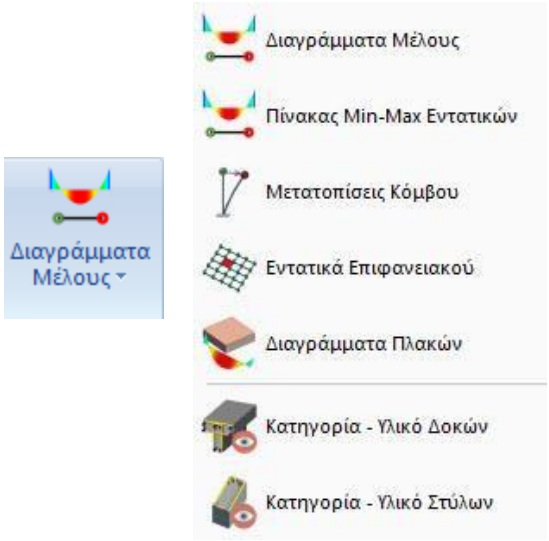
1.4.7. Selective deletion

Command to selectively delete a consolidation.
You select the command and then click any member that belongs to consolidated one. Then right-click to delete the consolidation.

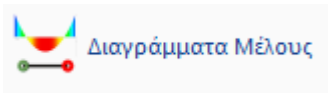
1.4.8. Total deletion

Command to delete all consolidations of your organization in total. Command useful if you want to define the consolidations from the start

2. Member Charts

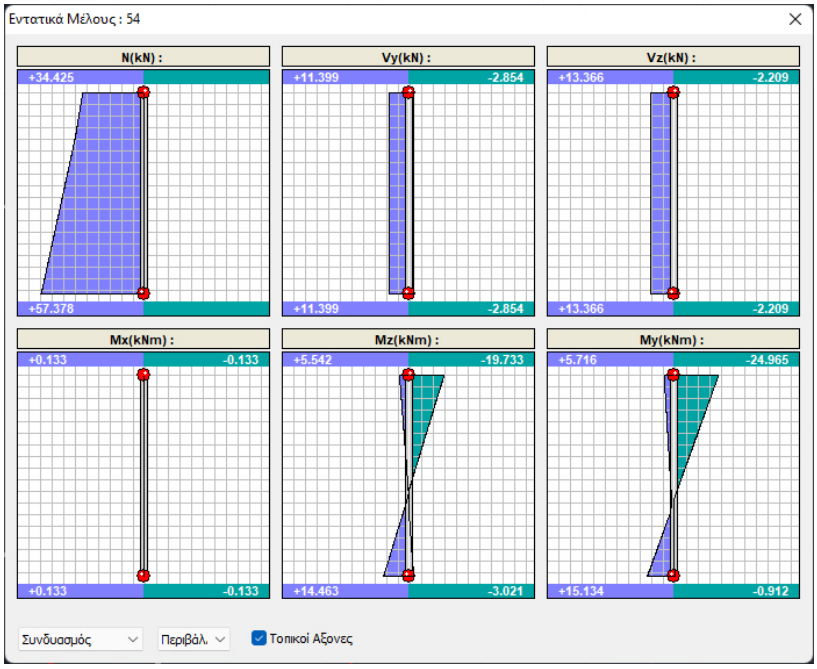


2.1 Member Charts



Command to display the member charts.

Select the command and point to a member with the mouse. In the dialog box that opens, all the diagrams of the intensive magnitudes of that member are displayed. Select a combination or load and move the mouse within the frame of the diagrams to read the corresponding values of the intensive magnitudes along the member.



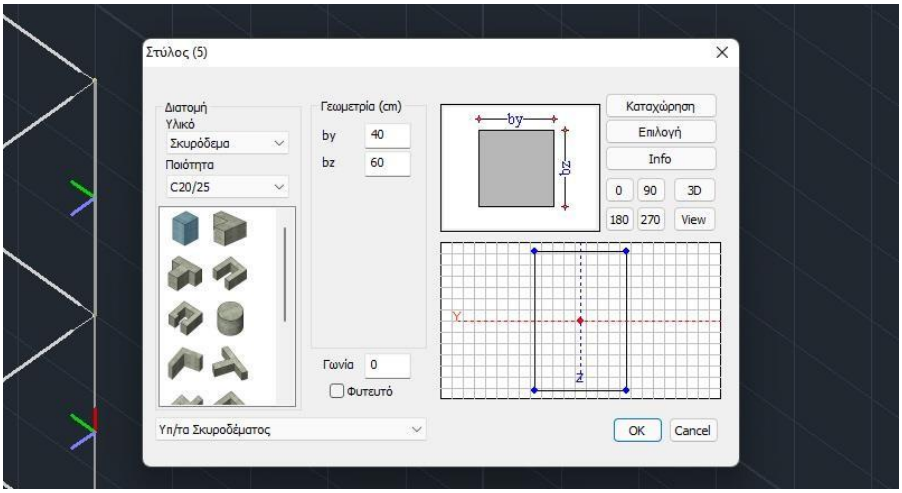
⚠ OBSERVATIONS:

1. A new check has been added to the display of the column-wall diagrams, by unchecking which the diagrams of the intensities are converted from the local axes (as derived from analysis results) to the axes of the dimensioning. The dimensioning axes are in fact the axes for beta=0 and positioning angle=0

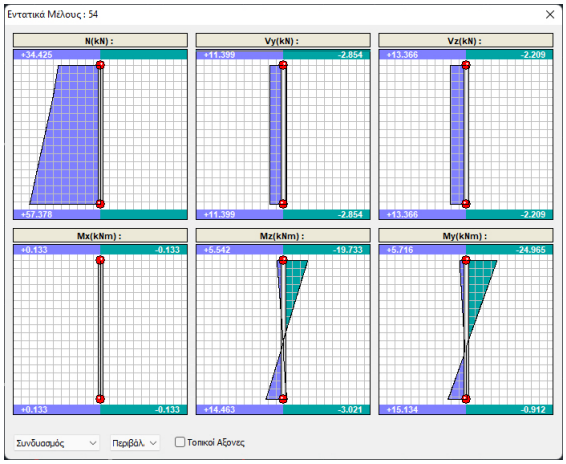
In order for all columns - walls to have common reference axes for their stresses and reinforcement, the dimensioning is done with respect to the dimensioning axes (these are the axes for angle beta = 0 and placement angle 0). The system of dimensioning axes has a rotation with respect to the local system of each member by angle Beta minus the placement angle of the element. Thus, for a member with angle beta = 0 and angle of emplacement = 0, the local axes and the dimensioning axes coincide.
Recall that when a column is placed with a placement angle other than zero, the corresponding linear element created has a beta angle equal to the placement angle.

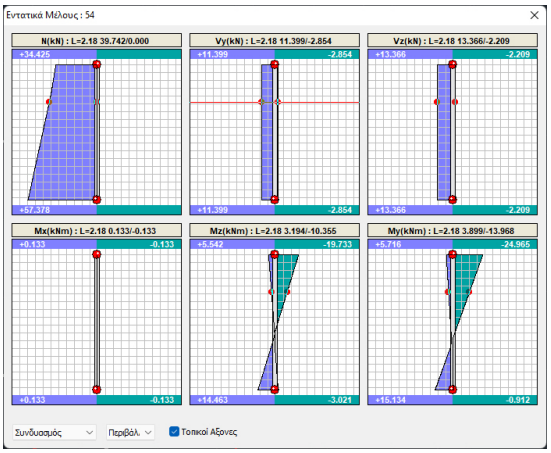
EXAMPLE for local axes

Mounting angle 0, angle beta=0
In the image below



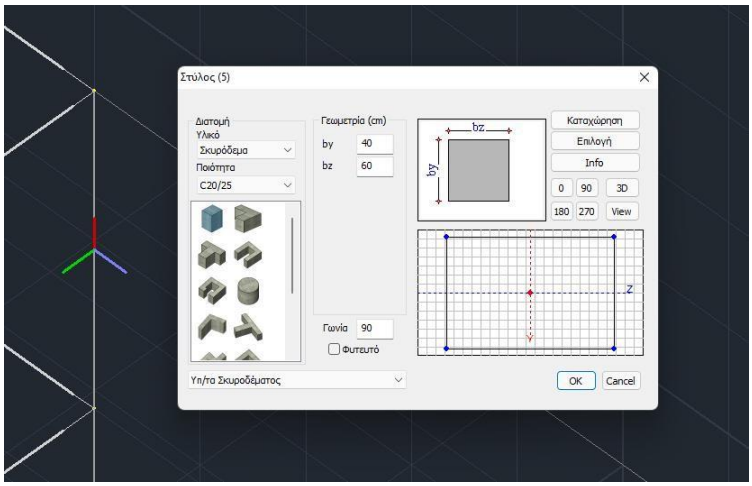
The upper post is inserted with a **mounting angle of 0 and therefore an angle beta=0**. Thus, its local axes are the same as the dimensioning axes. In the diagrams of the element's stresses, the diagrams on the local axes therefore coincide with the diagrams of the dimensioning



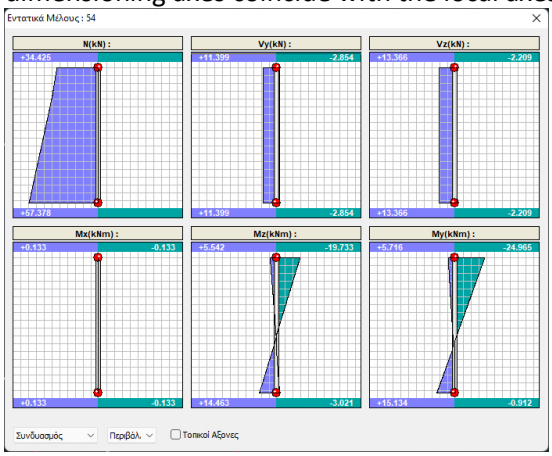


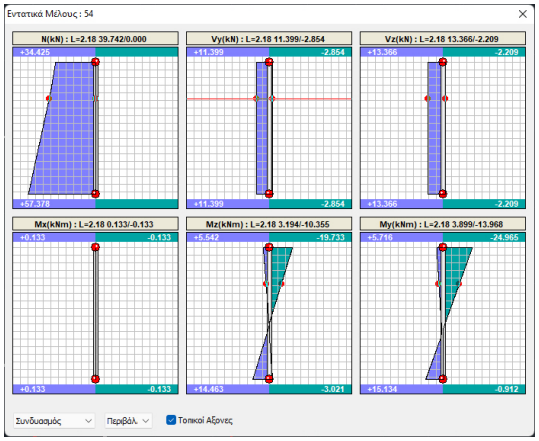
Mounting angle 90, angle beta=90

If the same element is now placed at **a 90 degree placement angle, (beta angle 90 degrees)**, the local axes are these (turn counterclockwise)



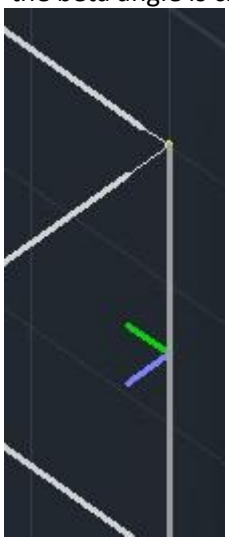
The situation will not change because the rotation angle (beta angle - mounting angle) is 0, the dimensioning axes coincide with the local axes and so the diagrams are again the same.



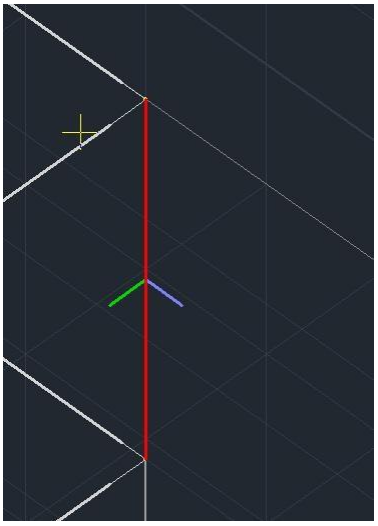


Ex-post change of beta angle

If finally the beta angle is changed afterwards from 0 degrees to 90 degrees (counter-rotation),

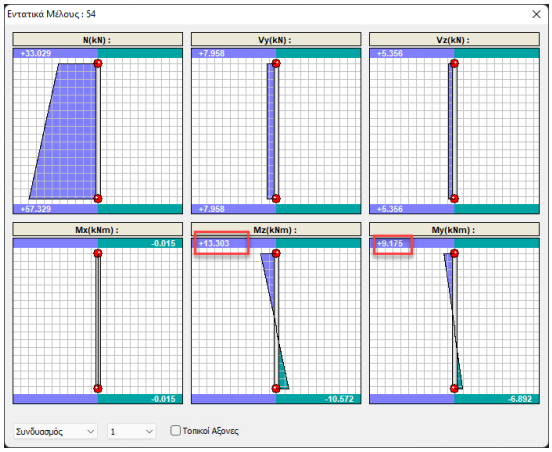


0 degrees

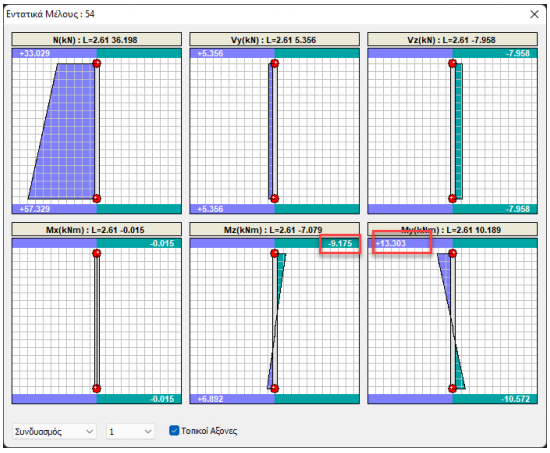


90 degrees

the dimensioning axes are now twisted by -90 degrees, i.e. they are in the 0 degree state. So the dimensioning torque My(0) is equal to -Mz(90) of the local and the torque Mz(0) sizing is equal to My(90) of local.



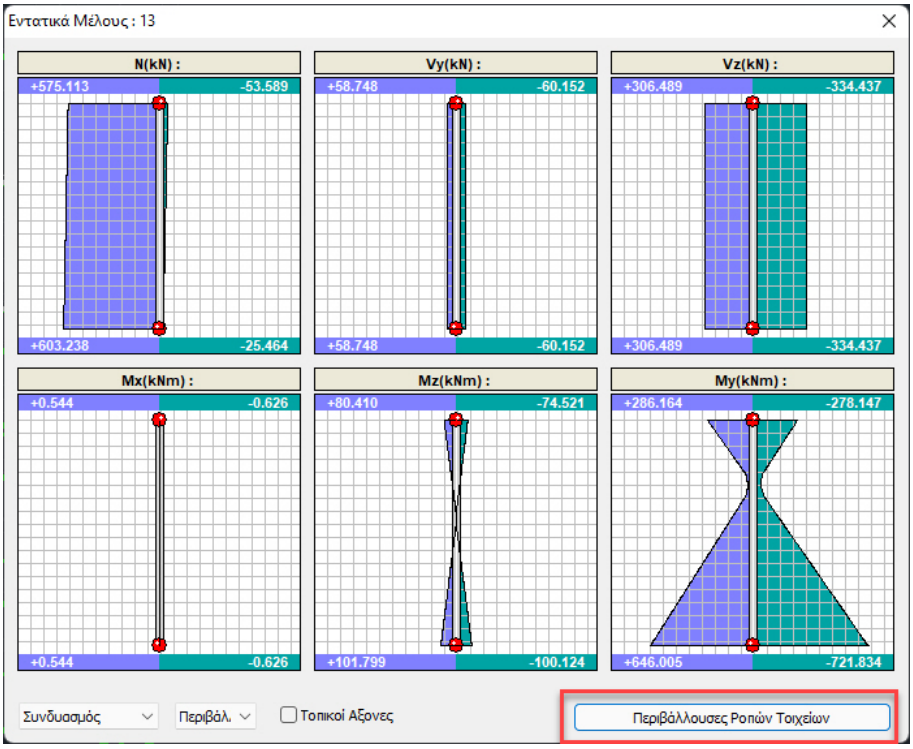
0 degrees (sizing)



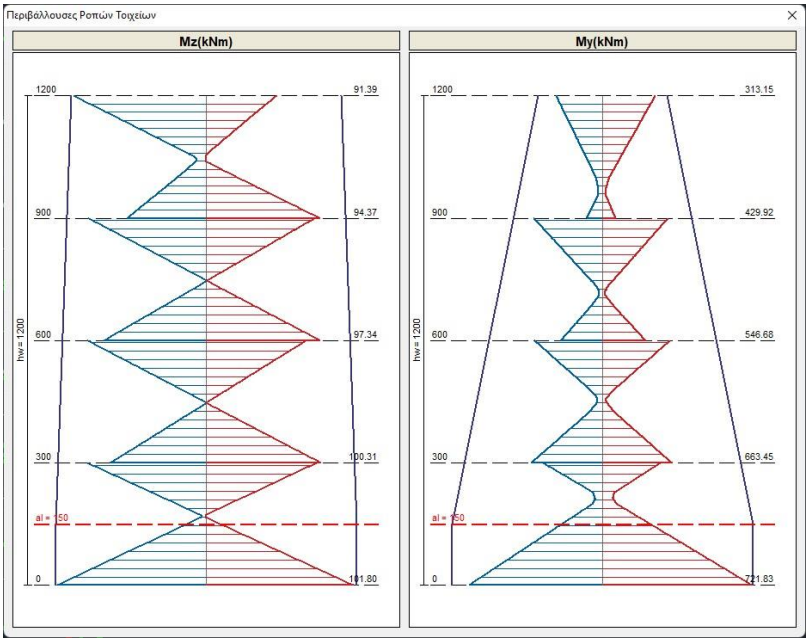
90 degrees (local axes)

My(0)=-Mz(90)= 9.175 =-(-9.175)
Mz(0)=My(90)=13.3

2. The new version of SCADA Pro has incorporated the design procedure of the normative envelope of the bending moment according to paragraph 5.4.2.4(4,5) of EC8 for the satisfactory design of walls. In particular, in the dialogue box of the diagrams in the dimensioning and when the element is a wall, the option "Wall Moment Envelopes" is displayed



which displays the following image

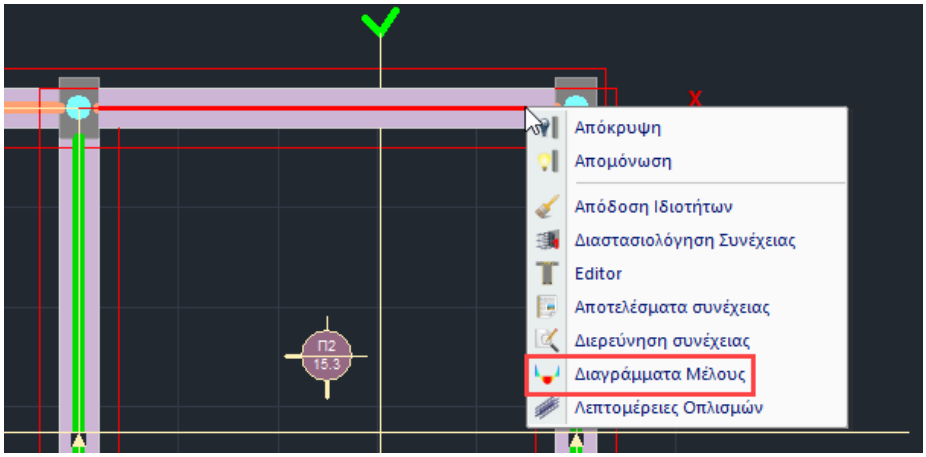


the envelope shall be plotted vertically for the whole wall and at each level a torque value per direction shall be calculated which is the largest absolute value of the corresponding positive and negative values.

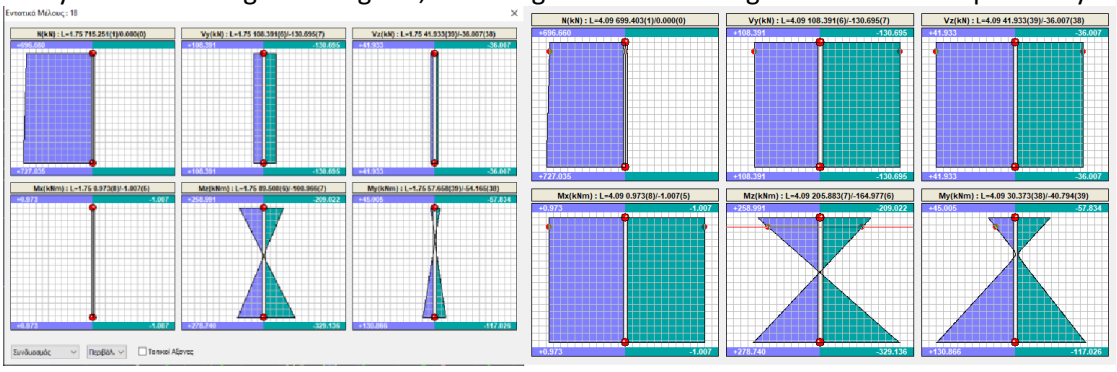
For example, in the graph above:
The two blue outer lines on either side are the envelopes and the "sawtooth" diagrams are the original envelopes.
For level 1 (300) the value of Mz to be considered is Mz=100.31 kNm and the torque of My=663.45 kNm.

Finally, in the section on the investigation of the results, the values calculated and taken into account are listed in detail and by combination.

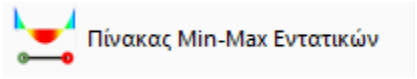
3. Right-clicking on the member opens a list of commands that includes the Member Diagrams command.



4. By double-clicking on a diagram, each diagram will be enlarged to show its shape clearly.

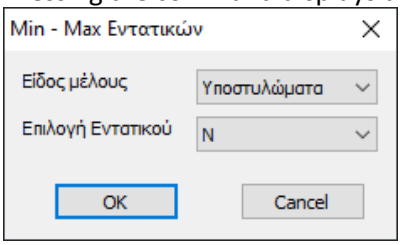


2.2 Table Min-Max Intensives



You can now see the diagrams with the Min-Max intensive sizes in tabular form.

Pressing the command displays the following communication window

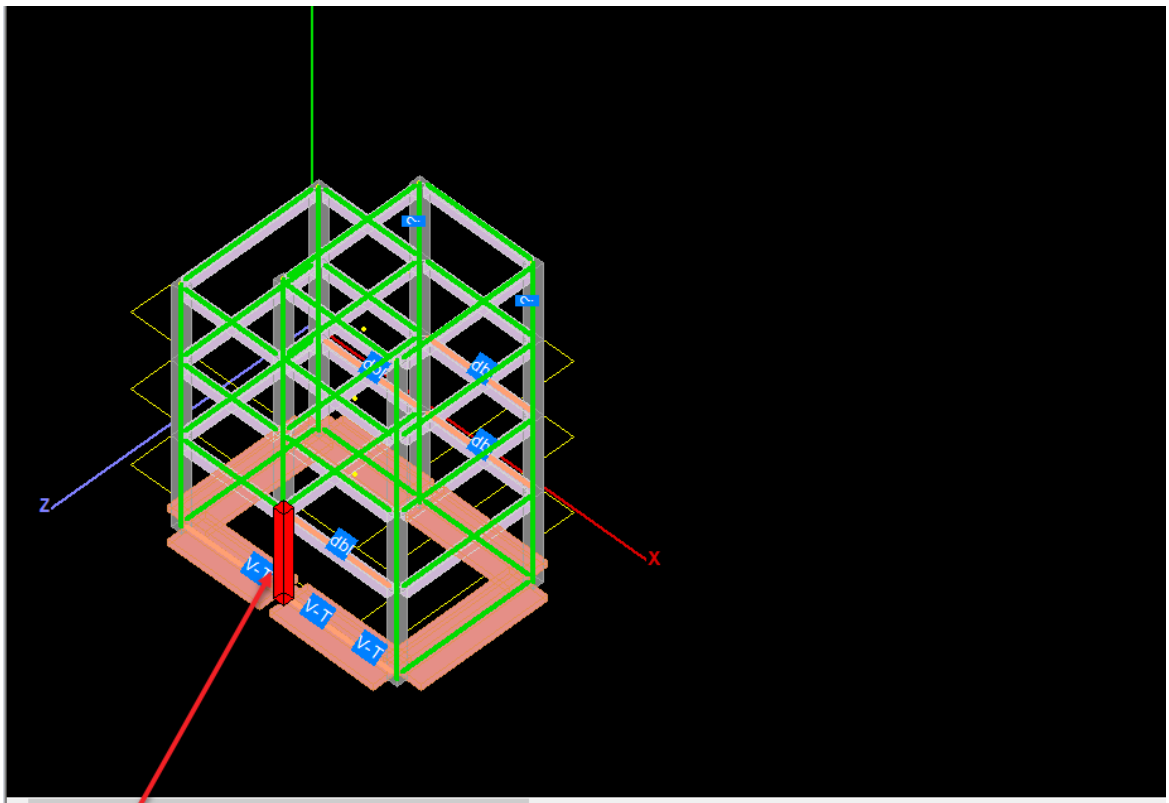


, in which you are asked to enter the type of member you wish to view the intensives, as well as the (initial) intensive size you wish to investigate.

Ονομα	Min-Max	Συνδ.	L (m)	N (kN)	Vy (kN)	Vz (kN)	Mx (kNm)	Mz (kNm)	My (kNm)
11	Max	1	0.0000	1214.4580	64.0065	15.6819	1.2453	129.6775	37.7756
11	Min	1	3.6000	1195.0180	64.0065	15.6819	1.2453	98.2310	68.2419
12	Max	1	0.0000	943.8152	34.3001	64.8409	1.2453	59.0216	162.0816
12	Min	1	3.6000	924.3752	34.3001	64.8409	1.2453	66.8870	81.3490
8	Max	1	0.0000	854.2269	54.7029	32.0265	1.2453	112.4745	48.1420
8	Min	1	3.6000	834.7869	54.7029	32.0265	1.2453	78.2952	19.8092
24	Max	1	0.0000	826.6706	52.3440	3.9750	1.2316	75.1374	0.0000
24	Min	1	3.0000	810.4706	52.3440	3.9750	1.2316	96.0096	88.8653

Apart from the initial selection of intensive size you are exploring, you are given the option to tap on any intensive size and it will display - first in ascending order (by tapping once) or descending order (by tapping twice) - the list of the corresponding members and their intensities.

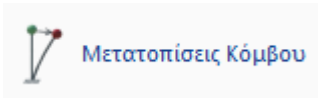
Each member selection from the list is also visible in the model for the best possible overview.



Πίνακες

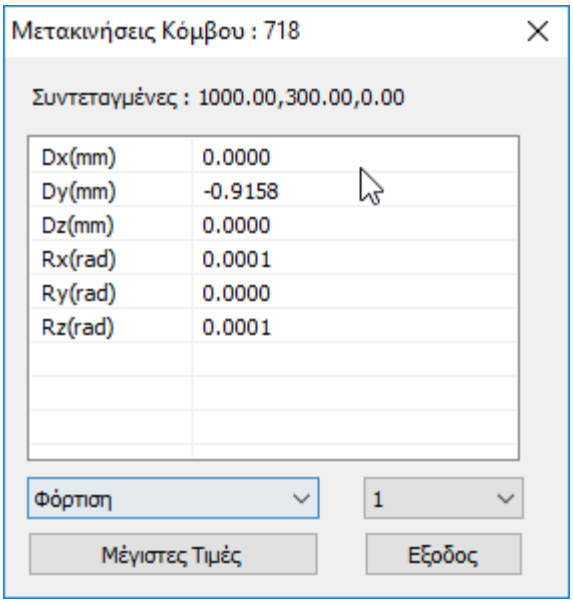
Ονομα	Min-Max	Συνδ.	L (m)	N (kN)	Vy (kN)	Vz (kN)	Mx (kNm)	Mz (kNm)	My (kNm)
11	Max	1	0.0000	1214.4580	64.0065	15.6819	1.2453	129.6775	37.7756
11	Min	1	3.6000	1195.0180	64.0065	15.6819	1.2453	98.2310	68.2419
12	Max	1	0.0000	943.8152	34.3001	64.8409	1.2453	59.0216	162.0816
12	Min	1	3.6000	924.3752	34.3001	64.8409	1.2453	66.8870	81.3490
8	Max	1	0.0000	854.2269	54.7029	32.0265	1.2453	112.4745	48.1420
8	Min	1	3.6000	834.7869	54.7029	32.0265	1.2453	78.2952	19.8092
24	Max	1	0.0000	826.6706	52.3440	3.9750	1.2316	75.1374	0.0000
24	Min	1	3.0000	810.4706	52.3440	3.9750	1.2316	96.0096	88.8653

2.3 Node displacements

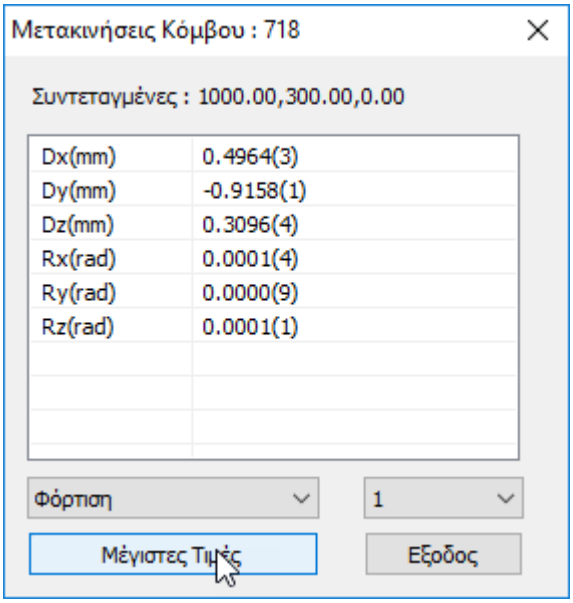


It is now possible to see the displacements of the nodes.

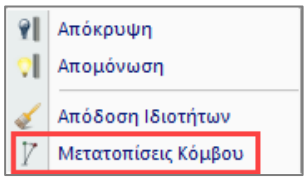
Select the command and point to a node with the mouse. In the dialog box that opens, all moves of that node are displayed. Select a combination or a charge to read the corresponding values of the movements.



Select Maximum Values to see the maximum value for each move and turn, the combination from which it comes. The number of that combination is shown in brackets next to the value.



⚠ OBSERVATION

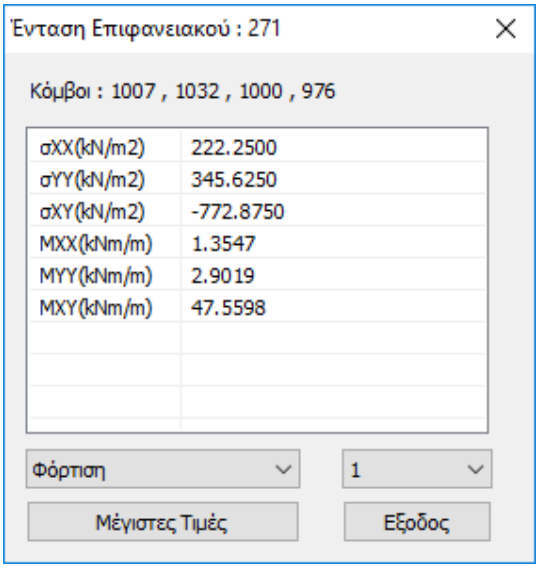


Right-clicking on the node opens a list of commands that includes the Node Shift command.

2.4 Intensive Surface

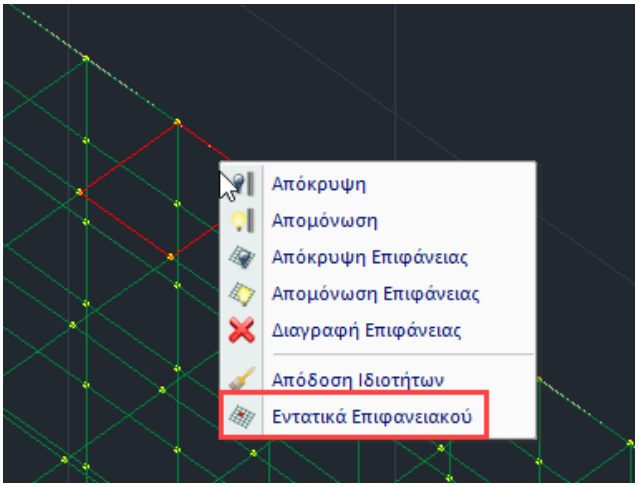
It is now possible to view the intensities of the surface elements.

Select the command and point to a surface element with the mouse. The dialog box that opens displays all the trends and moments for that surface element. Select a combination or load to read the corresponding values of the stresses and moments.



Select Maximum Values to see the maximum value for each voltage and torque, and the combination from which it is derived. The number of this combination is shown in brackets next to the value.

⚠ OBSERVATION



Right-clicking on the node opens a list of commands that includes the Surface Intensity command.

Similarly, if the element is solid, by selecting "Surface Intensities" and pointing to the element, its 6 trends are displayed in the following window:

Ένταση Επιφανειακού : 308

Κόμβοι : 644 , 617 , 645 , 669 , 1551 , 1550 , 1545 , 1546

	Face 0	Face 1	Face 2	Face 3	Face 4	Face 5	Face 6
σXX(kN/m2)	-1.9084	-1.9288	-1.8880	-1.9027	-1.9141	-1.1848	-2.6320
σYY(kN/m2)	-29.5794	-29.3450	-29.8138	-29.5523	-29.6065	-29.5796	-29.5793
σZZ(kN/m2)	-0.0000	-0.0000	-0.0000	0.0000	-0.0000	-0.0000	0.0000
σXY(kN/m2)	-0.4934	-0.5147	-0.4720	-0.7837	-0.2030	-0.5027	-0.4841
σYZ(kN/m2))	0.0154	0.0154	0.0154	0.0158	0.0151	-0.0736	0.1044
σZX(kN/m2)	-0.0014	-0.0014	-0.0014	-0.0093	0.0066	-0.0009	-0.0018

Φόρτιση | 2

Μέγιστες Τιμές | Εξοδος

OBSERVATION:

In the current version of the program, the results given by the solid data are those mentioned above. In the next version, planning and valuation will be integrated.

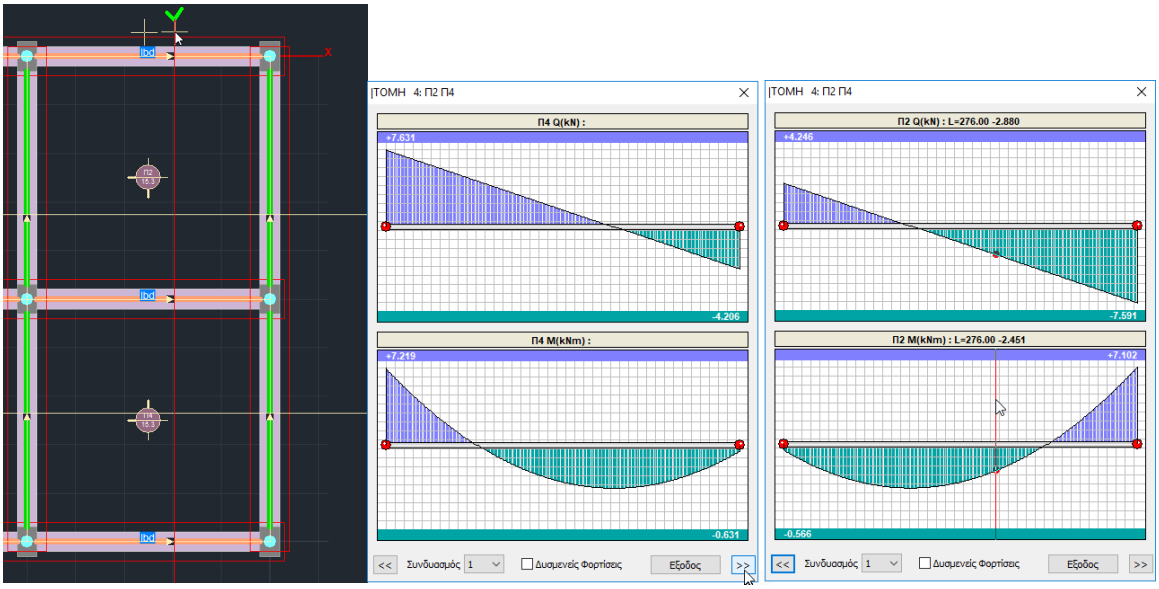
2.5 Plate diagrams

It is now possible to see the diagrams of the cross-sections of the plates. Select the command and point to an intersection with the mouse. In the dialog box that opens, all the diagrams of intensive magnitudes per facet are displayed. Again select a combination or a loading and move the mouse within the width of the diagrams to read the corresponding values of the intensive magnitudes along the section. On

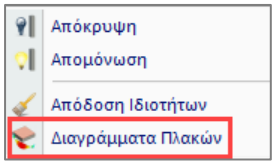
in cases of unfavourable loads, activate the 

to see the resulting diagrams. Go to the next panel with the arrows on the

bottom of the window. 



⚠ OBSERVATION



Right-clicking on the intersection opens a list of commands that includes the Plate Diagrams command.

2.6 Category - Beam Material / Pillar Material

Writing the characteristics of the material on the elements

The new version of SCADA Pro gives you the ability to see on the members (Beams - Columns) the characteristics of the material that has been used, as well as an indication whether the material is a New Member or an Existing Member.
Simply select the command for the Beams and/or the Poles.
The indications are shown separately for Beams and Columns.