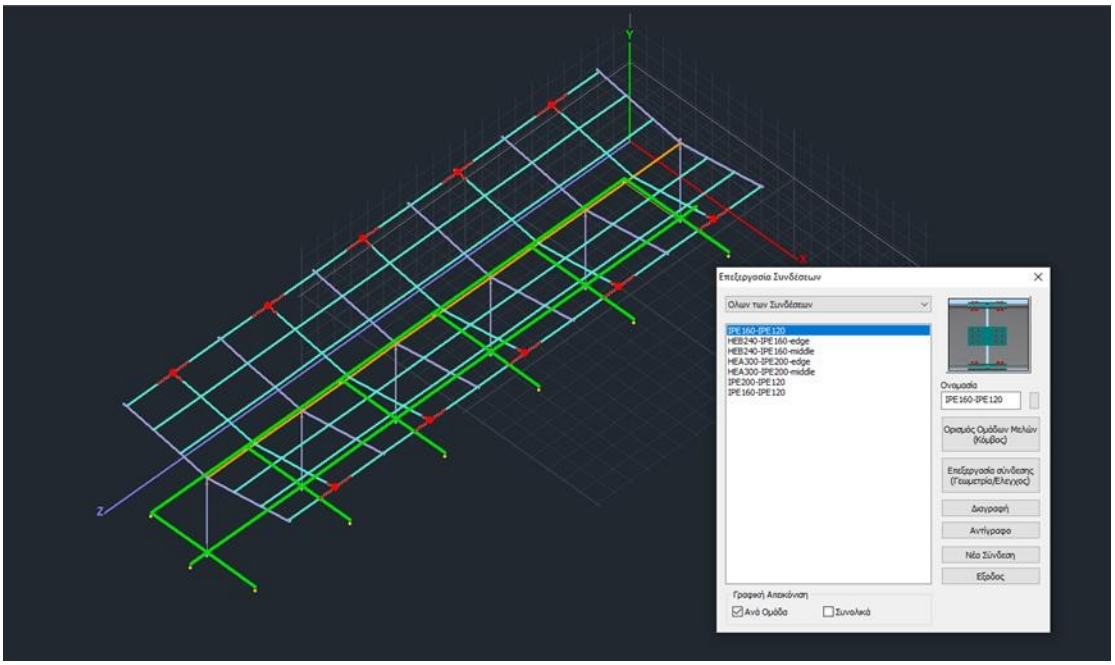
SCADA Pro 25™
Structural Analysis & Design

User Manual

10C. DIMENSIONING

Part 3/4: Iron-Wooden



ACE HELLAS
INTEGRATED SOLUTIONS

CONTENTS

Chapter 10C: Sizing - Iron-Wall (part 3/4)

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

- Scenarios - GENERAL
- Beams
- Equitable Control
- Pillars — BETTON
- Sandals
- Slabs-Mesh
- Iron
- Wooden
- Masonry
- Charts GENERAL

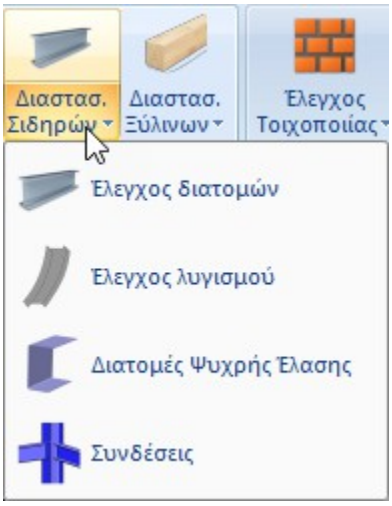
⚠ After the completion of the model, the input of the loads, the execution of the analysis and the creation of the combinations, the "Dimensioning" of the structural elements of the design follows, where the adequacy check is performed, based on the regulation selected in the "Dimensioning scenario"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

The Sizing manual is divided into 4 parts:

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

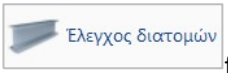
Sizing of Irons



The "Iron" field includes the commands related to the solution of the metallic cross-sections with the adequacy check and buckling check for the **Hot Rolled** cross-sections, the **Cold Rolled** cross-sections and the dimensioning of the **Connections**.

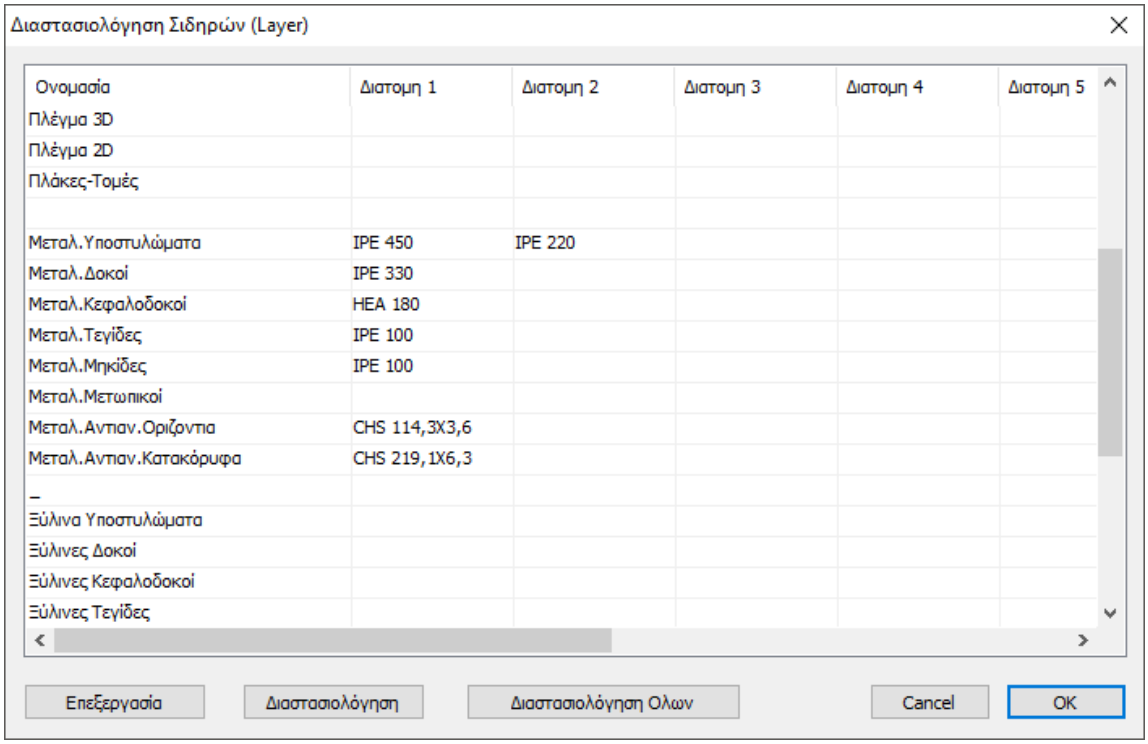
- A prerequisite for sizing is that you have called and executed the corresponding combination file in the configuration dialog box

1.1 Cross-sectional inspection (Hot rolled)

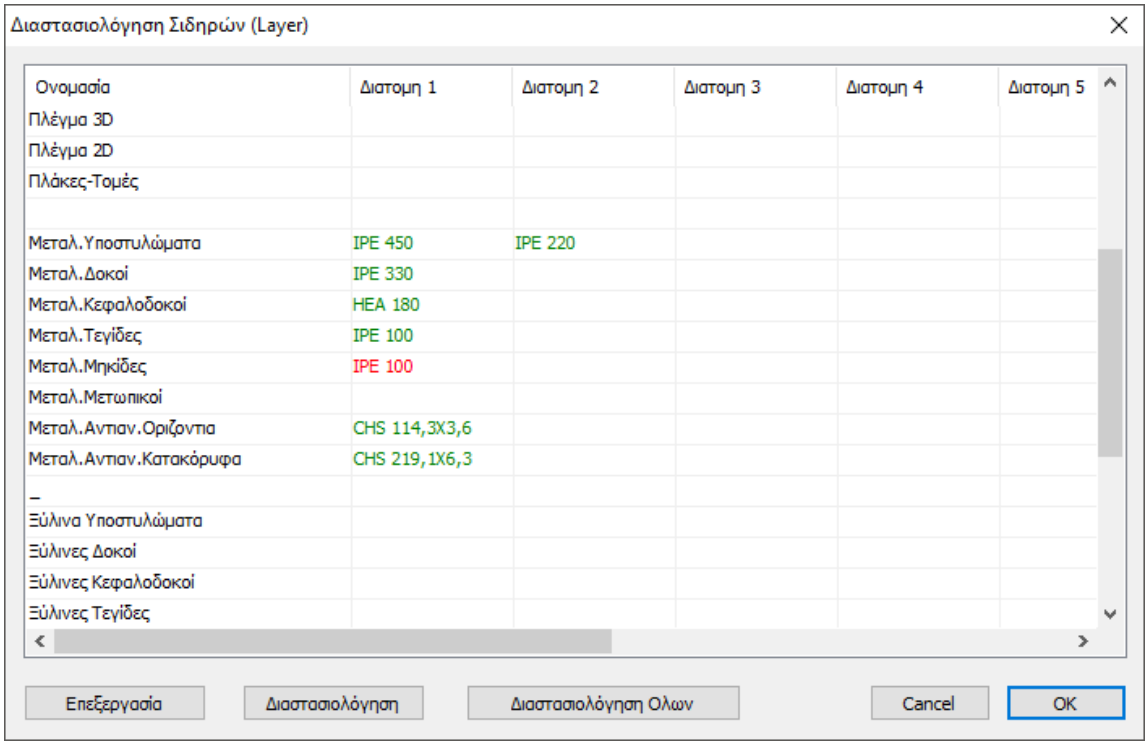


for checking the adequacy of the metallic sections.

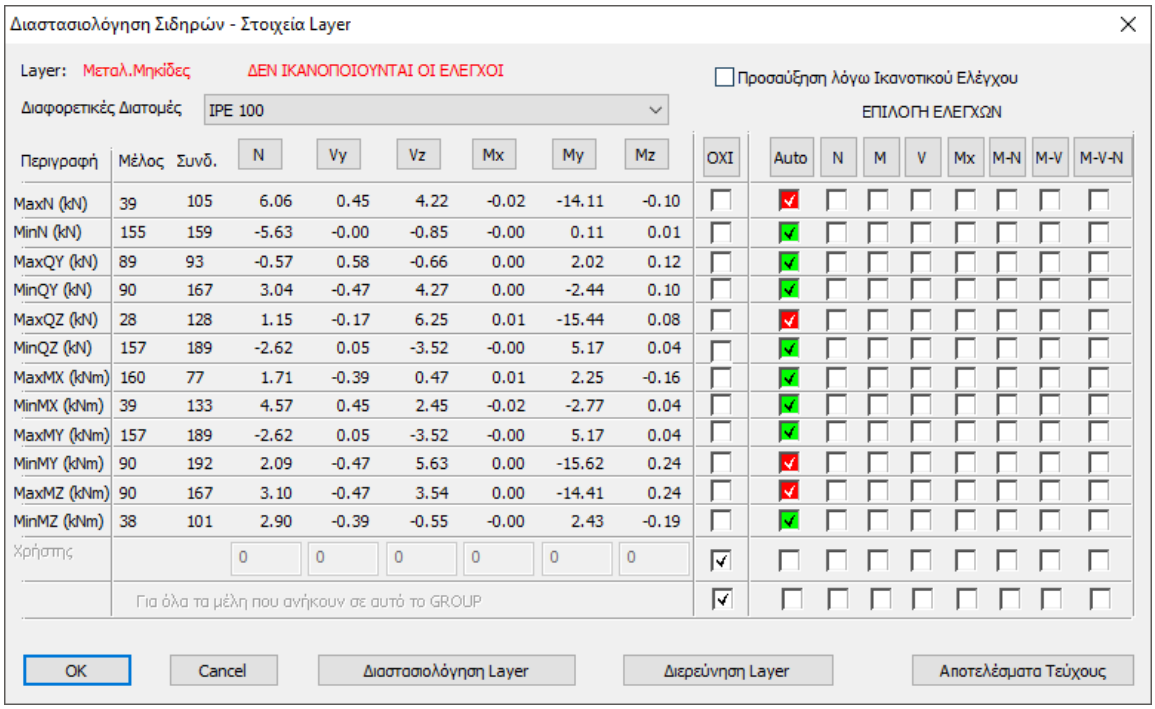
Using the command, the following dialog box appears.



The first column is the layers that exist in this study and in the following columns are the types of metallic cross-sections that exist in these layers.
With the "Dimensioning" option and after you have selected a layer, the dimensioning (the check of the cross sections) of the specific layer is done, the program "colors" the specific layer green if all the elements involved in it do not fail and red if some of them fail.
Alternatively, with the "Dimensioning of all" option, the dimensioning (checking of the cross-sections) of all metallic cross-sections is done.



EXAMPLE:
In the above example, the layer 'Metal Beams' was selected in which different cross-sections in type and dimensions (HEA, CHS, IPE) have been used, of which the IPE 100 cross-section failed. By selecting the 'Edit' button the following dialog box appears:



1.1.1 Procedure for sizing a layer

STEP 1 :

For each layer that the designer has created, e.g. Steel Beams, and for each different cross-section used in this layer, the maximum and minimum values of each of the 175 combinations of external loads and for each of the 6 intensive quantities (**Mx, My, Mz, Qx, Qy, Qz**, 6 columns in the dialogue box) are calculated, as well the values of the other intensive quantities corresponding to this combination (the 12 rows in the above dialogue box).

This results in 2 sextets for each intensive quantity (one for the maximum and one for the minimum value).

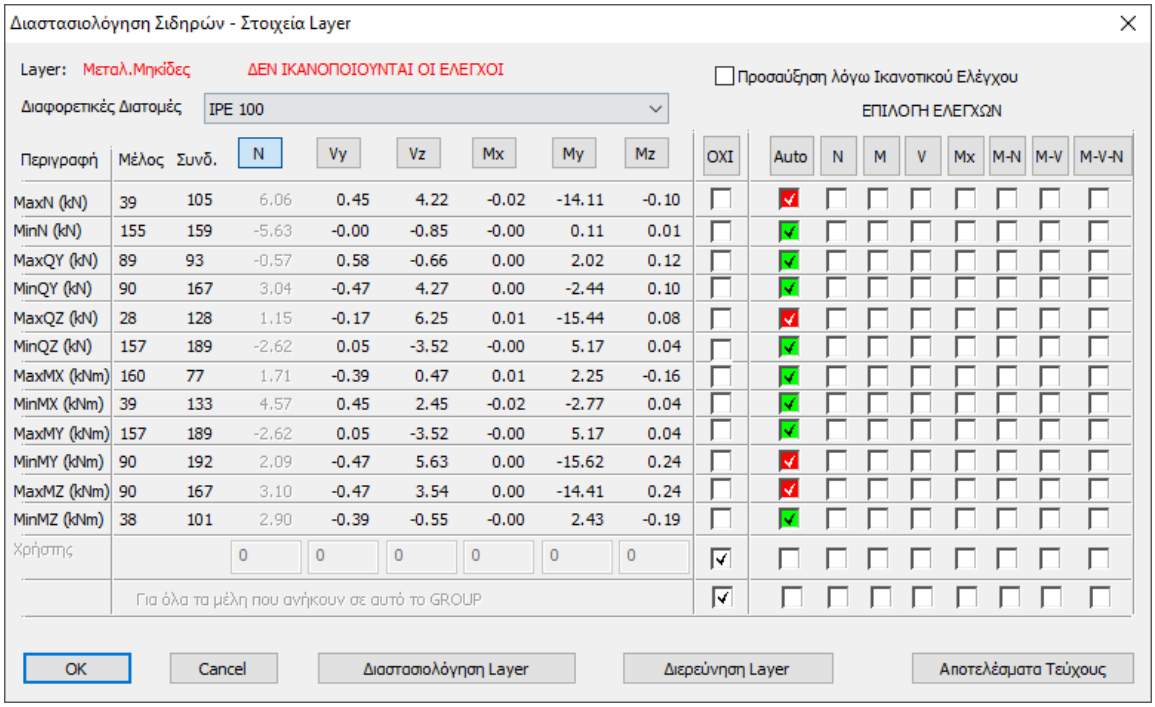
In total for all 6 intensities there will be 2x6=12 hexagons of .

In the above example, for the members of the Steel Beams layer in which the IPE 100 cross-section has been used, the maximum axial force (First Column N and first line Max N) developed in member 39 has a value of 6.06 kN, it was obtained from the combination 39 and the other intensive quantities of this combination are those indicated in the first line. Min N was developed in member 155 from combination 159 and the remaining intensive magnitudes, are those shown in the second line. Similarly for the moment My, the max My was developed in member 157 by combination 189 and the remaining intensive magnitudes of this combination are shown in the corresponding line. The min My was developed in member 90 by combination 192 with the corresponding intensive magnitudes shown in the corresponding line. The checks therefore, where axial N plays a decisive role, will be performed with the sixes obtained from combinations 105 and 159. Similarly, the checks where the decisive role is played by the moment My will be carried out with the above hexagons obtained from combinations 189 and 192. The same applies to the other 5 intensive quantities.

So there are 12 lines

-Max N ...and the corresponding intensive quantities Mx, My, Mz, Qx, Qy
-Min N ...and the corresponding intensive quantities Mx, My, Mz, Qx, Qy
-Max Mx...and the corresponding intensive quantities N, My, Mz, Qx, Qy
-Min Mx...and the corresponding intensive quantities N, My, Mz, Qx, Qy
-Max My...and the corresponding intensive quantities N, Mx, Mz, Qx, Qy
-Min My...and the corresponding intensive quantities N, Mx, Mz, Qx, Qy
-Max Mz ...and the corresponding intensive quantities N, Mx, My, Qx, Qy
-Min Mz ...and the corresponding intensive quantities N, Mx, My, Qx, Qy
-Max Qy ...and the corresponding intensive quantities N, Mx, My, Mz, Qx
-Min Qy ...and the corresponding intensive quantities N, Mx, My, Mz, Qx
-Max Qz ...and the corresponding intensive quantities N, Mx, My, Mz, Qy
-Min Qz ...and the corresponding intensive quantities N, Mx, My, Mz, Qy

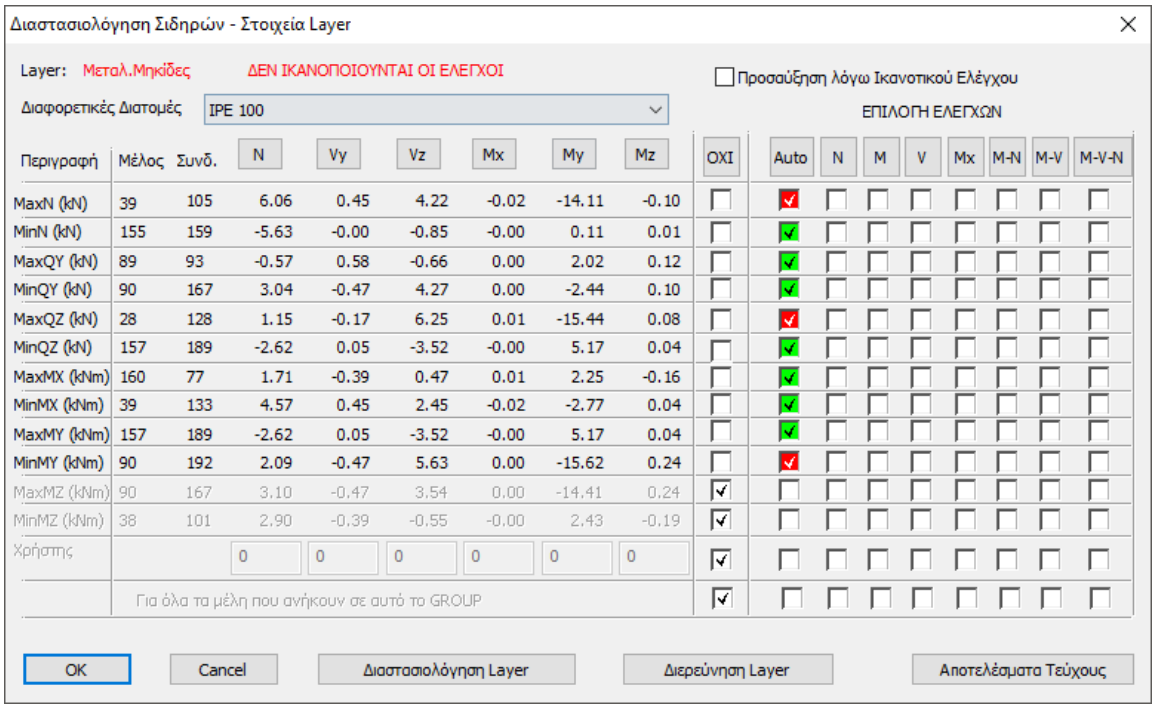
If for some reason you want to completely exclude one or more intensive sizes from the layer sizing, press the corresponding button of the intensive size column. In the following example:



has completely excluded the CT N.

By checking the corresponding options in the "**NO**" column, the program excludes the corresponding minimum or maximum intensive size (the corresponding hex) from the layer checks.

In the example below:



The max Mz and min Mz have been excluded. This means that the program will not check for the two extreme values of Mz. In the other checks, however, Mz is normally included in the respective hexagon.

STEP 2 :

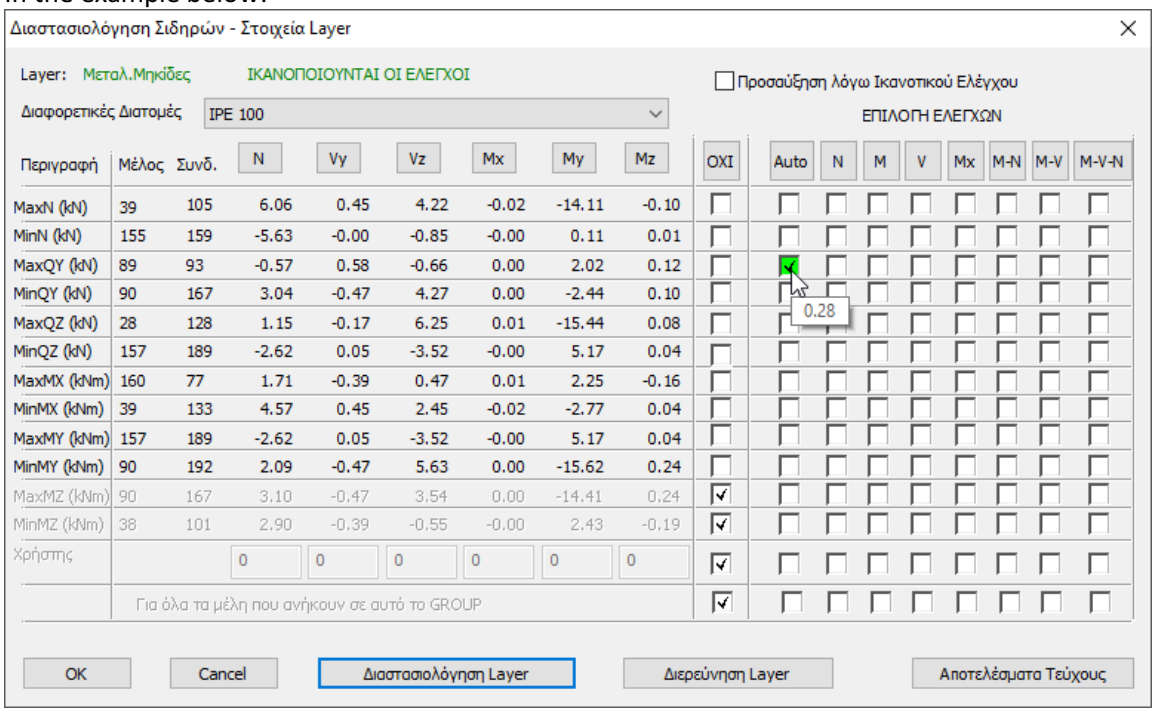
If you select the automatic procedure (column **Auto**) then the program calculates for each series (six) of intensive quantities which check it should make based on the values corresponding to each intensive quantity. So if in a hexadecade there are only **N, My,Mz** values while **Mx=Qy=Qz=0** then the program will perform the checks for Bending, Axial Bending, Compression & Tensile (it will not perform the check against torsion, shear etc.).

So the issue will print the 12 worst reasons, one for each intensive size. (12 rows of 6 reasons each).

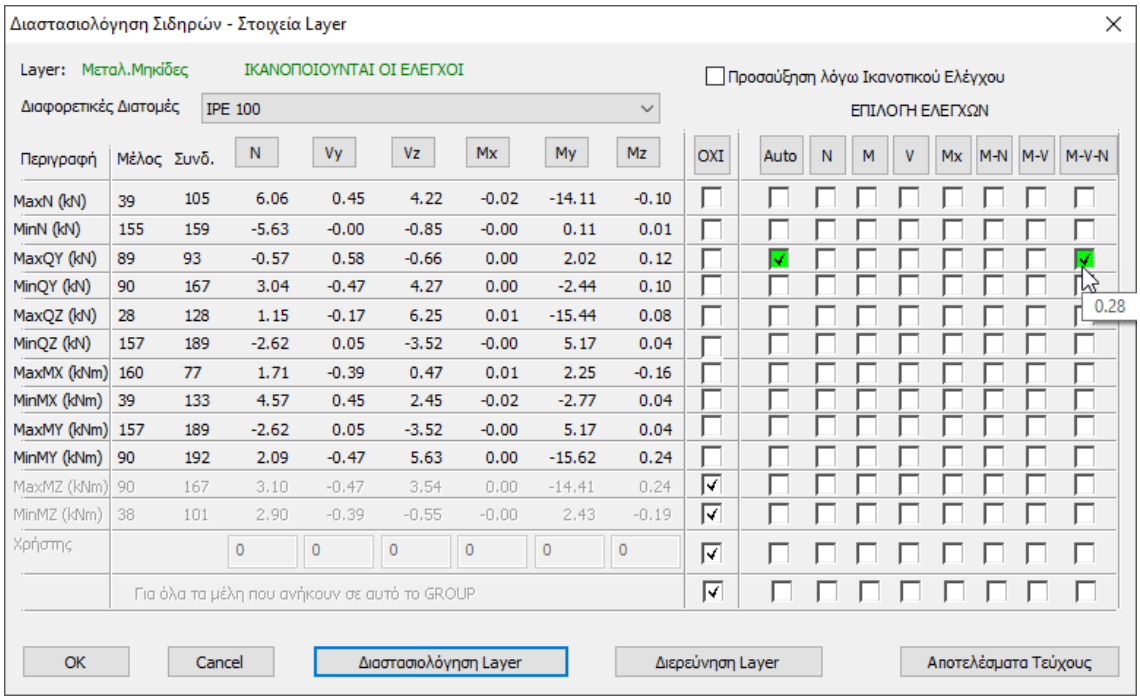
At this point the following should be emphasised:

If you select the **Auto** column, the program performs the corresponding checks based on the intensities present in the corresponding hex. Thus, the individual ratio of the corresponding intensive size participates in the resulting strength ratio with its sign, which means that some sizes may have a "relieving" effect, so that the total ratio is smaller than if you had manually performed an individual check, e.g. only in axial.

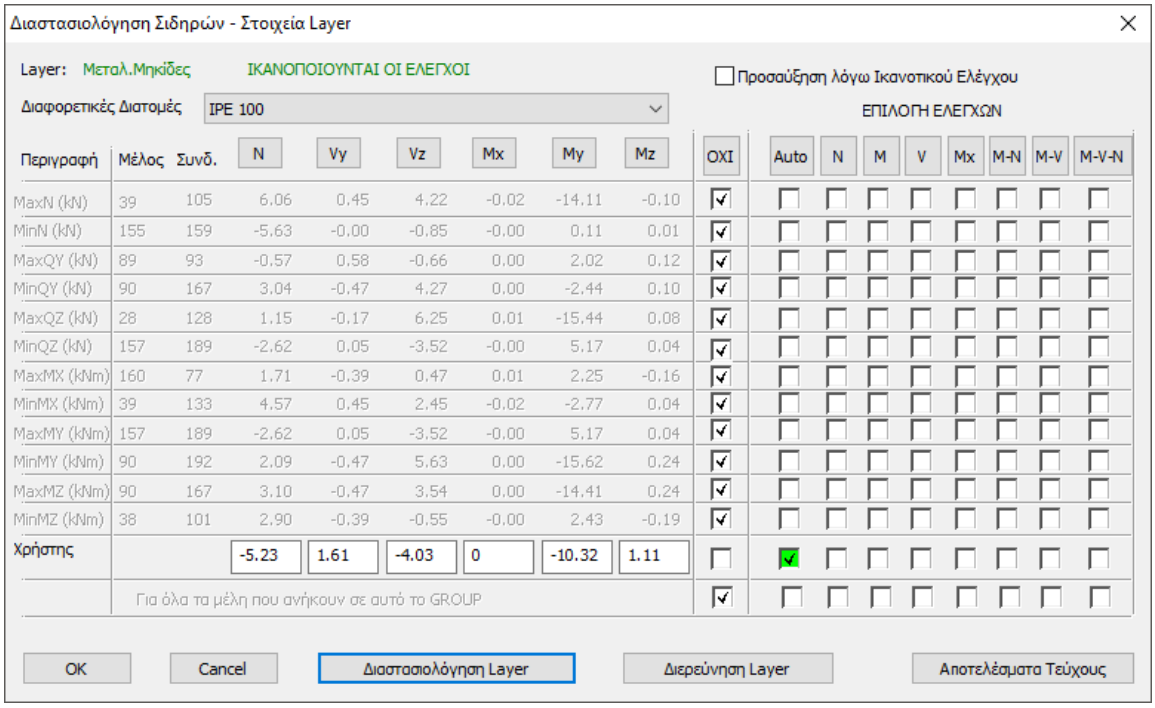
In the example below:



has been selected to check only for max Qy. The automatic check gave a strength ratio 0.28 (holding the mouse arrow over the check box, displays the ratio). In this particular hexadecade of intensive sizes, all intensive sizes (except Mx) are present, and the program did the check by taking into account the individual strength ratios from all intensive sizes. The result is the same as if you had manually checked the "M-V-N" column.



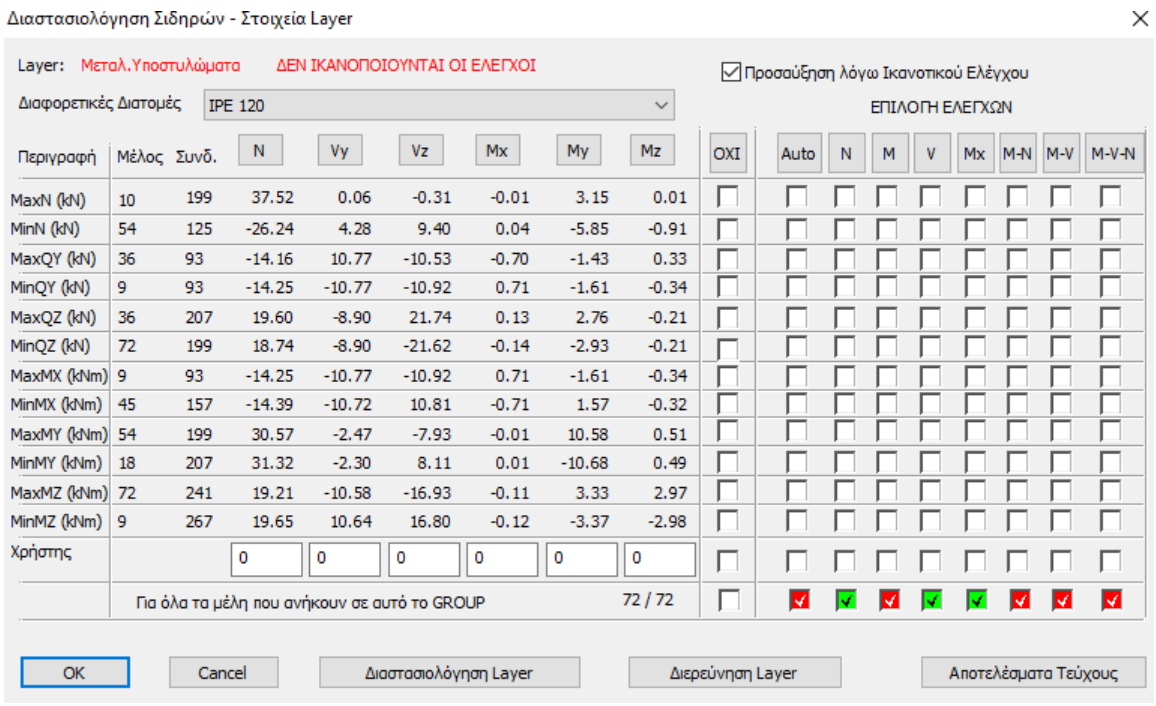
If you choose the manual procedure, you have the option check which checks, for each hex, you want to perform. This will print in the issue for each hexadecade the reasons for the respective checks selected.
Finally, in the "User" option you can set your own intensive sizes, in order for the program to dimension the specific cross-section. In the next dialog box:



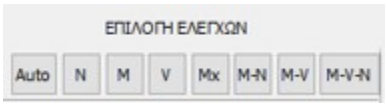
specific intensities have been given by the designer and the intensities calculated by the program from the analysis have been deactivated.

OBSERVATION:

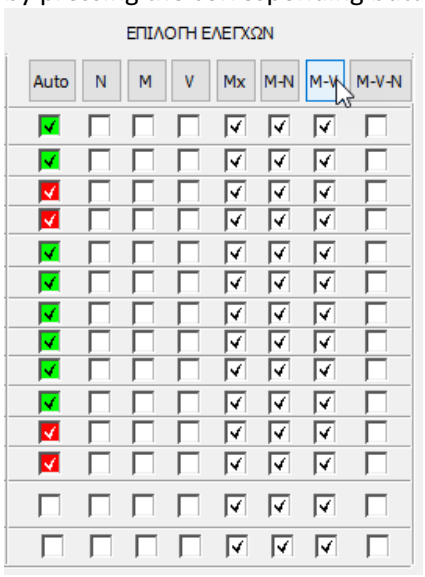
In the new versions of SCADA Pro a new feature was added that was activated in the cross-section control of metallic cross-sections. Recall that the cross-section control was previously done at the layer level and at the level of each cross-section but only the worst member for each intensive size was controlled.
Now the check is still done at layer level and for each cross-section but now each member is checked by activating the option at the bottom as shown in the next picture.



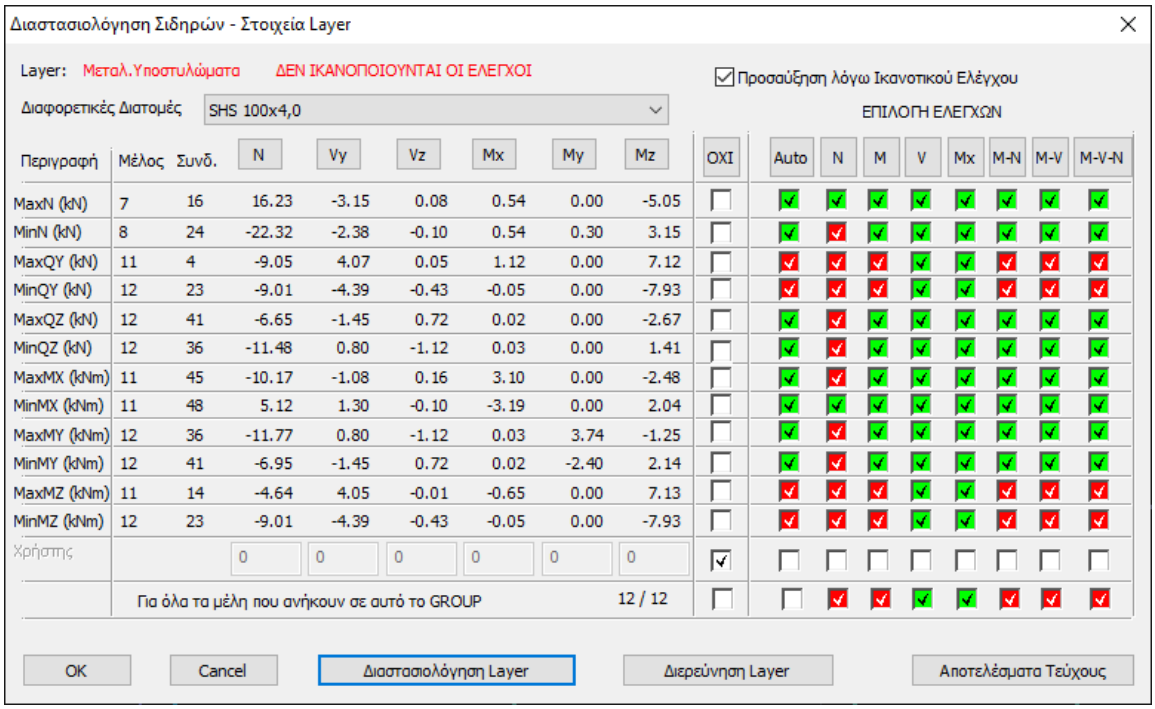
Select one or more controls from the list



by pressing the corresponding button



Then select *Layer Sizing*, For all members belonging to this GROUP



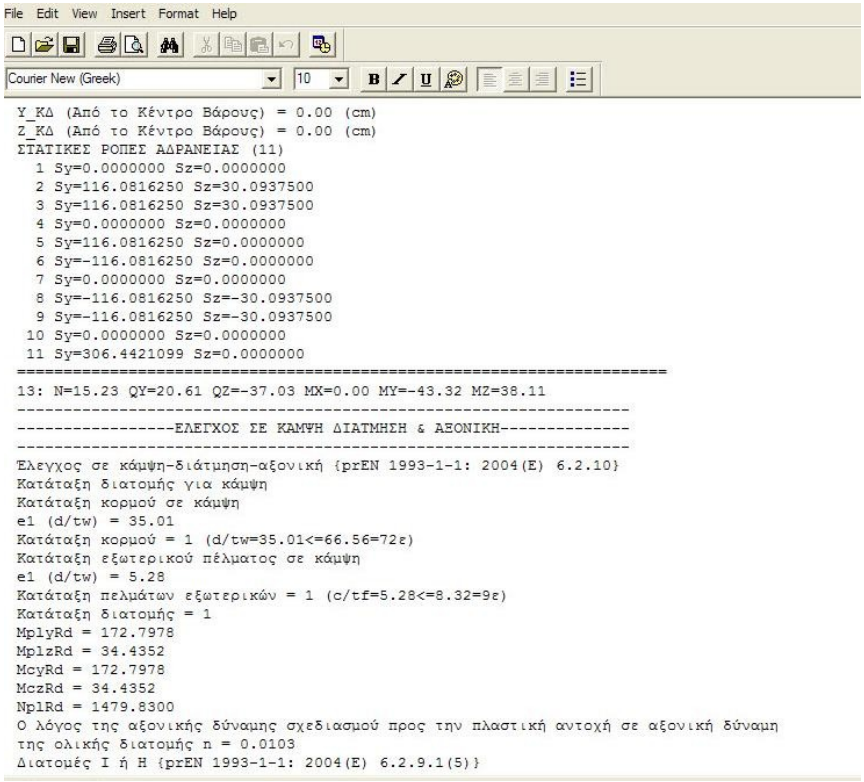
In this example all the checks (General (Auto) and individual) have been done.

ATTENTION:

The convention followed for the sign of the axial force is as follows: In SCADA Pro intensive axial force magnitude:

- with a **negative** sign means **Tensile** and
- with a **positive** sign means **Sadness**.

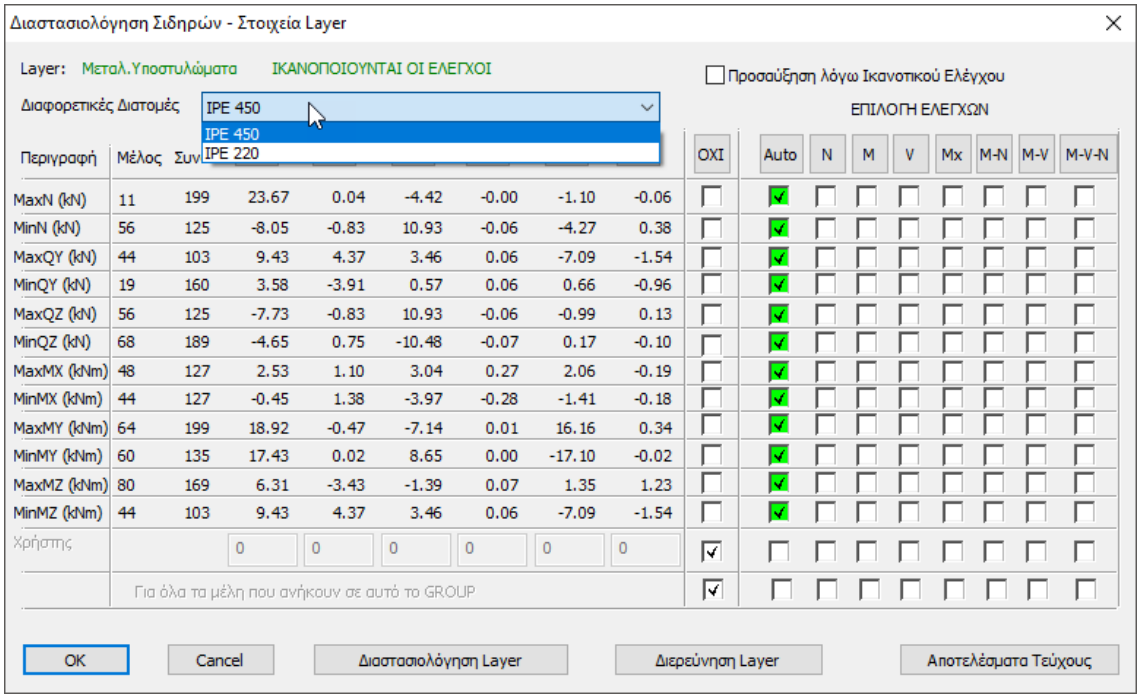
But in the investigation and in the results issue



the axial is shown with a positive sign. Of course it is still tensile but here the classical convention is followed:

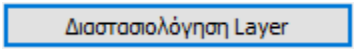
- (+) Tensile
- (-) Sadness

In the section "Different Cross-sections"



shows the different cross-sections included in the "Metal Beams" layer.
The same procedures described above can be followed in order to manually dimension the remaining cross-sections or see the results booklet and the investigation.

- 

Activate the Capability Check surcharge and then select the command 

, to take into account the surcharges foreseen by the Eurocode for the Capability Check.

1.2 Bending control (Hot rolled)

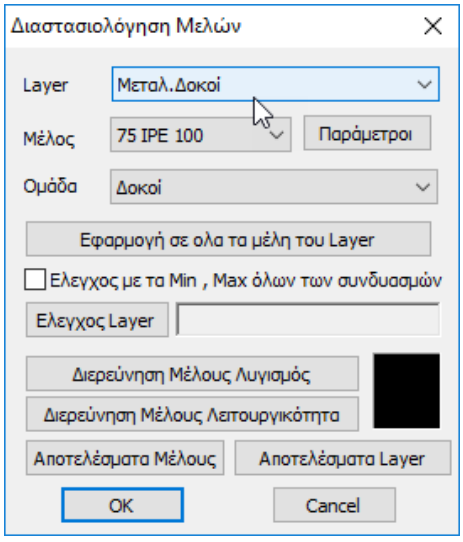
The use of this command is used to check for bending. That is, the checks are performed for each member belonging to the given layer:

Limit State of Failure
- Check for bending (lateral) buckling due to axial compressive force
- Torsional buckling check due to bending moment.
- Check for torsional bending due to the simultaneous presence of axial compressive force and bending moment.

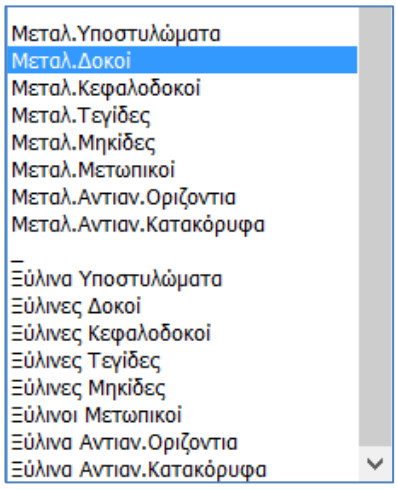
Limit State of Functionality
- Member deformation control
- Edge (node) movement control

OBSERVATION:
A prerequisite sizing is that you called and executed the corresponding combination file in the configuration dialog box

Using the command, the following dialog box appears.

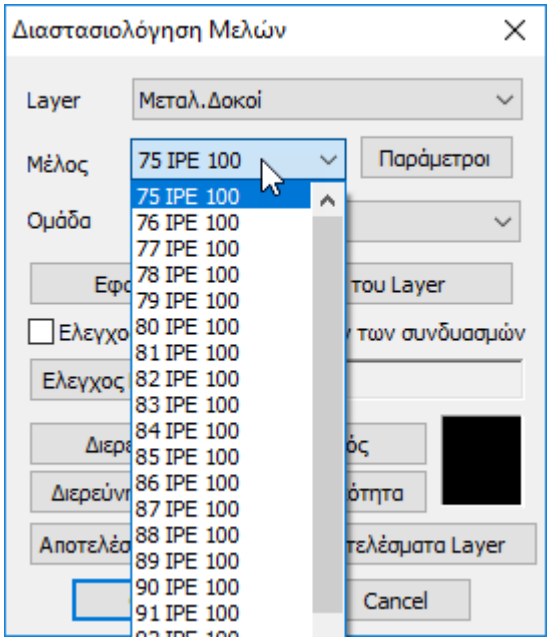


The check is done per layer. So first select from the list



the layer (e.g. Metal Beams) you want to dimension.

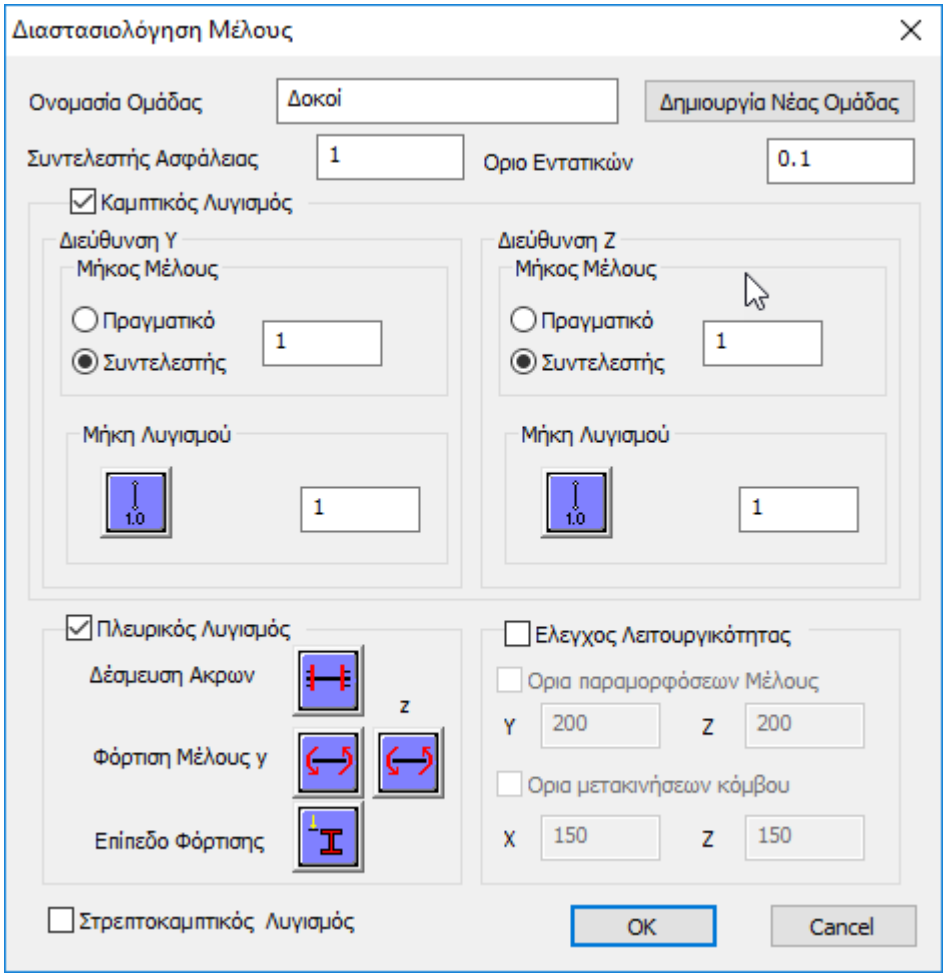
By selecting the layer, all the members of this layer and their cross-section are displayed in the "Member" list.



The first step in dimensioning the layer is to define the dimensioning parameters. Because it is possible that for some of the layer members you may want to define different parameters, it is possible that within the same layer you can define different groups of parameters to which the layer members belong.
The program has two predefined groups of parameters: 'Beams' and 'Columns'.

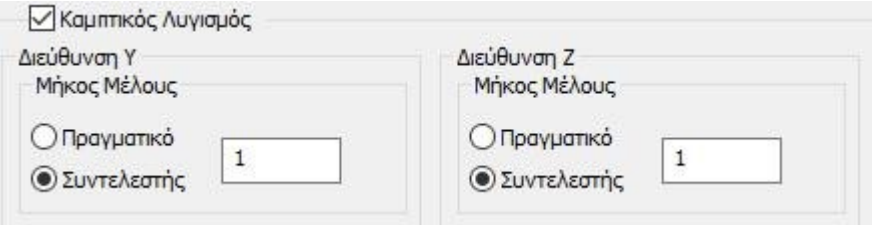
If you want to have the same parameters for all members of the layer, you set them once with the procedure we will see below, keep the default name "Beams" and press the "Apply to all members of the layer" button.
The checks will be done with the same parameters for all members of the layer.
In the different case that you want to set different parameters for some of members of the layer, you will set another set of parameters using the procedure explained below.
But first we will look at how the parameters are defined.

Selecting the "Parameters" button will display the following dialog box:



The "Group Name" field contains the name of the parameter group. If you want to create your own group, enter a new name and press the "Create New Group" button.
In the "Safety factor" field you can set the threshold against which the program checks the ratio of the design value (the intensive value) to the corresponding strength of the member. The default value is 1. In the field "Intensity limit" there is the limit of the intensive sizes below which the program does not take the intensive sizes into account.

The rest of the dialog box is divided into three parts, each of which deals with the parameters of the **Flexural Bending**, **Lateral Bending** and **Functionality Checks**.
In the **Flexural Bending** section you first define whether you want to check the flexural bending by ticking the corresponding option.



OBSERVATION

- In earlier versions of SCADA Pro and prior to the creation of the

command, the user was prompted to set the member length and the buckling length in both the Y and Z directions respectively, following the procedure below:

In "**Length of Membership**":

- if you select "*Actual*" you must enter in the field the actual length of the member in m.
- if you select "*Coefficient*" you must enter a coefficient by which the different lengths of the members belonging to this parameter group will be multiplied.

If you want the program to take the actual lengths of the members into account when checking the bending bend, select "Coefficient" with a value of 1.
If you have some members with different or equal lengths that are laterally secured at the same distances (e.g. at 1/3), then you give the value 0.33 and of course create a separate parameter group to which these members belong.

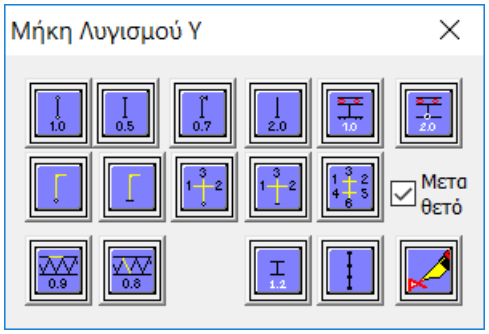
In the new versions of SCADA Pro the definition of the bend length is done through the "**Consolidate Members**" command (see Sizing-General) and no action is required in this field. Having therefore followed the procedure of Member Consolidation, in the Parameter field, specifically Member Length, leave as is and proceed with the definition of the remaining parameters.

The next parameter is the member's **buckling length** which depends on the support conditions of the nodes of the member ends always within the buckling plane.

OBSERVATION

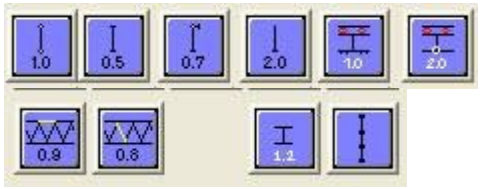
If there has been a Consolidation, then the Bending Length refers to the Consolidated Member.

Pressing the  button displays the following dialog box

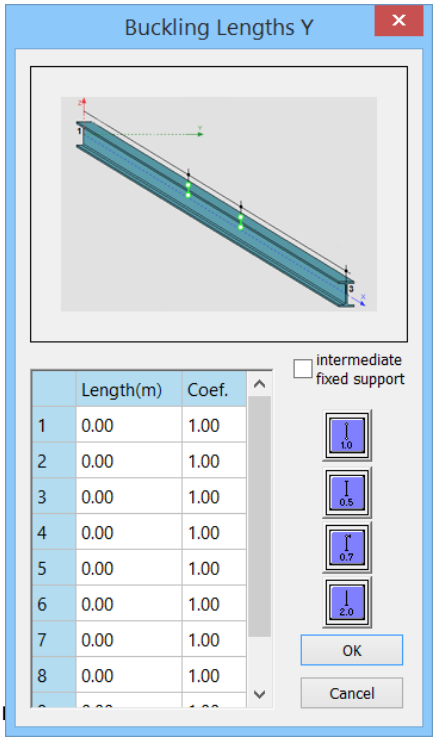


where you select the icon with the support conditions of the member and the program enters the corresponding coefficient for the buckling length.
The icons are divided into two groups:

1) The first group includes the icons with a specific coefficient depending on the support conditions of the member



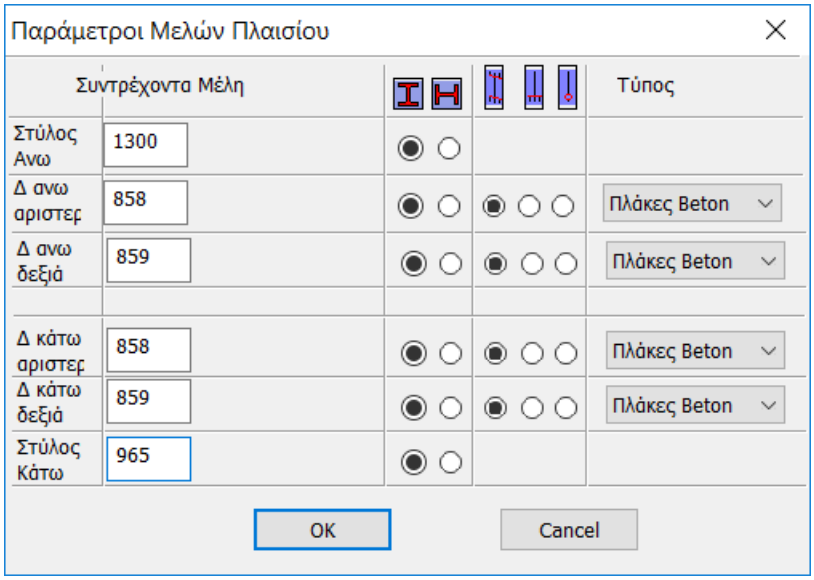
By selecting the icon  you can set the positions of the lateral securing, if any, for the specific member so that the corresponding reduced bending lengths are obtained. This feature will be enabled in a later version of the program.



2) The second group

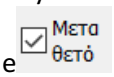
includes the member cases in multi-storey steel structure fra
concurrent members in the node.

By selecting the  icon (the most complex case) you define for the vertical member the 6 members (2 vertical and 4 horizontal) that are present in it (3 at the beginning and 3 at end). Selecting the icon displays the following dialog box:

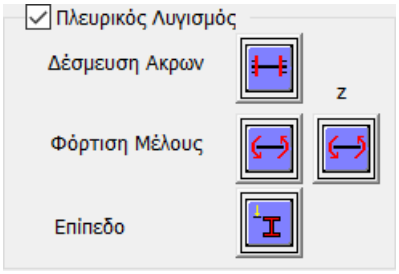


where for the respective fields you DRAW graphically with the mouse the respective members that occur at the start and end nodes of the member you are specifying the bending length.

Every time you select a member with the mouse, number, cross-section and length are automatically displayed in the corresponding field. You must indicate the concurrent members by the title (Pillar Upper, D upper left, etc.) indicated in the corresponding row. Once you have completed the process of defining the members, you must define for these members their orientation and especially for the beams the type of support at their other end and the type of load imposed on them. By pressing "OK" button, in the buckling length the corresponding icon and the coefficient -1 is displayed, which generally means that the program, based on the data you have provided, automatically calculates the buckling length for the given member.

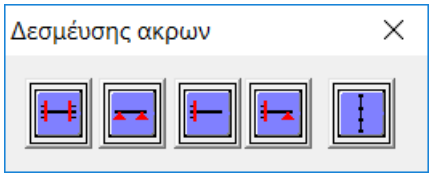
Finally, with the  option you have the possibility to enter your own value in the corresponding field of the bending length, while with the 

option you define whether the frame to which the member belongs is transposable or not transposable.

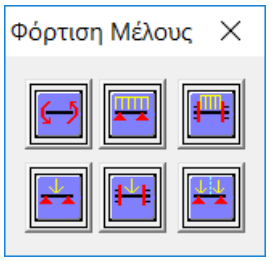


In the **Side Bending** section you first define whether you want to check the side bending by checking the corresponding option.

You then set the type of binding of the element's edges by selecting the appropriate icon. You choose from three types of supports Amphipact, Amphibious and Protrusive.

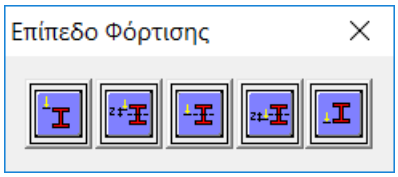


The edge binding formula is used to calculate the lateral buckling coefficient.
The next parameter concerns the type of loading of the member along its local y and z axis respectively.
By selecting the corresponding icon, the following options are displayed



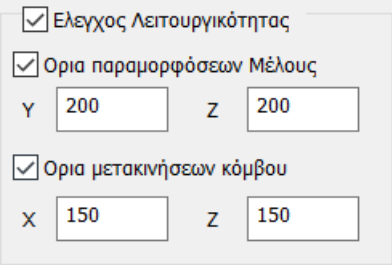
Where you select the corresponding type of Charge.

Finally, the last parameter concerns the determination of the loading level of the member. By selecting the icon, the following 5 options are displayed.



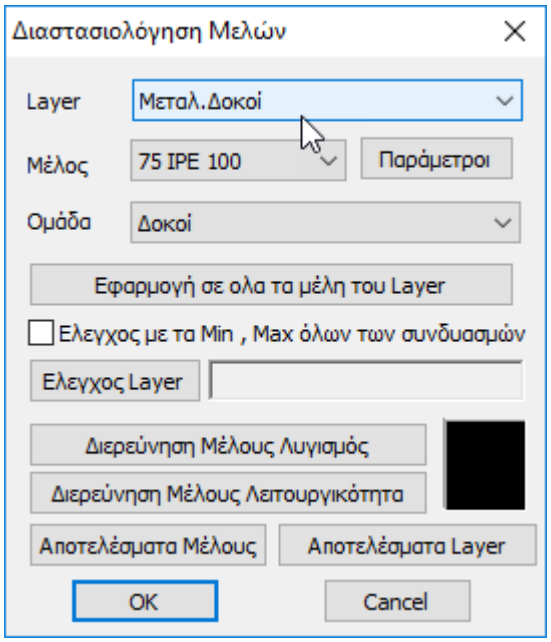
Το πρώτο εικονίδιο αφορά επίπεδο φόρτισης στο άνω πέλμα του στοιχείου, το δεύτερο αφορά επίπεδο φόρτισης κοντά και προς τα πάνω από τον άξονα συμμετρίας του στοιχείου, το τρίτο αφορά επίπεδο φόρτισης στον άξονα συμμετρίας του στοιχείου, το τέταρτο αφορά επίπεδο

loading level near and below the symmetry axis of the element and finally the fifth option is a loading level at the lower foot of the element.
3) The third set of parameters concerns the parameters of **functionality**



where you specify whether you want to perform the functionality check, the individual Member Deformation and Node Displacement checks, and the respective upper limits (l/220 and l/150 where l is the length of the element) for these checks.

Finally, check the "**Torsion bending**" option if you want to do this check.
When the configuration settings are complete, press the "OK" button and return to the previous dialog box



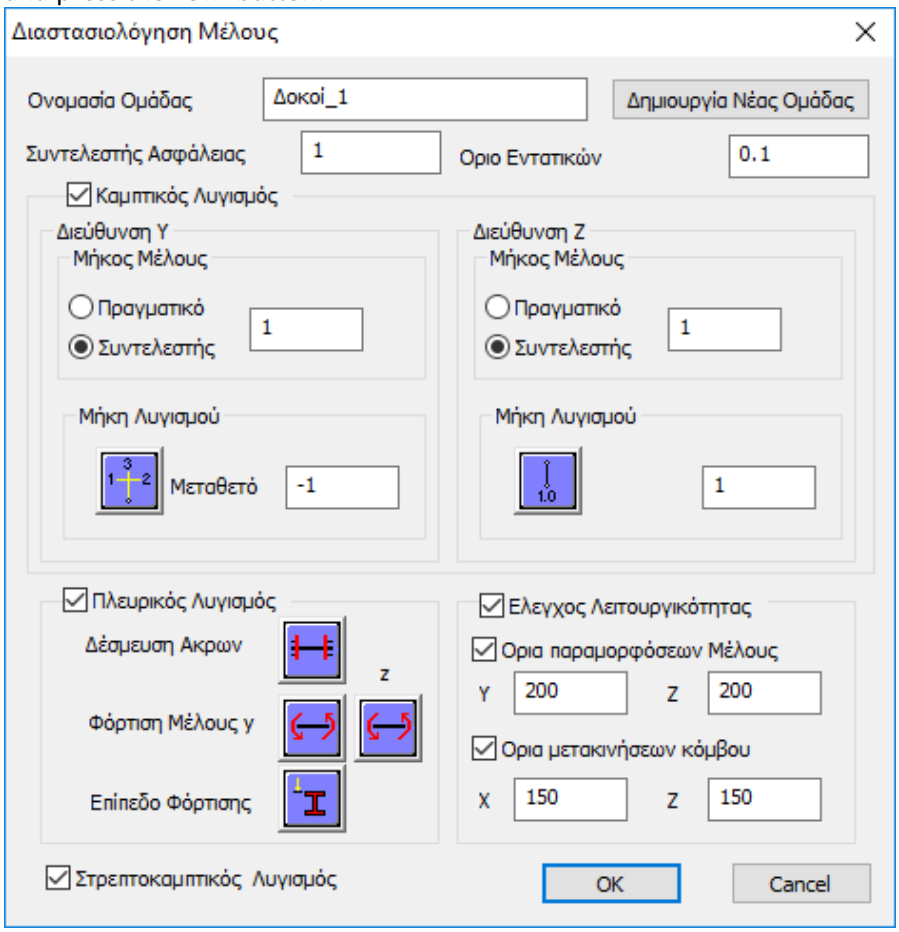
By pressing the "Apply to all members of the Layer" button, the program applies the parameter group you just defined with the default name "Beams" to all members of the "Metal Beams" Layer you had selected. Then you press the "Check Layer" button and the program starts the process of executing the layer for the specified layer "Metal Beams". Beams".


By activating the option 

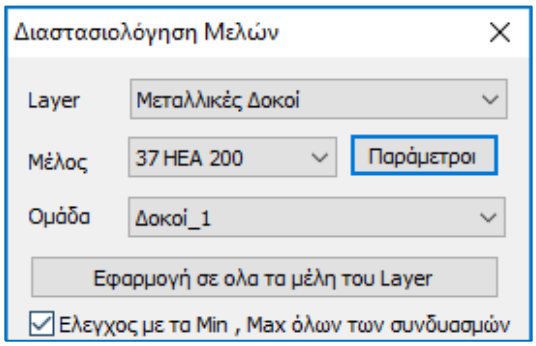
, the check will be performed taking into account only the maximum and minimum values of the intensities obtained from all combinations, excluding intermediate values, so that the process is completed in significantly shorter times.


EXAMPLE:

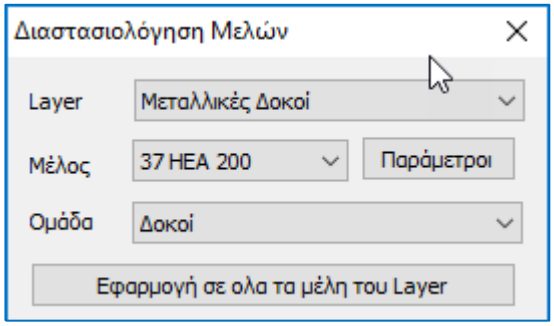
If you would now like to specify another group of parameters to which some of the members of the layer belong, follow the procedure below:
Press the "Parameters" button and open the configuration dialog box again. In the "Group Name" field, enter a name for the new parameter group you will create e.g. "Beams_1" and press the "Create New Group" button. Then you set the parameters based on what was mentioned before and press the "OK" button.



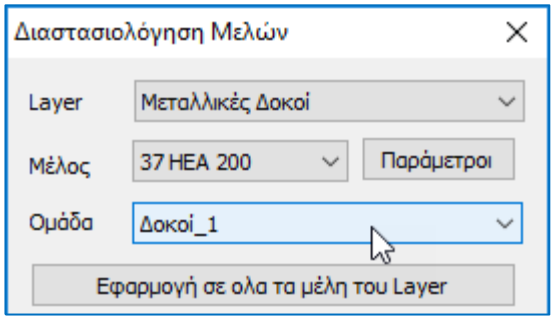
The next step is to define which members from the layer will belong to this group of "Structures_1" parameters.
Returning to the original dialog box, the only member that gets the parameters automatically is the current member in the list of members



That is the 37 HEA 200 member. All other members have the parameters of the "Beams" group. To change the members you want from one group to another, select them one by one from the list and from the list of the "Group" section select the "Beams_1" group.
For example, you select from the list the member 37 HEA 200

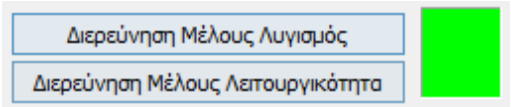


and shows you that he already belongs to the "Beams" group. Open the list of groups and select the group "Beams_1"



Now member 37 HEA 200 belongs to the group "Doctors_1". Follow the same procedure for the other members that you want to change their parameter group.

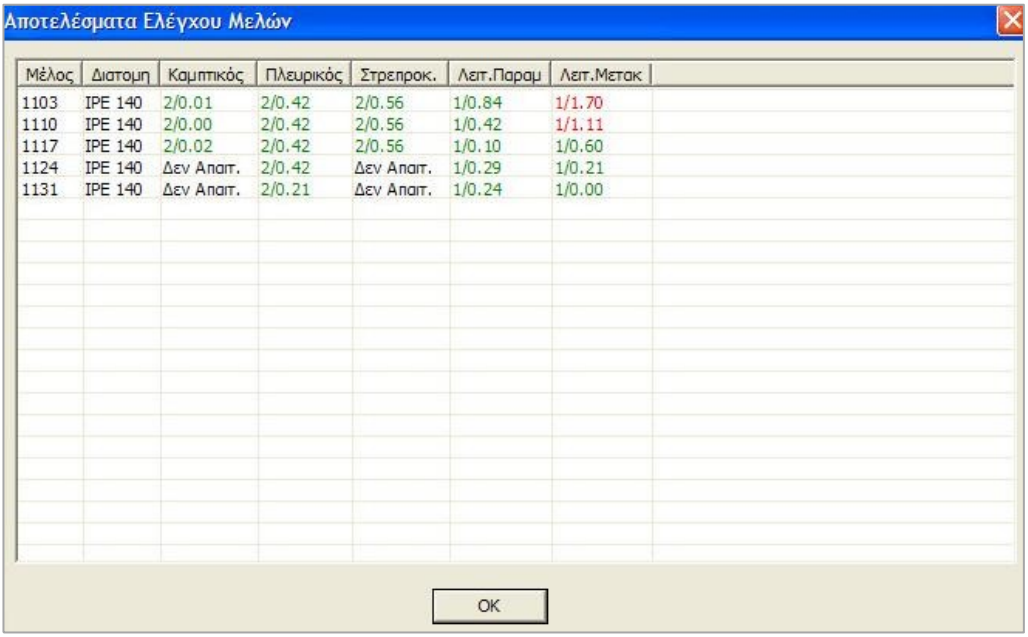
Upon completion of the testing process for the specific layer the icon next to the "Investigate Bending Member" and "Investigate Functionality Member" buttons



is coloured in a corresponding colour:

- Red if there is a failure and
- Green if there is none.

By double-clicking on this coloured icon, the following dialogue box appears



with the summary results of the audit of the members.

1: In the first column is the number of the member, 2: In
the second column is the cross-section, and
3-7: In the next 5 columns the worst strength ratio and the number of the combination from which this ratio was derived.

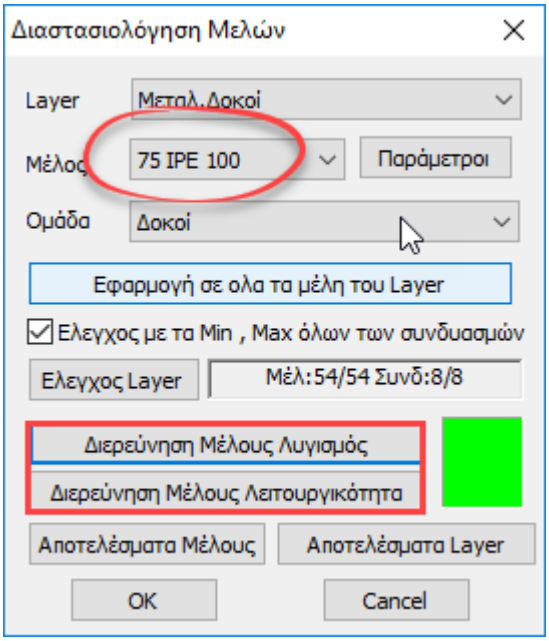
Green are ratios below the unit and red are ratios above the unit. Where the phrase 'not required' is shown, it means that the corresponding intensive magnitude was not available or that the axial force was tensile and not compressive.

By selecting the button "Layer Bending" the program displays the summary results of the check in Bending (i.e. for each member the results from the worst combination) while by selecting the button "Investigation Layer Bending" the program displays a complete but very large file with the results of the checks for each member from all combinations. Similar applies to the "Functionality Issue" and "Investigate Layer Functionality" buttons.

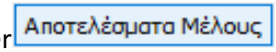
The check for the three types of buckling is carried out for each member and for all combinations. However, for each combination, i.e. for each triad N, My and Mz the checks are carried out 4 times based on the following combinations:

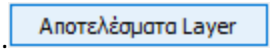
- N with min My and min Mz
- N with min My and max Mz
- N with max My and Min Mz
- N with max My and max Mz

That is why both in the results of the issue and in the investigation, two numbers are mentioned in the combination number: The first refers to the number of the combination and the second refers to the number for each of the 4 previous cases.



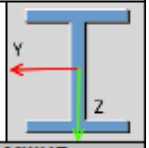
Choosing the Member Investigation (Bending / Functionality) open files containing the detailed results of all checks for all combinations for the active member

Selecting Results opens the files containing the summary results of the checks for the active member 

and for all members of the active layer . 

Σελίδα : 1

ΔΙΑΣΤΑΣΙΟΛΟΓΗΣΗ ΣΙΔΗΡΩΝ ΚΑΤΑΣΚΕΥΩΝ - ΕΛΕΓΧΟΣ ΜΕΛΩΝ

Layer :	Μεταλ.Δοκοί			75 76 77 78 79 80 81 82 83	
Μέλος	75	IPE 100			
Κόμβος Αρχής	716	Κόμβος Τέλους	767		
Ενοποίηση y-y		Ναι	316.23 cm		
Ενοποίηση z-z			cm		

ΚΑΜΠΤΙΚΟΣ ΛΥΓΙΣΜΟΣ

Τελική Κατάταξη διατομής σε θλίψη:				1			
Κορμού	1	Πελμάτων		0	1		
Συνδυασμός		1 / 1					
Εντατικά Μεγέθη		N (kN) =	-4.70				
My(kNm) =	-0.26	Mz(kNm)=	0.06				

Μέγεθος	Τιμή y-y	Τιμή z-z	Μονάδες
Lcr	316.23	30.00	cm
Καμπύλη	a	b	
Συντ.ατελειών α	0.210	0.340	
lamda1	93.900		
lamda*	77.695	24.159	
lamdaT	0.827	0.257	
NEd	4.70		kN
Ncr	354	3666	kN
NEd/Ncr**	0.01325	0.00128	
X	1.000	0.980	
Nb,Rd	189.142	237.646	kN
NEd/Nb,Rd	**0.025**	**0.020**	
ΕΠΑΡΚΕΙΑ	**Ναι**	**Ναι**	

ΠΛΕΥΡΙΚΟΣ ΛΥΓΙΣΜΟΣ ΛΟΓΩ ΚΑΜΨΗΣ (ΣΤΡΕΠΤΟΚΑΜΠΤΙΚΟΣ ΛΥΓΙΣΜΟΣ)

Διεύθυνση y-y	Συνδυασμός:	1 / 3	
Εντατικά Μεγέθη	N (kN) =	-4.70	
My(kNm)=	-0.48	Mz(kNm)=	0.06

Μέγεθος	Τιμή	Μονάδες
Lcr,y	316.23	cm
Συντελεστής K	0.50	
Συντελεστής c1 (Mcr)	1.48	
Συντελεστής c2 (Mcr)	0.00	
Συντελεστής c3 (Mcr)	2.27	
zg (Mcr)	5.00	cm
Mcr	538	kNm
lamdaLT_bar***	0.131	
FLT	0.501	
XLT	1.000	
MyED	-0.478	
MyED/Mcr****	-0.001	
MbRd	9.261	kNm
MyED/MbRd	0.052	
ΕΠΑΡΚΕΙΑ	**Ναι**	

ΛΥΓΙΣΜΟΣ ΛΟΓΩ ΚΑΜΨΗΣ ΚΑΙ ΑΞΟΝΙΚΗΣ ΘΛΙΨΗΣ

Τελική Κατάταξη διατομής				1	Συνδυασμός:	41 / 3	N (kN) = -3.87
Κορμού	1	Πελμάτων	0	1	My(kNm) =	-0.37	Mz(kNm)= 0.14

Μέγεθος	Τιμή y-y	Τιμή z-z	Μονάδες
Lcr	316.23	30.00	cm
Καμπύλη λυγισμού	a	b	
Συντελεστής ατελειών α	0.210	0.340	
lamda1	93.900		
lamda	77.695	24.159	
lamdaT	0.827	0.257	
X	0.780	0.980	
Ratio (1) Εξ.6.61	0.081		kN
Ratio (2) Εξ.6.62	0.081		kN
ΕΠΑΡΚΕΙΑ	**Ναι**		

Υπολογισμός Mcr		
Μέγεθος	Τιμή	Μονάδες
Συντελεστής K	0.500	
Συντελεστής c1	1.472	
Συντελεστής c2	0.000	
Συντελεστής c3	2.273	
zg	5.000	cm
Mcr	537	kNm
lamdaLT_bar***	0.131	
FLT	0.501	
XLT	1.000	

*Αν lamda <= 0.2 ο έλεγχος αγνοείται

**Αν NEd/Ncr <= 0.04 ο έλεγχος αγνοείται

***Αν lamdaLT_bar <= 0.2 ο έλεγχος αγνοείται

****Αν MyED/Mcr <= 0.04 ο έλεγχος αγνοείται

ΠΑΡΑΜΟΡΦΩΣΕΙΣ ΜΕΛΩΝ (ΤΟΠΙΚΟΙ ΑΞΟΝΕΣ)

Μέγεθος	Τιμή y-y	Τιμή z-z	Μονάδες	Συνδυασμός
δmax	0.003	0.001	cm	99
Lcr	316.23	316.23	cm	
Συντελεστής	150.00	150.00		
Ratio	0.002	0.000		
ΕΠΑΡΚΕΙΑ	**Ναι**	**Ναι**		

ΜΕΤΑΚΙΝΗΣΕΙΣ ΚΟΜΒΩΝ (ΚΑΘΟΛΙΚΟΙ ΑΞΟΝΕΣ)

Μέγεθος	Τιμή y-y	Τιμή z-z	Μονάδες	Συνδυασμός
Δumax	0.001	0.000	cm	99
Lcr	316.23	316.23	cm	
Συντελεστής	150.00	150.00		
Ratio	0.000	0.000		
ΕΠΑΡΚΕΙΑ	**Ναι**	**Ναι**		

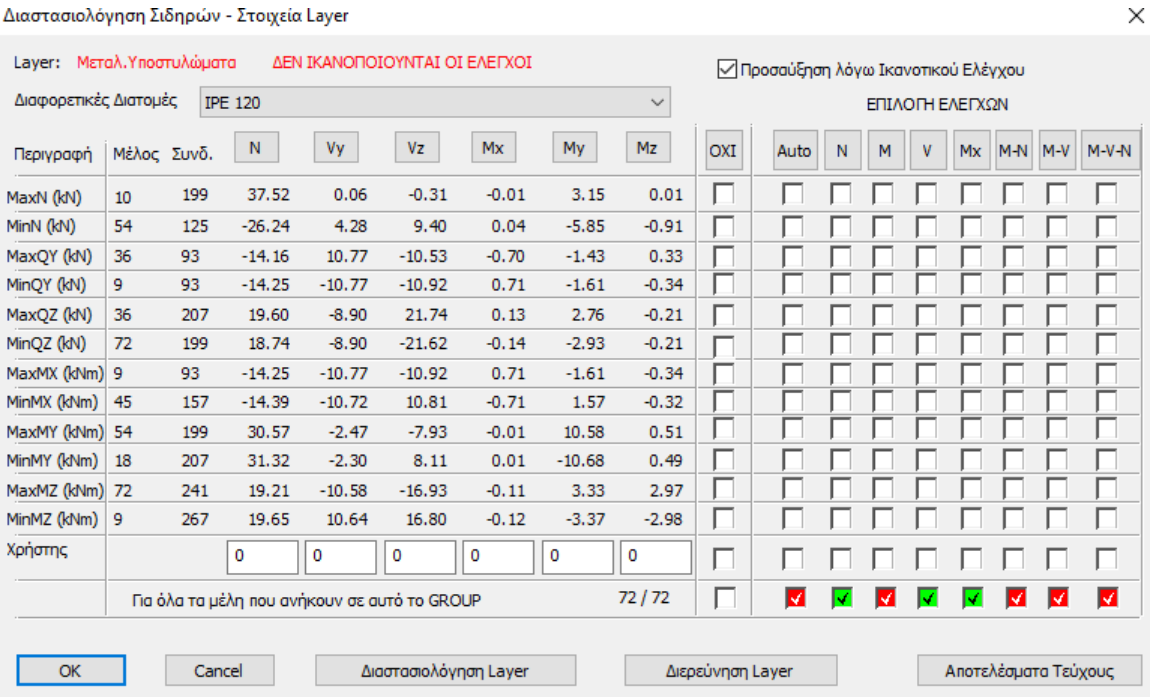
Show reasons for exhaustion with Color Gradation

In the case of steel structures the checks are common for beams and columns and represent a single value. More specifically, the following depletion ratios are calculated. :

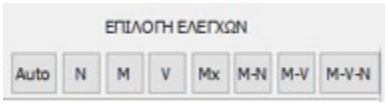
1) Reasons for Auto Exhaustion
2) Reasons for depletion N
3) Reasons for exhaustion M
4) Reasons for exhaustion V
5) Reasons for Mx exhaustion
6) Reasons for M-N depletion
7) Reasons for M-V exhaustion
8) Reasons for exhaustion M-V-N
9) Reasons for Flexural Bending Exhaustion
10) Lateral Flexion Exhaustion Reasons
11) Reasons for Torsion Flexure Exhaustion
12) Reasons for deformation exhaustion
13) Reasons for Travel Exhaustion

IT IS RECALLED THAT:

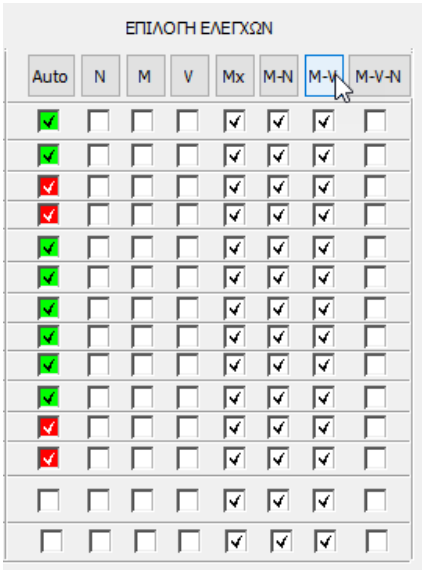
In the new versions of SCADA Pro a new feature was added that was activated in the cross-section control of metallic cross-sections. Recall that the cross-section control was previously done at the layer level and at the level of each cross-section but only the worst member each intensive size was controlled. Now now the check is still done at layer level and for each cross-section but now each member is checked by activating the option at the bottom as shown in the next figure.



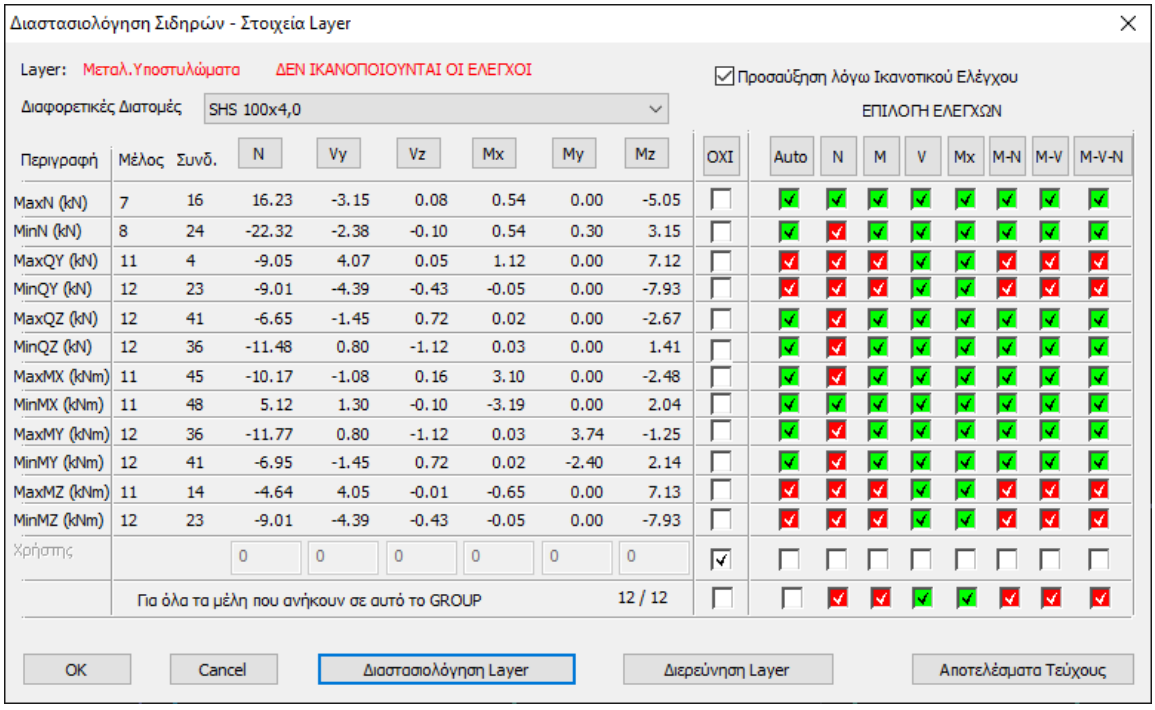
Select one or more controls from the list



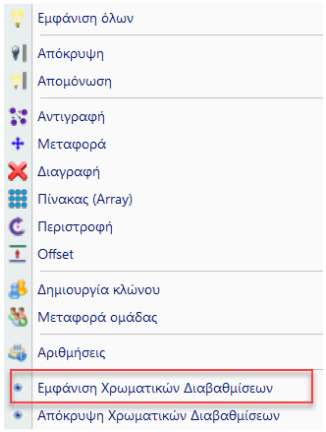
by pressing the corresponding button



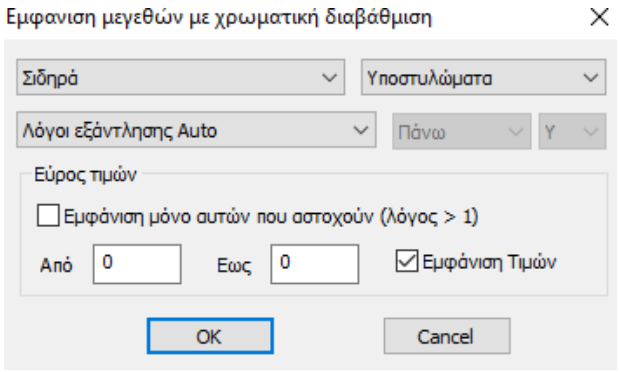
Then select *Layer Sizing*, For all members belonging to this GROUP



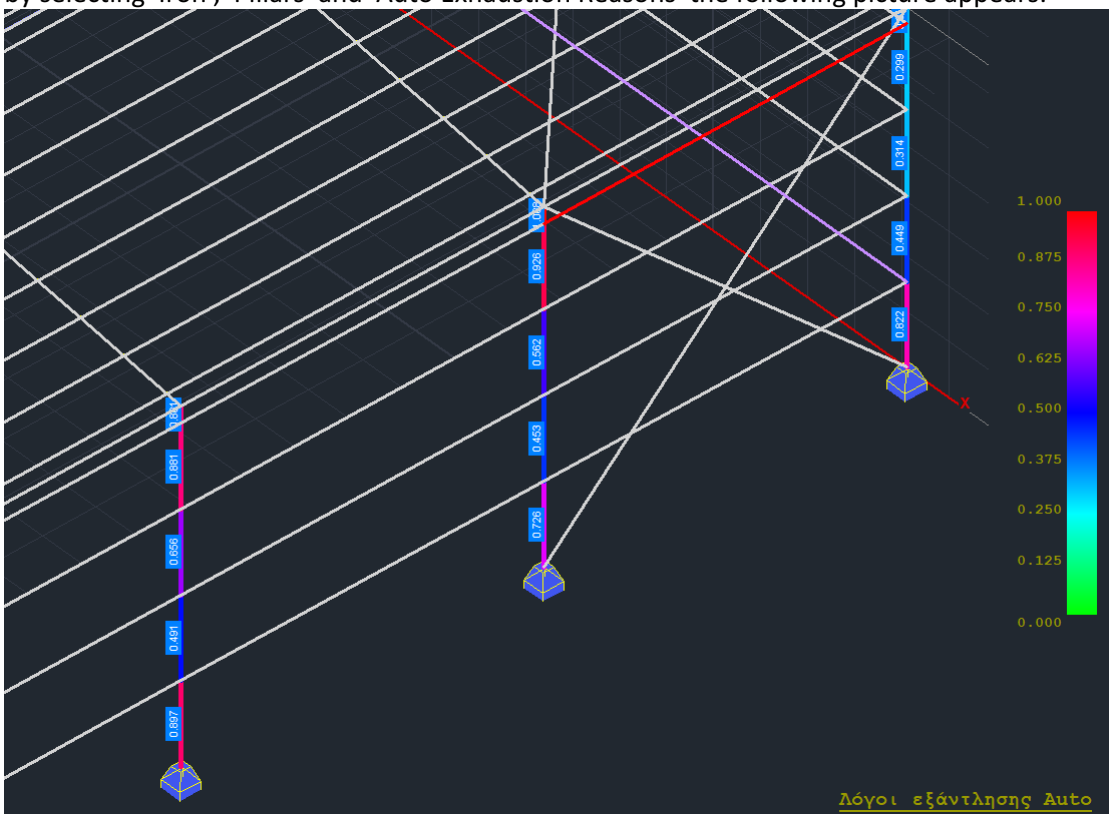
In this example all the checks (General (Auto) and individual) have been done. Right-clicking on the desktop



and selecting "Show Color Gradients" opens the dialog box:



by selecting 'Iron', 'Pillars' and 'Auto Exhaustion Reasons' the following picture appears:

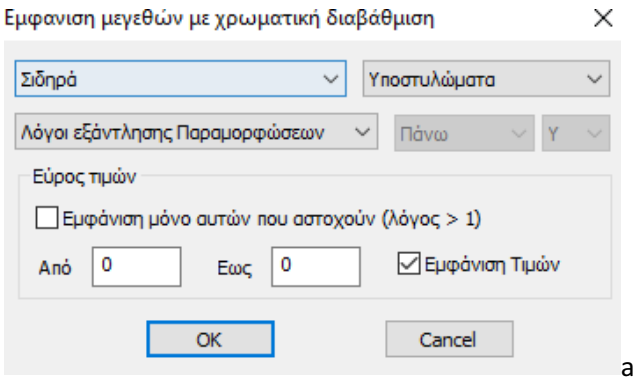


Other reasons for exhaustion (cases 2-8) appear in the same way

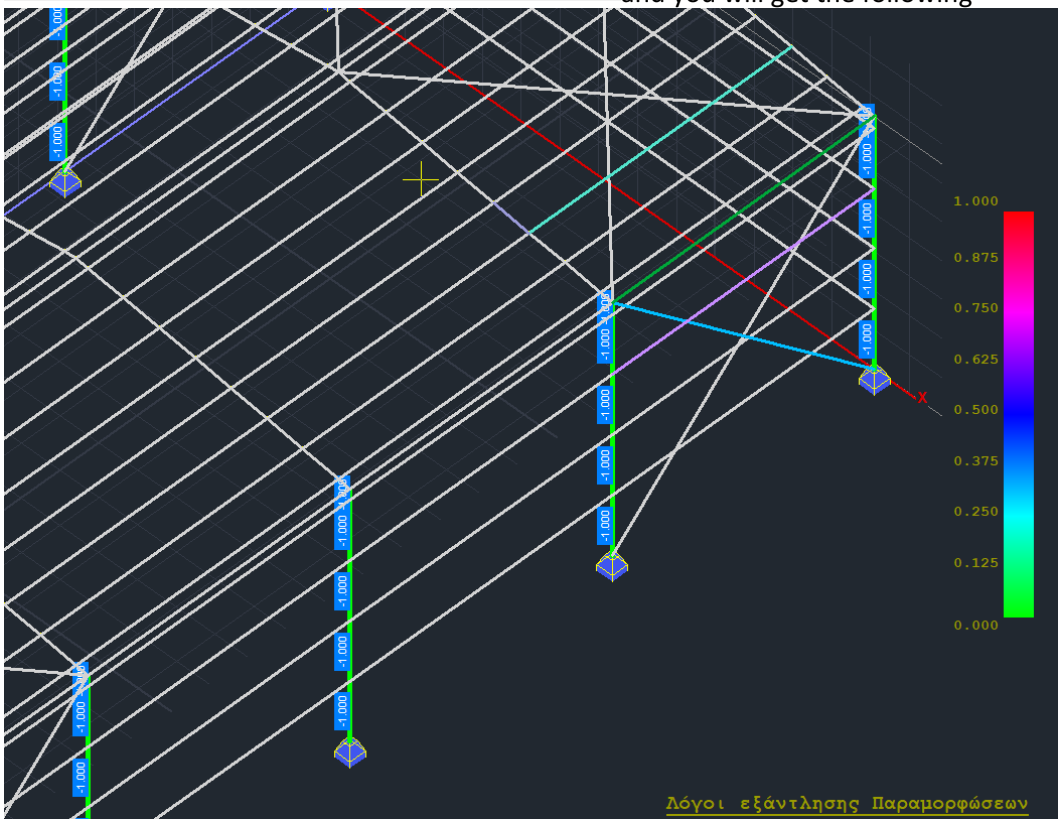
OBSERVATIONS

- The ratios for all elements that are pillars and belong to this layer are displayed.
- If you had run the checks the old way, only the members with the worst sizes would appear in the display.

In the checks (9-13) for bending, deformation and movement there are cases where this check is not required. In this case, in order to separate the member, the value entered is -1.
Let's look at control 12 for example. Reasons for deformation exhaustion.



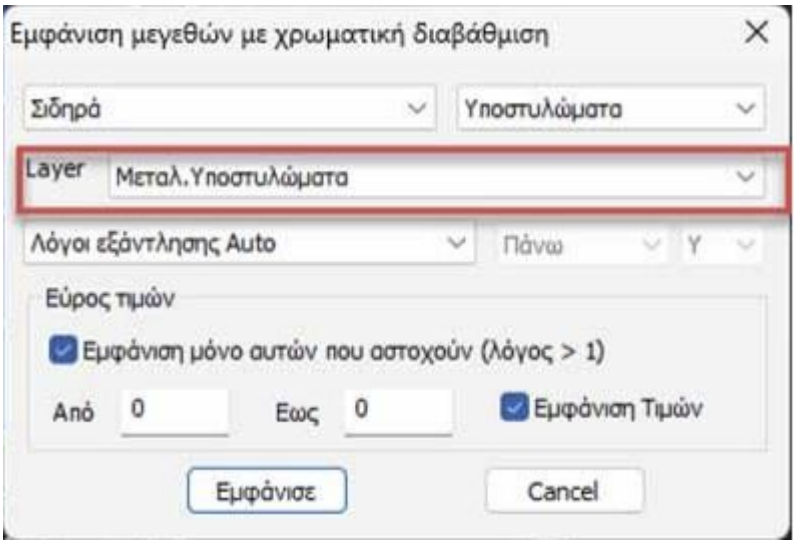
and you will get the following



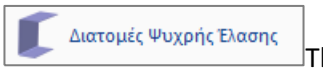
Similarly, you display the exhaustion reasons (1-13) for the beams.

NEW!

In the new version of the program the possibility of selection and per layer in the display of various strength ratios in steel structures was added



1.3 Cold Rolled Sections

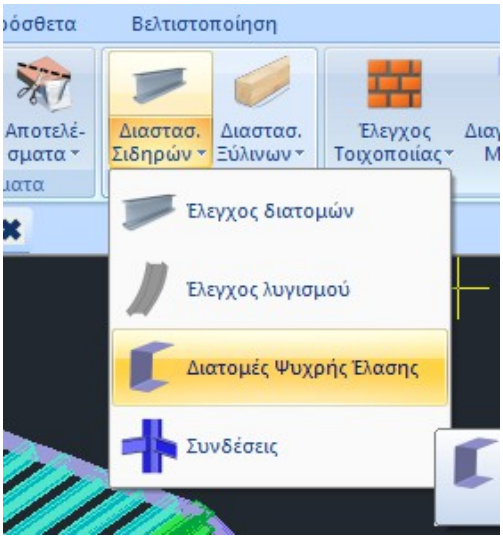


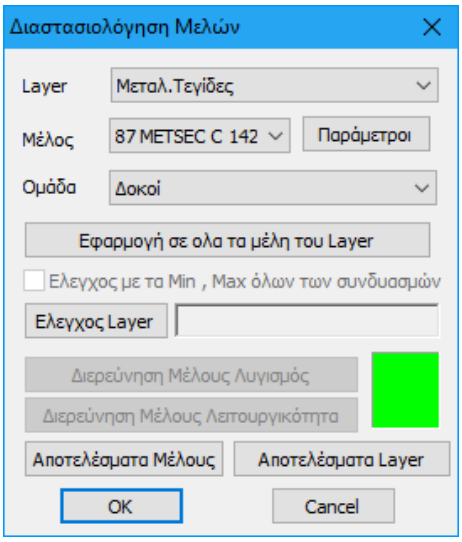
This instruction is for Cold Rolled cross-sectional checks.
The sizing of cold-rolled components involves:

- **Strength check at section level**
- **Strength check at member level**
- **Functionality check**

The procedure for selecting the members and the checks to follow is similar to that of bending for hot rolled products.

The main difference with hot rolled elements is that now the checks at section and member level are done with a **common command** (see picture) and not separately.
Another important feature is that all members and their cross-sections are checked for **all combinations**.





Otherwise, the sizing steps are the same as for hot rolled elements (per layer, member consolidation, bending parameters, etc.).

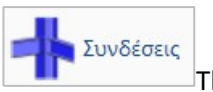
DIMENSIONING DOCUMENT

The results of the sizing are displayed either per member or per layer. In the second and more general case the structure of the issue is as follows:

- **Page 1: General cross-section data 1**
  Dimensional and property information of the original and ideal cross-section
- **Page 2: Active cross-section of cross-section 1 (Part A)**
  Active cross-sectional dimension information of N, My and Mz ratios
- **Page 3: Active cross-section of cross-section 1 (Part B)**
  Active cross-section properties information of N, My and Mz ratios
- **Page 4: Cross-section level check for the 1th member with cross-section 1**
  Endurance tests according to §6.1
- **Page 5: Member level check for the 1th member with cross-section 1**
  Durability checks according to §6.2 & 6.3 and functionality check §7
- **Repeat steps 4 & 5 :** In case of several members of the same cross-section within the layer.
- **Repeat steps 1 to 6:** In case of several cross-sections within the layer.

The issue per layer can also be extracted when creating the **Study Sheet.**
(See in the User Manual the chapter "Cold rolled sections")

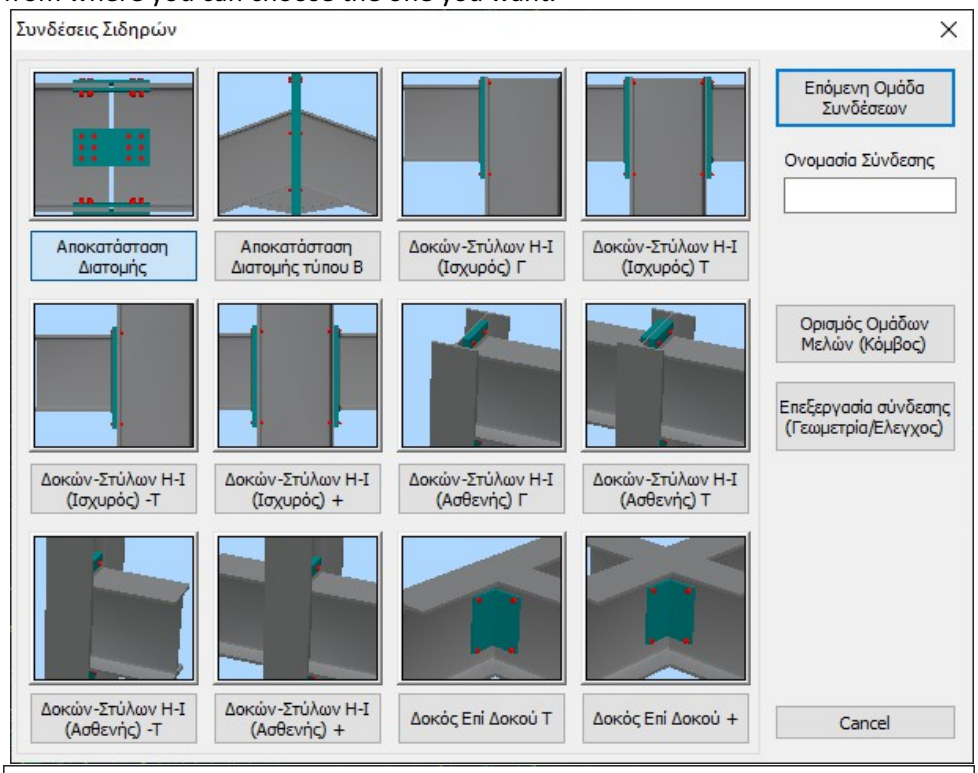
1.4 Links

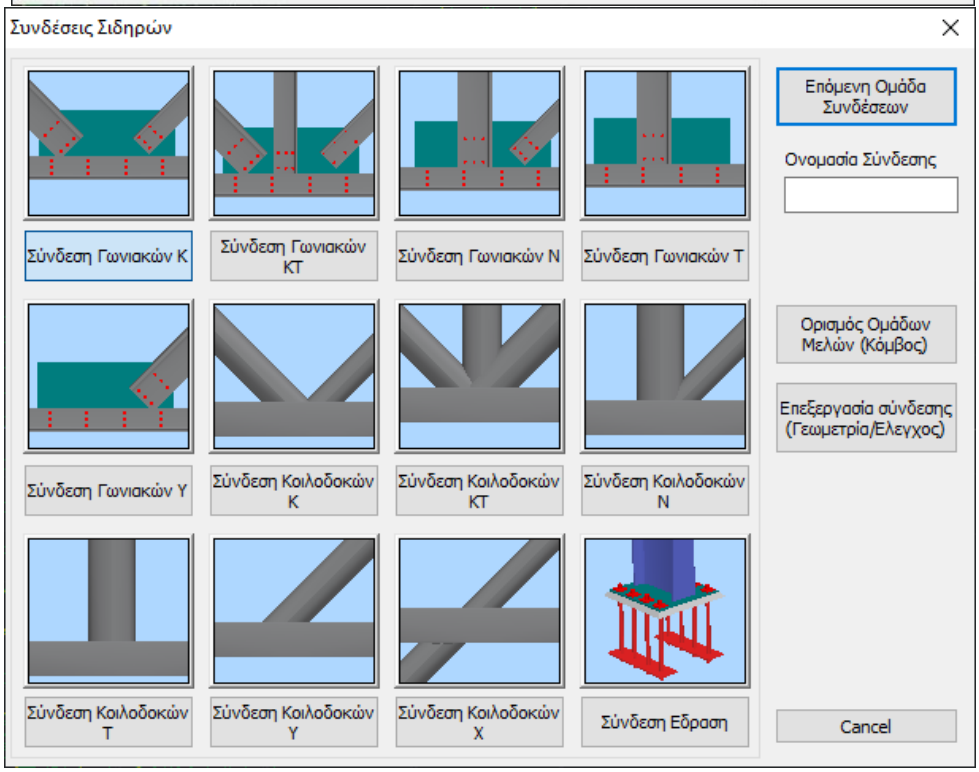


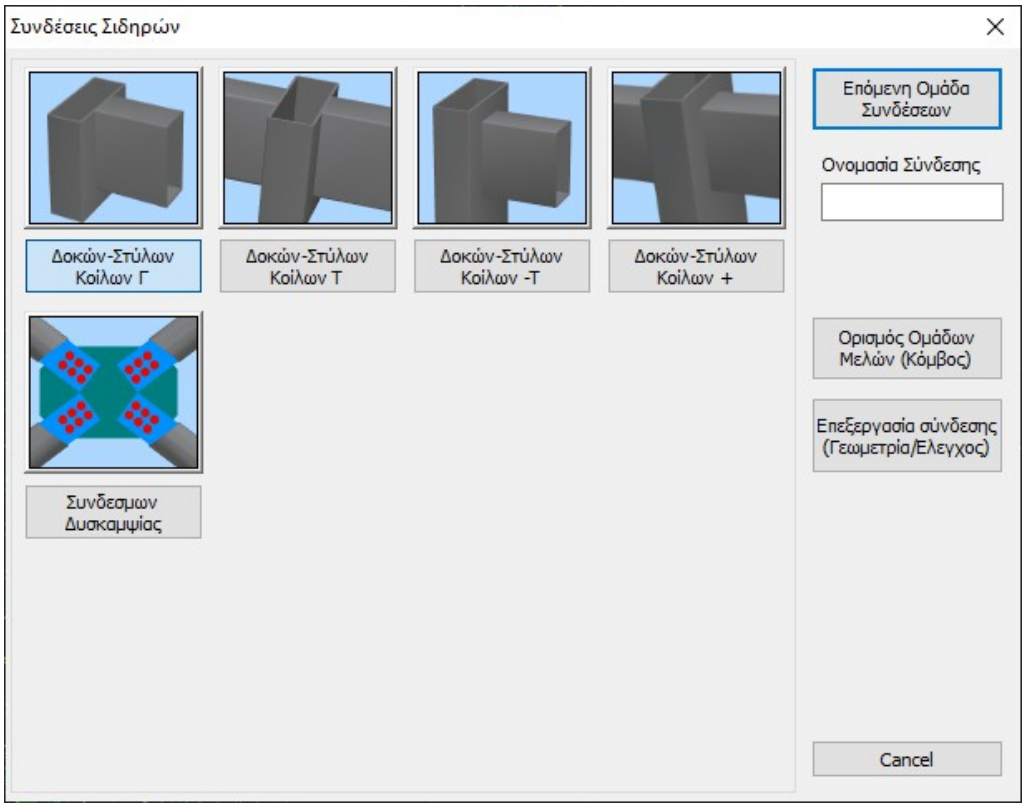
The last chapter of dimensioning for steel structures is the dimensioning of the connections of the structure. Select the command and you have two options to proceed with the sizing of the connections:

CHAPTER 10C 'DIMENSIONING'

A) Click on the "Connections" command and then right-click on the space () to display the library with all the available connections
from where you can choose the one you want.



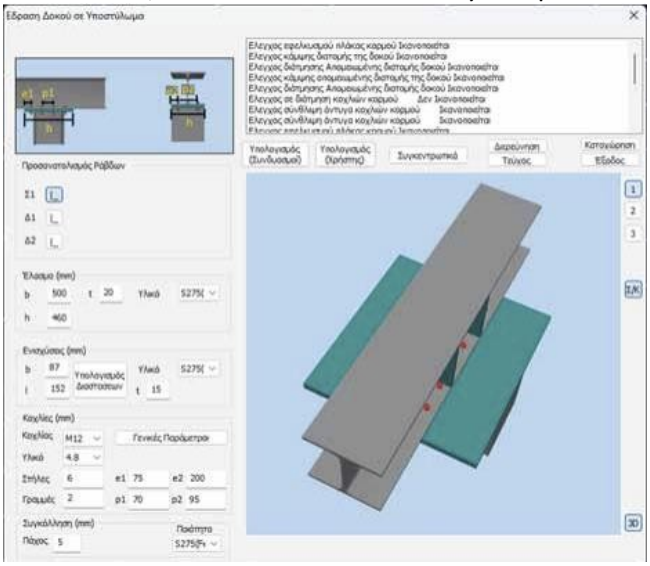




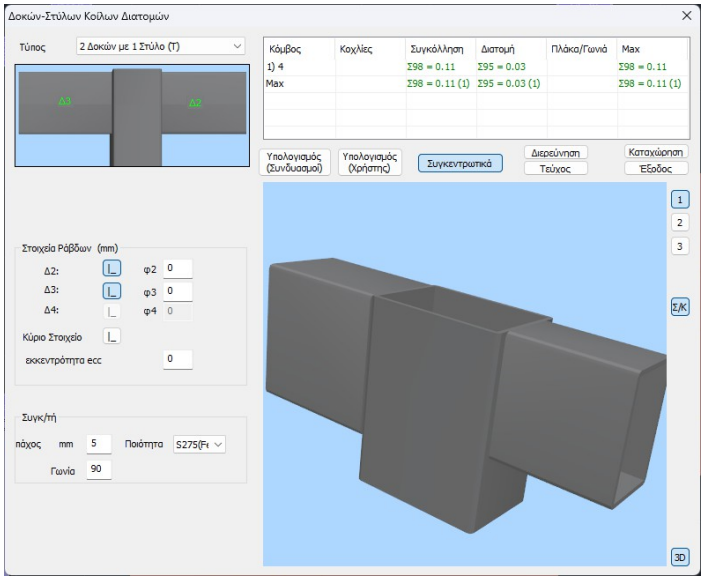
New Metal Connections

New connections have been added to the rich library of metal connections in SCADA Pro:

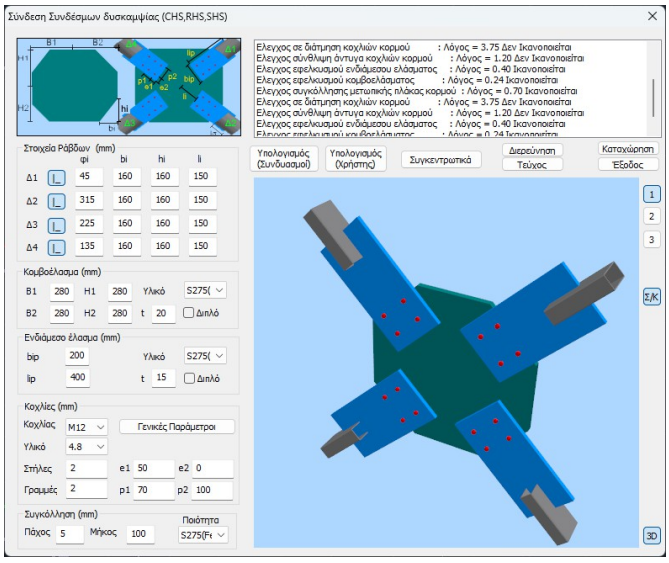
- The new Beam Seating Connection to Subframe with cavity beam or double tee sections, for which all the necessary EC3 part 8 checks are performed.



- The new beam - column connection with cavity beam sections, for all the necessary EC3 part 8 checks are carried out.

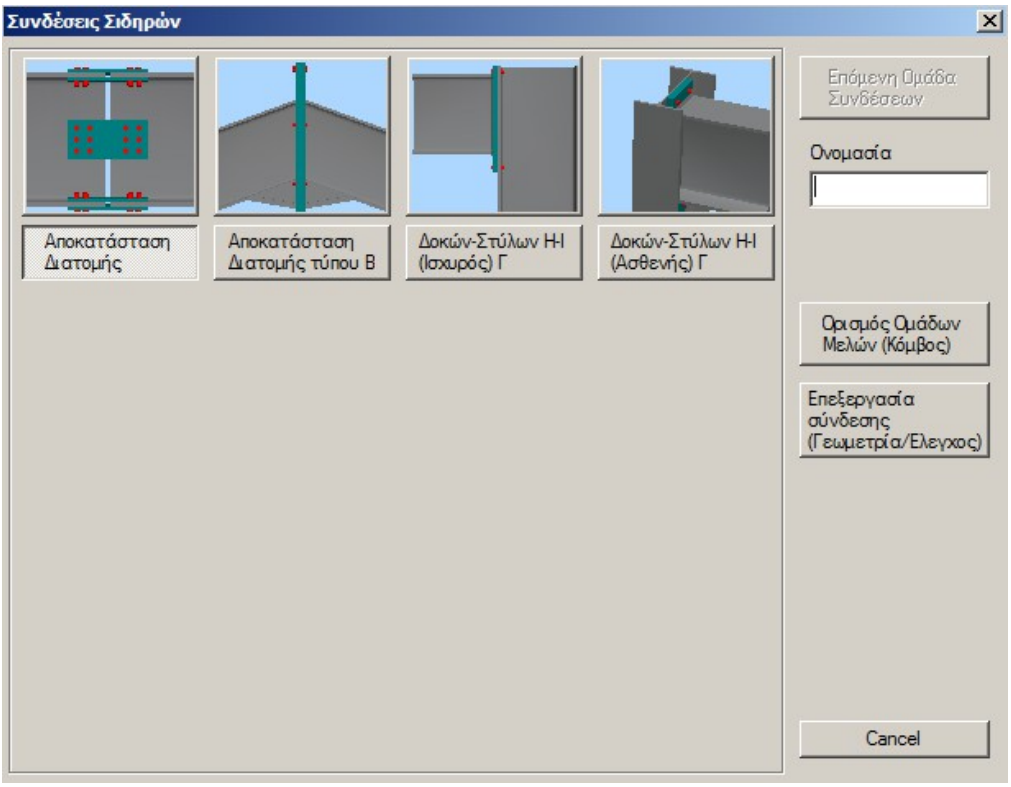


- The new connection of stiffness connectors (Cross-connection of anti-vibration connectors) with cavity cross-sections,



for which all necessary EC3 Part 8 checks are carried out.

B) Alternatively, you can click on the "Links" command and then left-click on the members you wish to link. Right-clicking then displays a window in which only possible connections consisting of only two members are included.



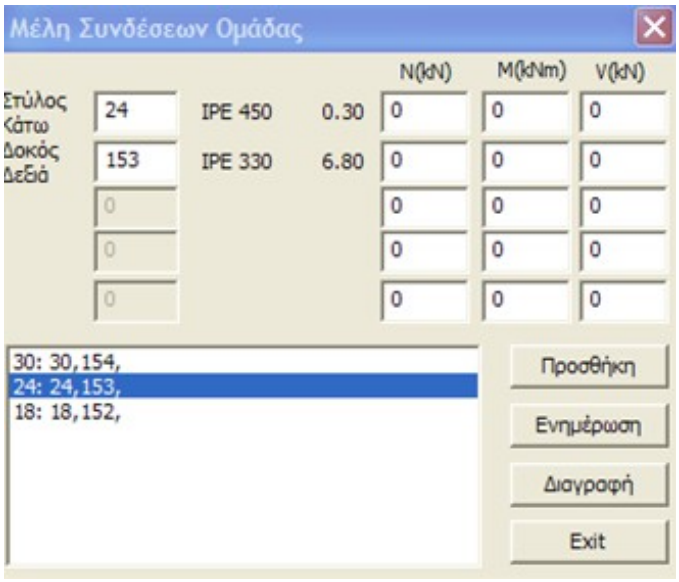
EXAMPLE:

For example, select member 30 (column) and member 154 (beam) in sequence. Right-click to display the window with the 4 possible connection types. Select the last (to the right) connection which corresponds to a Beam - Column connection of cross-section type H or I on the weak axis. You will then enter a name for this connection (e.g. dok_styl_asthenis).

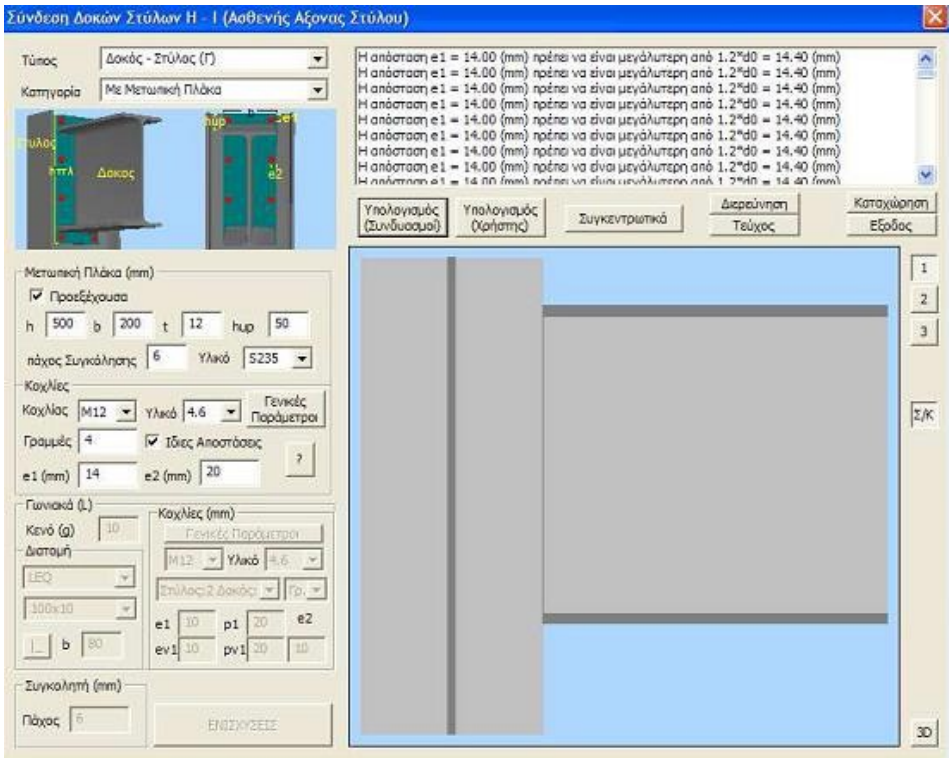
ATTENTION:

- The name should be in Latin and there should be no spaces between the words.

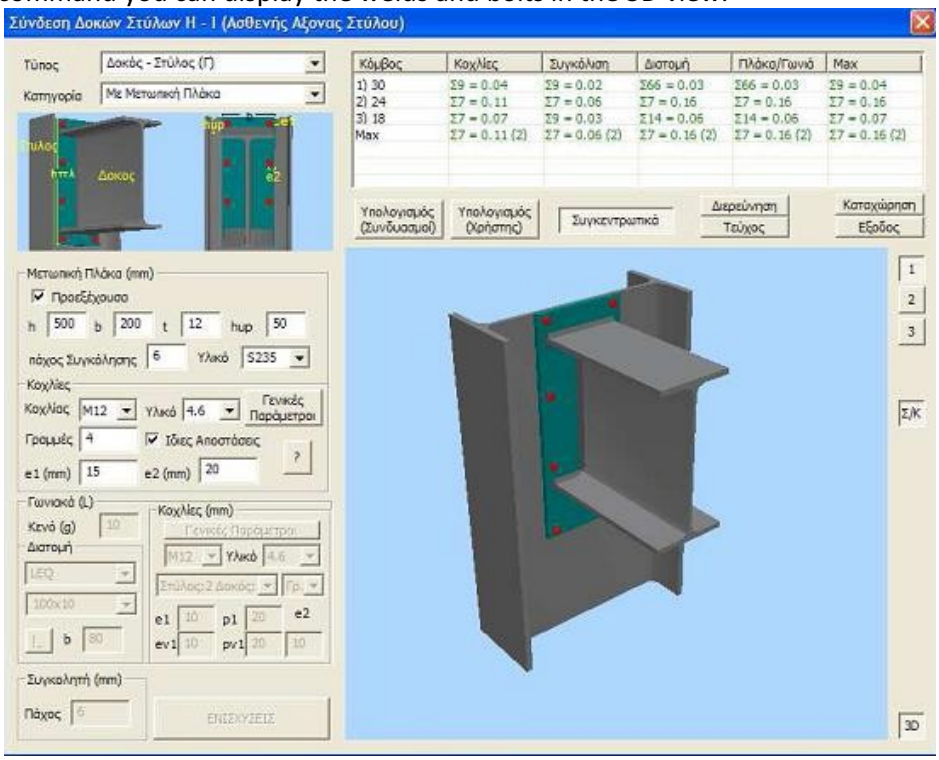
Then select the "Define member groups" command and in the dialog box you can add other similar cross-section pairs (column - beam) or add your own values for the N,M,V intensive sizes to the existing pair. To add other similar pairs, click on the "Column Bottom" field and then select Subcolumn 24 on the desktop. Similarly then click on "Beam Right" field and select beam 153 (or just type in the corresponding member numbers in the fields if and when you know them). To add your selections click on add.



Essentially this way you can massively size all of the
connections of the beam members of the girder which are connected to the weak axis in the same way (bolts or welds, plate geometry, etc.) and which have common
cross-sections (IPE 450 - IPE 330 beam). The program will automatically calculate the intensive sizes of each pair and proceed to the dimensioning of the connection based on the most unfavourable combination. This way you will not have to guess where in your construction the most unfavourable beam - post connection will be developed in the patient axis, while if one connection is satisfied, all other connections of the same type will automatically satisfied.
Then select "exit" and then "Edit Link-Geometry Check".
The window appears automatically, through which you can set the number of the
the type and geometry of the specific connection. Give the characteristic values shown in the figure or try to create your own connection.
To then check the adequacy of the connection with the combinations of
analysis select the "Calculate (Combinations)" command. Initially the program will perform a geometric check of the connection (e.g. if the bones are too close to the edge of the plates). If there is a problem an error message is displayed accordingly in the top right field . In this particular connection change the distance e1 from to 15 cm and click again on "Calculation (Combinations)".



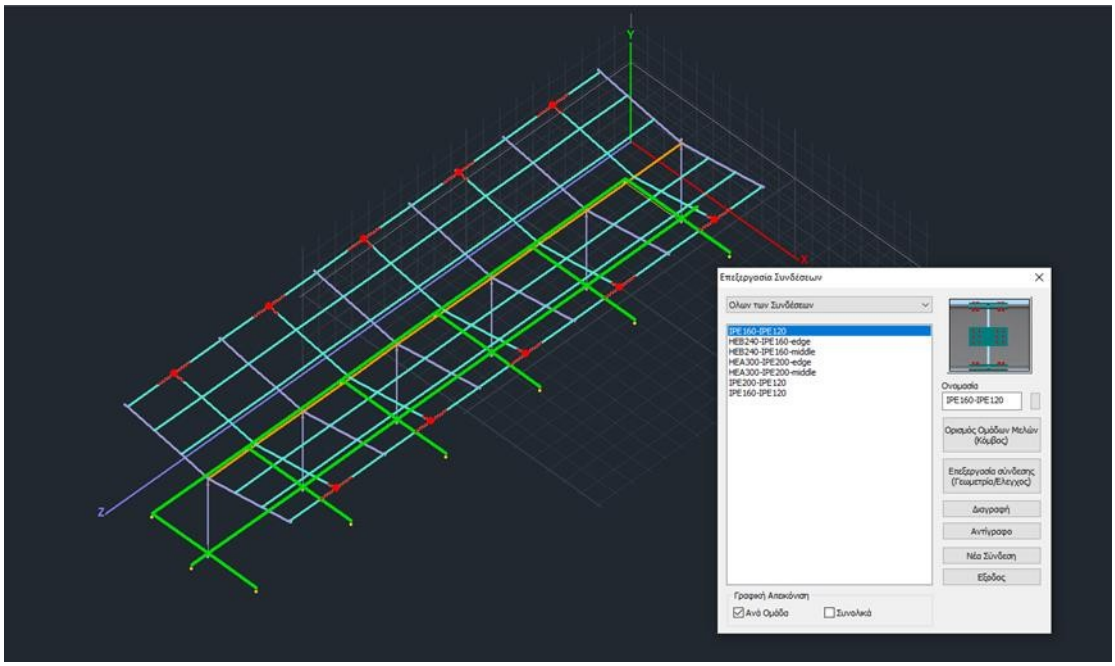
If you click on the 3D command (bottom right) you will see a three-dimensional representation of the connection which is dynamically updated as you make changes to the parameters. Buttons 1, 2, 3 correspond to side view -1, side view -2 and plan view -3 and via the S/C command you can display the welds and bolts in the 3D view.



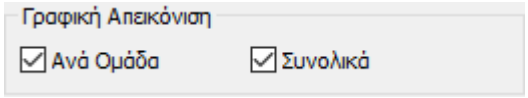
If the geometry errors are overcome, the program will perform the calculations and display all the checks required by Eurocode 3 for the specific connection. In summary you can see the results in the corresponding field. There, in green font, the adequacies will be displayed while in red font the failures of the connection will be displayed. If all checks are sufficient the program will be able to proceed to the registration of the connection as well as the automatic generation of the drawings. Otherwise the process is interrupted and then you will have to change some values of the connection to continue. In the investigation as well as in the issue you can see in text format the results of the checks in detail or in summary.
Finally, click on the entry and exit to return to the connection types window.

Graphical display on the 3D vector of the metal connections that have already been dimensioned

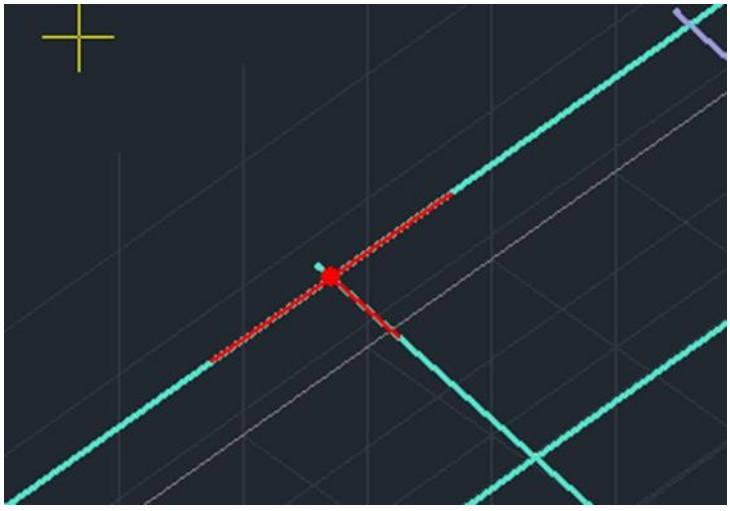
In the new versions of SCADA Pro a very useful tool has also been added, which allows you to see graphically on your metal carrier which connections you have dimensioned.
You can see them all in total, or by group of connections, and you can also see which members are participating in each connection! This way you can therefore have a better overview as to which links you have sized, whether you have correctly grouped similar links and finally whether you have correctly selected the members participating in those links.
Let's see in detail how the command works:
In the edit connections dialog box where all the groups connections I have created are displayed



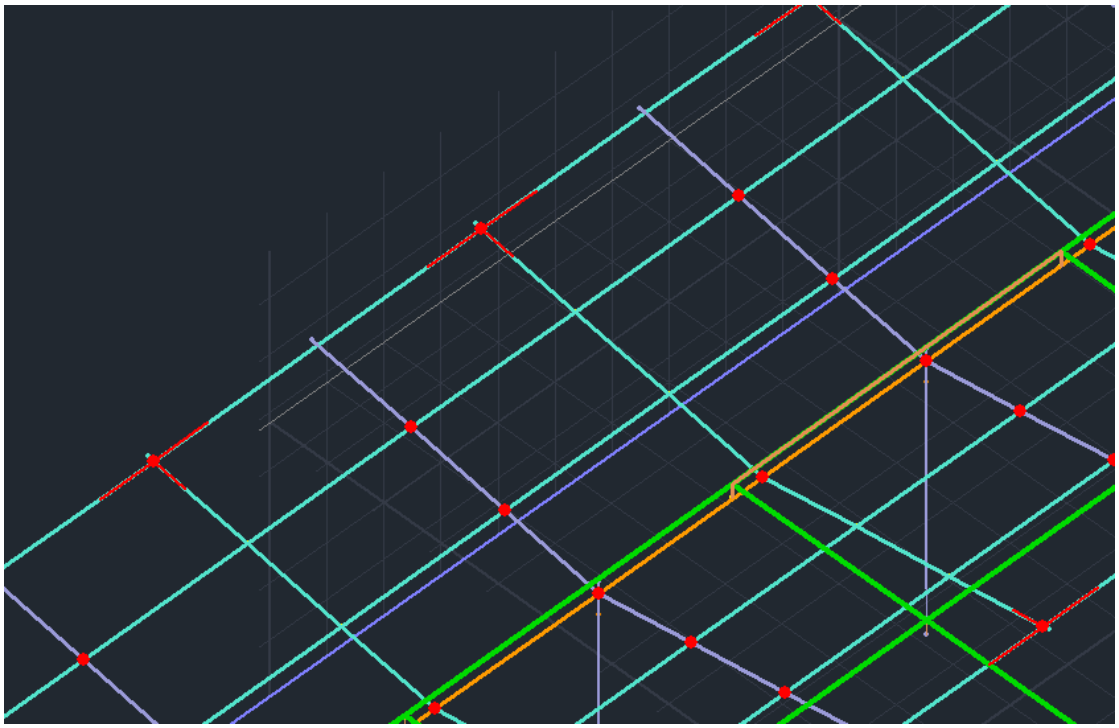
At the bottom of the dialogue box, the commands :



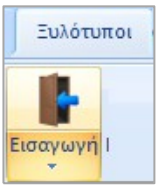
If you select "By Group" in the vector, the nodes where connections have been defined and belong to this group are displayed in red. The members connected to this node are displayed in red.



If you select AND "Total", all connections are displayed in the operator regardless of the group, but for the connections that do not belong to the selected group, the members connected to them are not displayed.

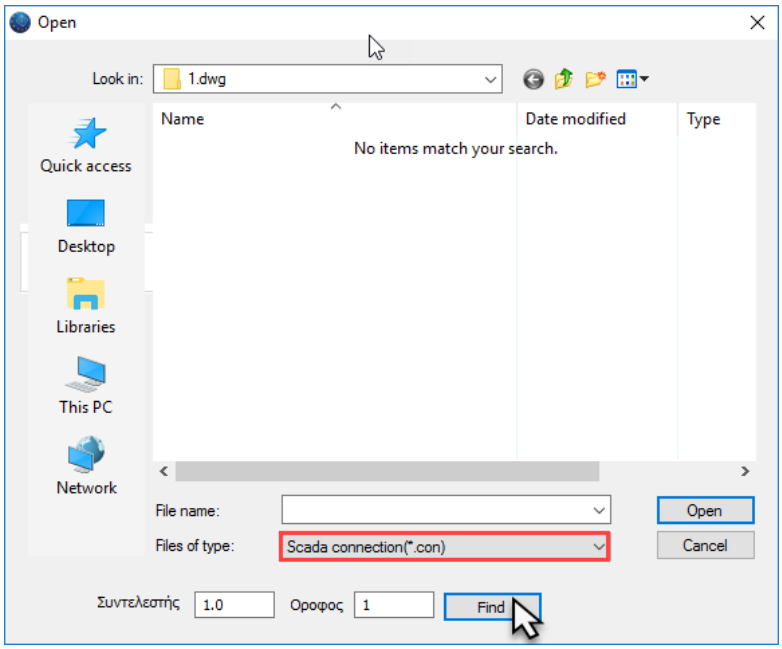


If you select only "Total" all connections are displayed without showing the connected members. The display remains on the screen even after you close the connections dialog box if one or both of the checkboxes are checked. This makes it easier when you want to see where you haven't created links so you can fill them in. Displaying also the members connected to the link prevents possible errors.
Plans of the registered connections are in the study file, specifically on the route:
C:\scadapro\ "Study" \scades_Synd\sxedia
And you open them within the scada drawing environment with the command:

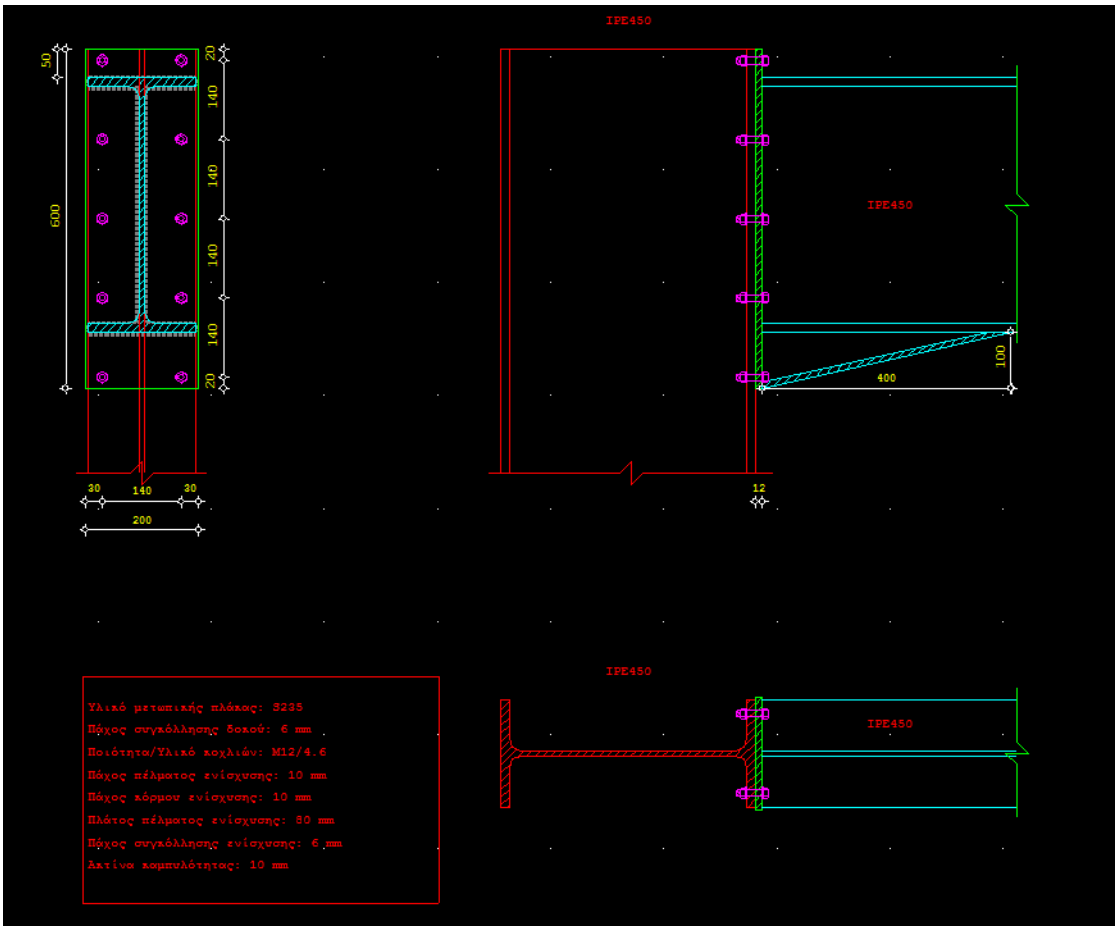


And in the dialog box:

- in Files of Type select **Scada Connection**
- press the **Find** button



In the Search File window that opens, you select the link and open its drawing that includes two views, a cross-section and the detailed table of the elements of the link.


IPE450
50
600
20
140
140
140
140
20
30
140
30
200
IPE450
400
100
12
IPE450
IPE450
Υλικό μετωπικής πλάκας: S235
Πάχος συγκόλλησης δοκού: 6 mm
Ποιότητα/Υλικό κοχλιών: M12/4.6
Πάχος πέλματος ενίσχυσης: 10 mm
Πάχος κόρμου ενίσχυσης: 10 mm
Πλάτος πέλματος ενίσχυσης: 80 mm
Πάχος συγκόλλησης ενίσχυσης: 6 mm
Ακτίνα καμπυλότητας: 10 mm

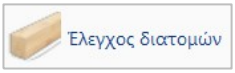
Wooden

The "Wooden" field includes the commands related to the resolution of wooden sections with the adequacy check, the buckling check and the dimensioning of the connections.

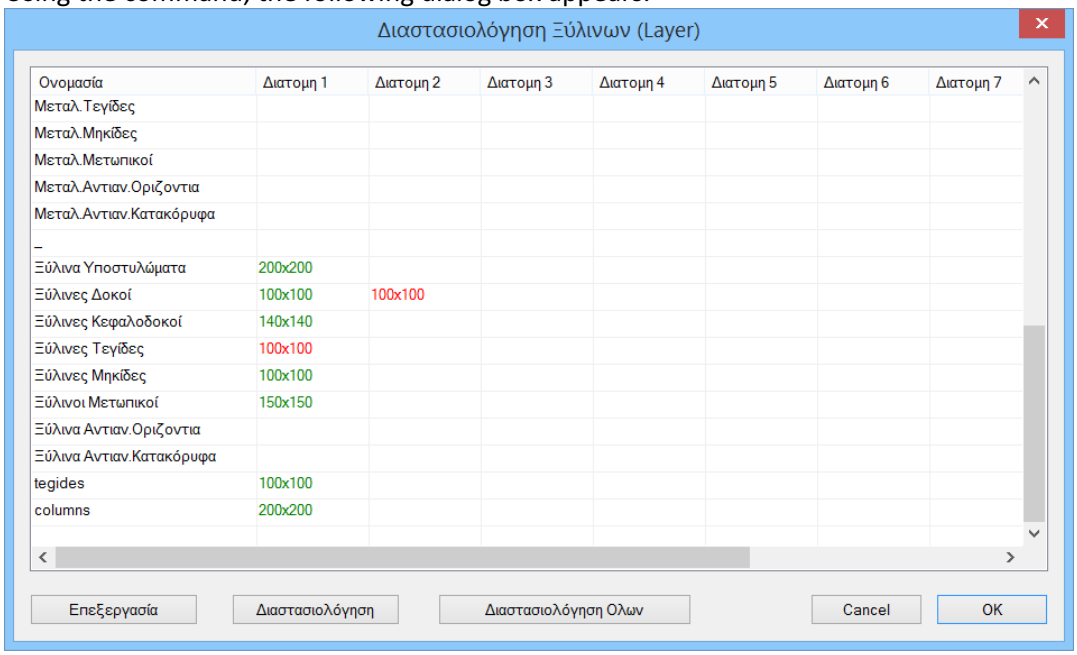
• A prerequisite for sizing is that you have called and executed the corresponding combination file in configuration dialog box.

OBSERVATION:
The dimensioning process of wooden cross-sections is similar to that of metal cross-sections.

2.1 Cross section control

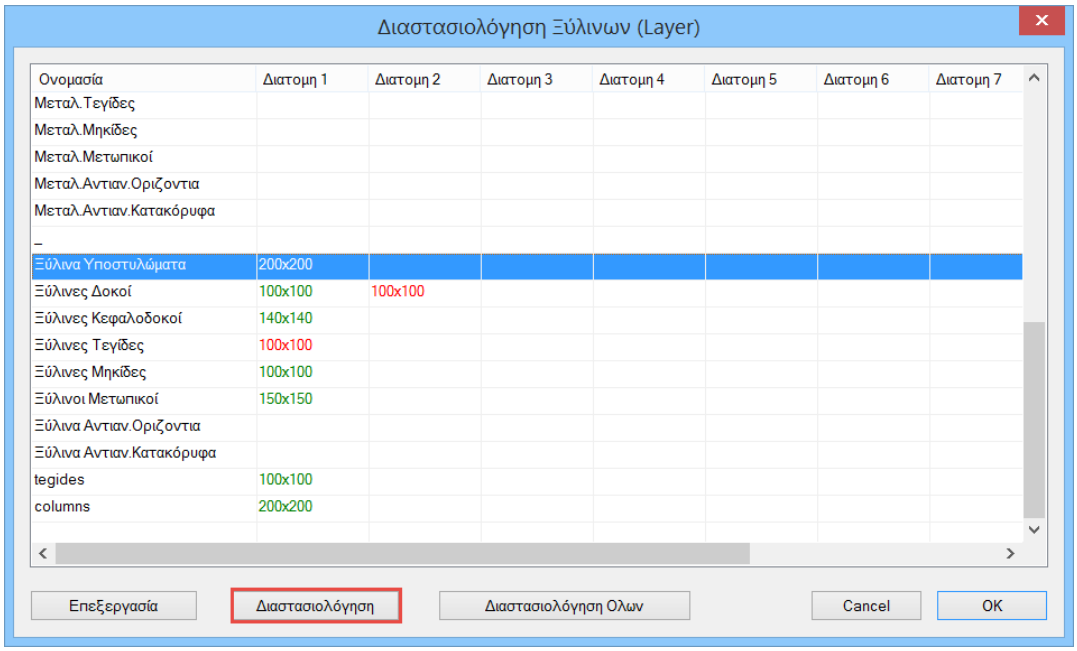


to check the adequacy of the wooden sections.
Using the command, the following dialog box appears.

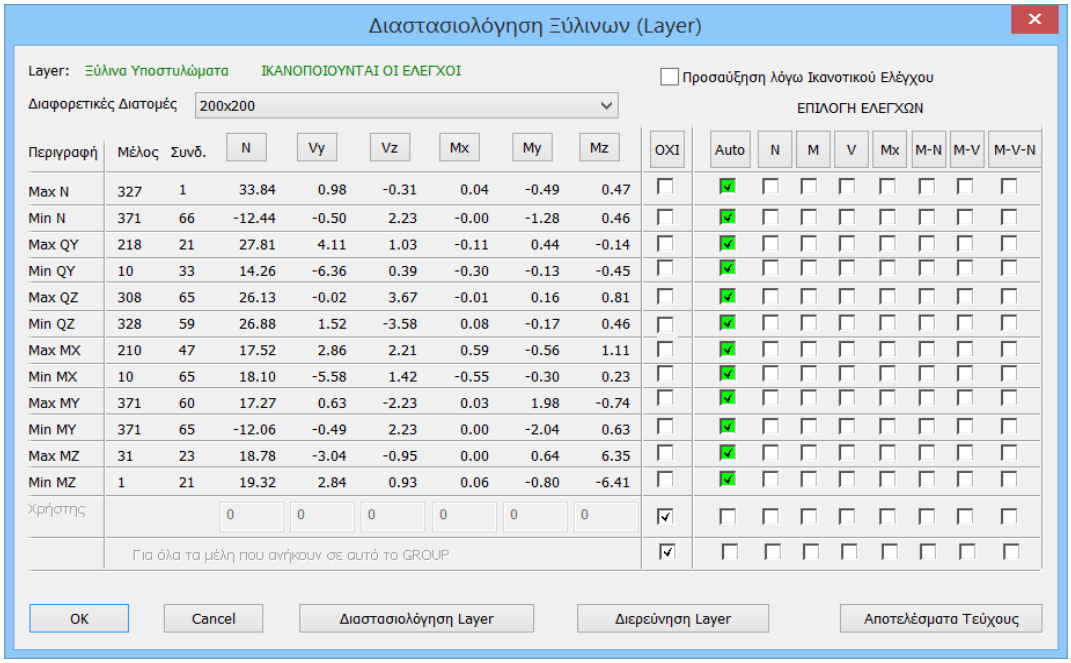


The first column is the layers that exist in this study and in the following columns are the types of wooden sections that exist in these layers.

With the "Dimensioning" option and after you have selected a layer, the dimensioning (the check of the cross sections) of the specific layer is done, the program "colors" the specific layer green if all the elements involved in it do not fail and red if some of them fail.



Selecting the "Edit" button will display the following dialog box:



OBSERVATION

The detailed procedure followed for the dimensioning of a layer is described in the corresponding paragraph of the metallic sections
(See **Iron>> Iron Sizing> Cross-sectional Check**)

2.2 Bending control

This mandate is used to control the members. That is, checks are performed for each member belonging to the specified layer:

Limit State of Failure

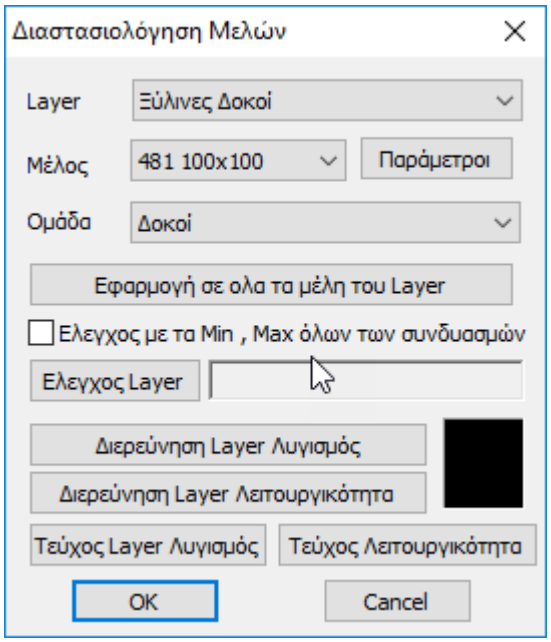
- Check for bending (lateral) buckling due to axial compressive force
- Torsional buckling check due to bending moment.
- Check for torsional bending due to the simultaneous presence of axial compressive force and bending moment.

Limit State of Functionality

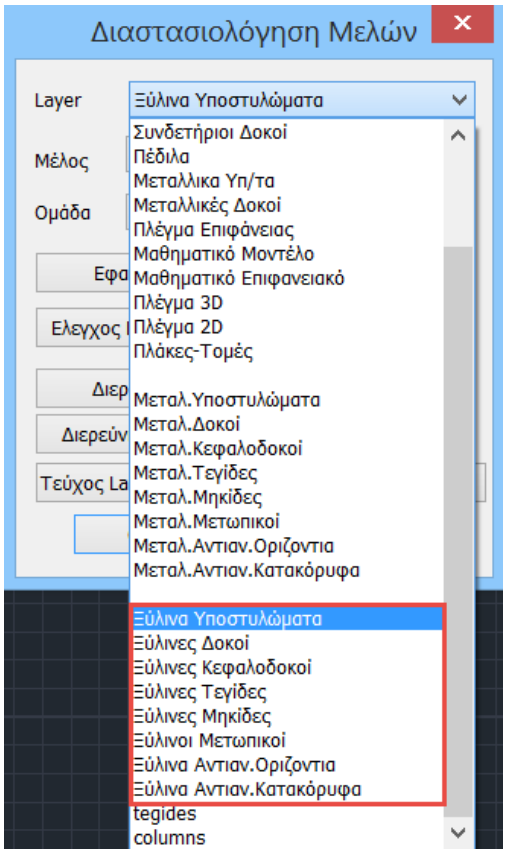
- Member deformation control
- Edge (node) movement control

- A prerequisite for sizing is that you called and executed the corresponding combination file in the configuration dialog box

Using the command, the following dialog box appears.

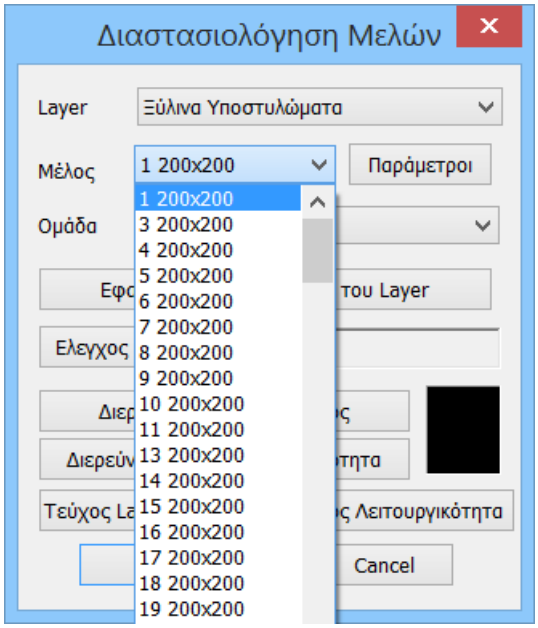


The check is done per layer. So first select from the list



the layer (e.g. Wooden Lodges) you want to dimension.

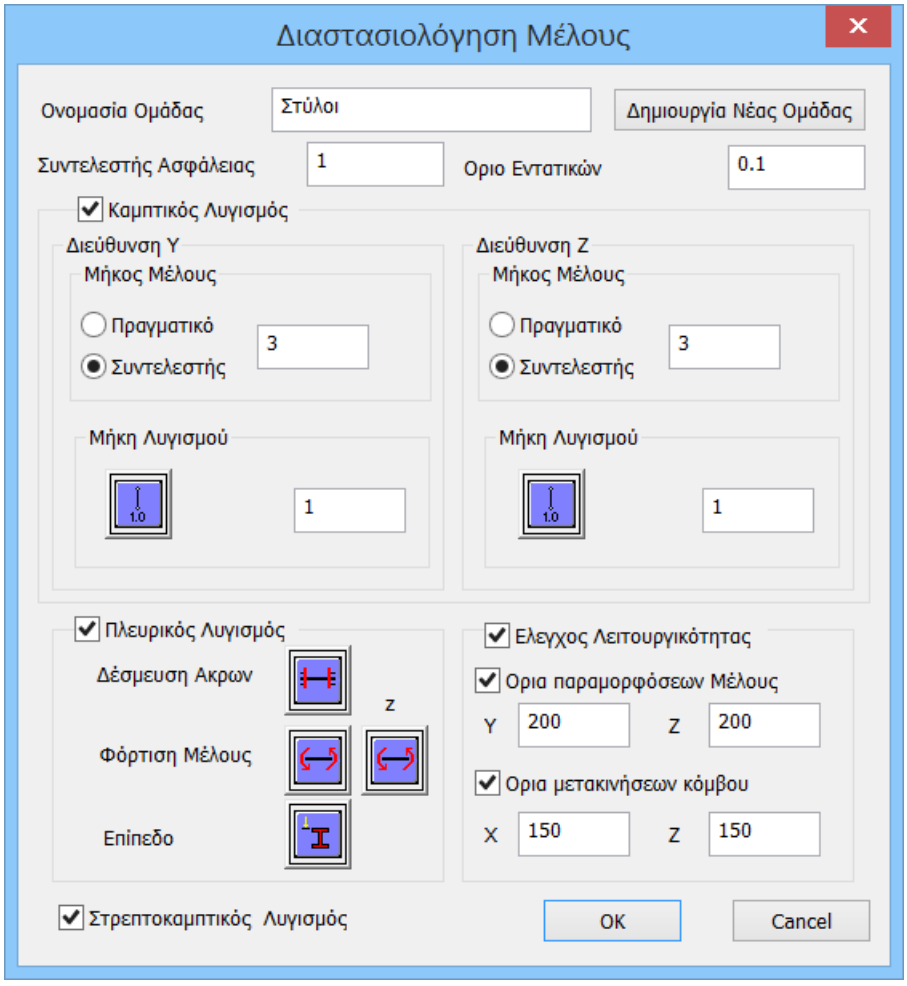
By selecting the layer, all the members of this layer and their cross-section are displayed in the "Member" list.



The first step in dimensioning the layer is to define the dimensioning parameters. Because it is possible that for some of the layer members you may want to define different parameters, it is possible that within the same layer you can define different groups of parameters to which the layer members belong. The program has two default parameter groups.

If you want to have the same parameters for all members of the layer, you set them once with the procedure we will see below, keep the default name "Beams" and press the "Apply to all members of the layer" button. The checks will be done with the same parameters for all members of the layer.
In the different case that you want to set different parameters for some of members of the layer, you will set another set of parameters using the procedure explained below. But first we will look at how to define the parameters.

Selecting the "**Parameters**" button will display the following dialog box:



The "Group Name" field contains the name of the parameter group. If you want to create your own group, enter a new name and press the "Create New Group" button.

In the "Safety factor" field you can set the threshold against which the program checks the ratio of the design value (the intensive value) to the corresponding strength of the member. The default value is 1.
In the field "Intensity Limit" there is the limit of the intensities below which the program does not take into account the intensities.

The rest of the dialog box is divided into three parts, each of which deals with the parameters of the Flexural Bending, Lateral Bending and Functionality Checks.
In the Flexural Bending section you first define whether you want to check the flexural bending by ticking the corresponding option. Then you define the length of the member and the buckling length in both Y and Z directions respectively.

OBSERVATION

- In earlier versions of SCADA Pro and prior to the creation of the

command, the user was prompted to set the member length and the buckling length in both the Y and Z directions respectively, following the procedure below:

In "**Length of Membership**":

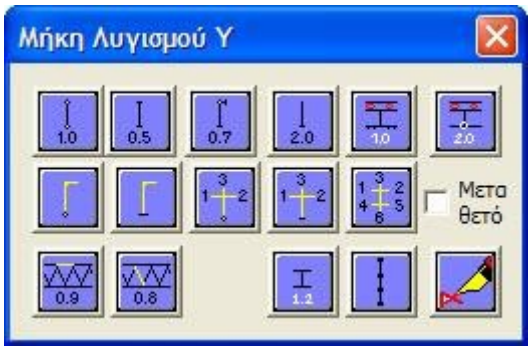
- if you select "*Actual*" you must enter in the field the actual length of the member in m.
- if you select "*Coefficient*" you must enter a coefficient by which the different lengths of the members belonging to this parameter group will be multiplied.

If you want the program to take the actual lengths of the members into account when checking the bending bend, select "Coefficient" with a value of 1.
If you have some members with different or equal lengths that are laterally secured at the same distances (e.g. at 1/3), then you give the value 0.33 and of course create a separate parameter group to which these members belong.

In the new versions of SCADA Pro the definition of the bending length is done through the command "**Consolidate Members**" (see the section Sizing - General) and no action is required in this field. Having therefore followed the procedure of Member Consolidation, in the Parameters field and specifically in the Member Length field, leave as is and proceed with the definition of the remaining parameters.

The next parameter is the member's buckling length which depends on the support conditions of the nodes of the member ends always within the buckling plane. By pressing the 

key the following dialog box appears

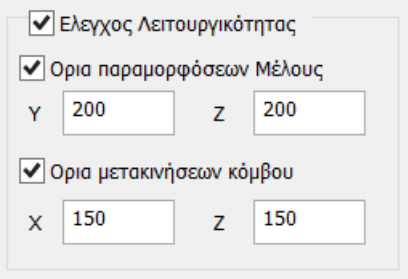


where you select the icon with the support conditions of the member and the program enters the corresponding coefficient for the buckling length.

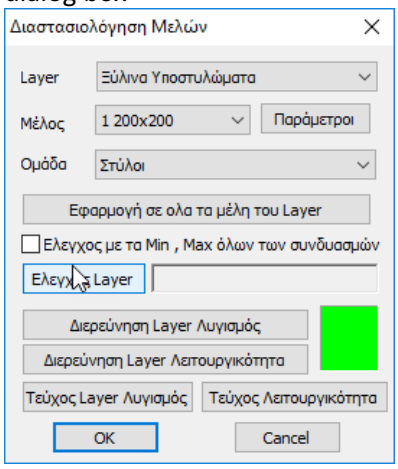
OBSERVATION

The icons are divided into two groups which are described in detail in the corresponding chapter for metallic cross-sections (See **Ferrous>>Sizing of Ferrous>Bending Control**).

The third set of parameters concerns the parameters of functionality



where you specify whether you want to perform the functionality check, the individual Member Deformation and Node Displacement checks, and the respective upper limits (l/220 and l/150 where l is the length of the element) for these checks.
Finally, check the "Torsion bending" option if you want to do this check.
When the configuration settings are complete, press the "OK" button and return to the previous dialog box



By pressing the "Apply to all members of the Layer" button, the program applies the group of parameters you just defined with the default name "Pillars" to all members of the "Wooden Lumberyard" Layer you had selected. Then you press the "Check Layer" button and the program starts the layer execution process for specified "Wooden Lumber Yards" layer.

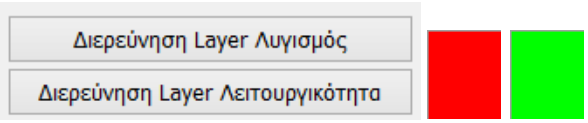
By activating the option 

, the check will be performed taking into account only the maximum and minimum values of the intensities obtained from all combinations, excluding intermediate values, so that the process is completed in significantly shorter times.

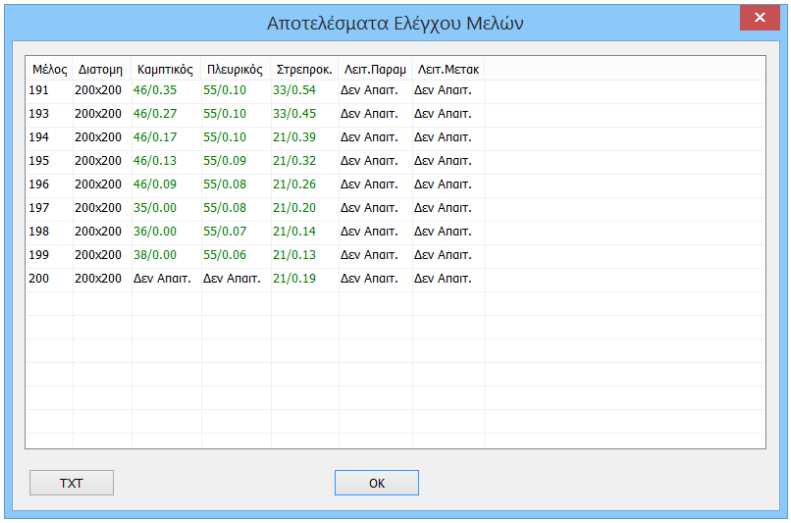
EXAMPLE:

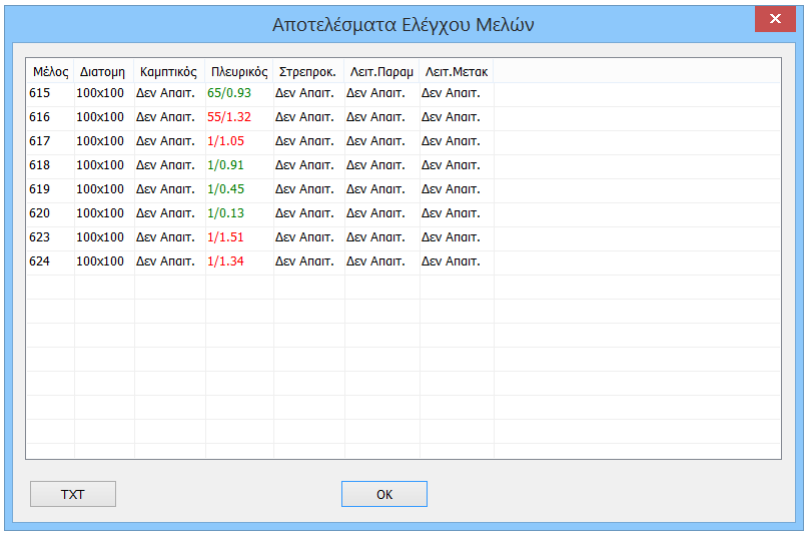
If you would now like to specify another group of parameters to which some of the members of the layer belong, follow the procedure below:
Press the "Parameters" button and open the configuration dialog box again. In the "Group Name" field, enter a name for the new parameter group you are going to create e.g. "L/D_1" and press the "Create New Group" button. Then you define the parameters based on what was mentioned before and press the "OK" button.
The next step is to define which members from the layer will belong to this group of parameters "Sub_1".

Upon completion of the testing process for the specific layer the icon next to the buttons "Investigate Layer Bending" and "Investigate Layer Functionality"



is coloured in a corresponding colour: Red if there is a failure and Green if there is no failure. Double-clicking on this colored icon displays the following dialog box





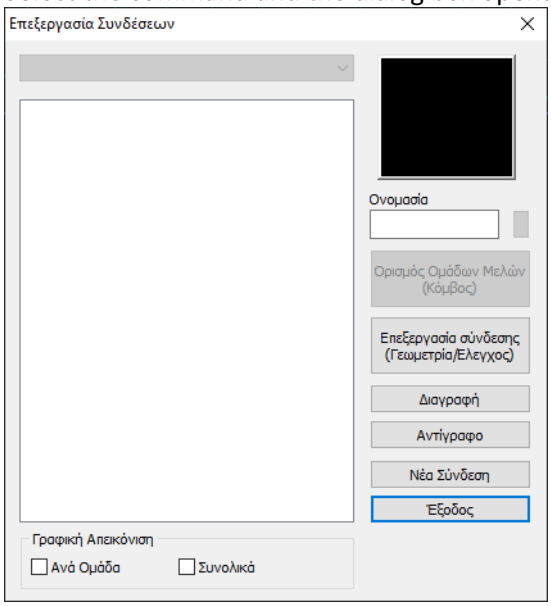
with the summary results of the audit of the members.
In the first column the number of the member, in the second column its cross-section and in the next 5 columns the worst strength ratio and the number of the combination from which this ratio was derived. Green are ratios below the unit and red are ratios above the unit. Where the words 'not required' are shown, this means that the corresponding intensive magnitude was not available or that the axial force was tensile and not compressive.
By selecting the button "Layer Bending" the program displays the summary results of the check in Bending (i.e. for each member the results from the worst combination) while by selecting the button "Investigation Layer Bending" the program displays a complete but very large file with the results of the checks for each member from all combinations. Similar applies to the "Functionality Issue" and "Investigate Layer Functionality" buttons.
The check for the three types of buckling is carried out for each member and for all combinations. However, for each combination, i.e. for each triad N, My and Mz the checks are carried out 4 times based on the following combinations:

- N with min My and min Mz
- N with min My and max Mz
- N with max My and Min Mz
- N with max My and max Mz

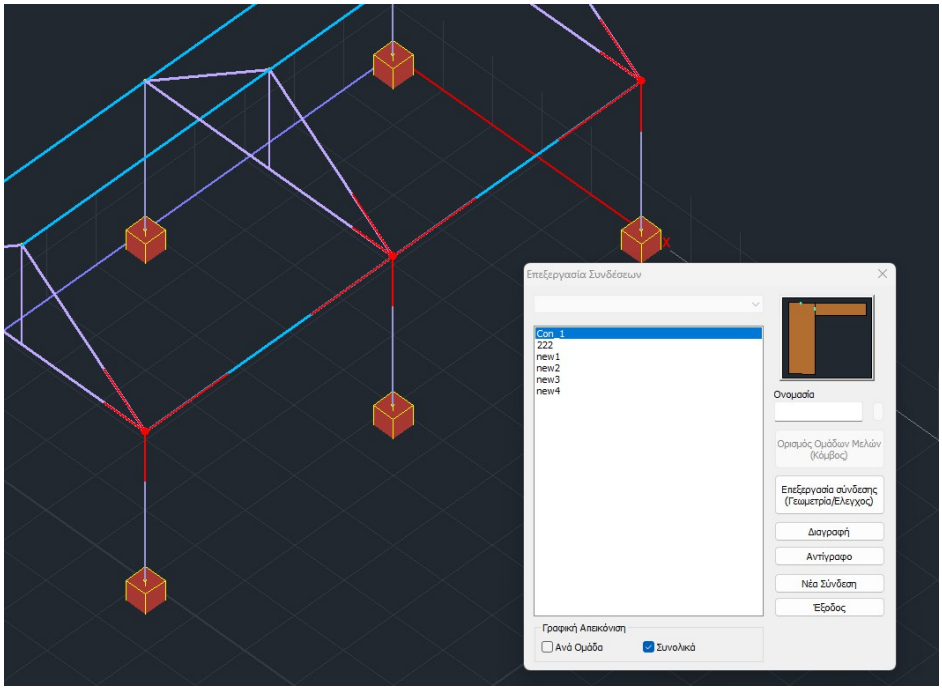
That is why both in the results of the issue and in the investigation, two numbers are mentioned in the combination number: The first refers to the number of the combination and the second refers to the number for each of the 4 previous cases.

2.3 Links

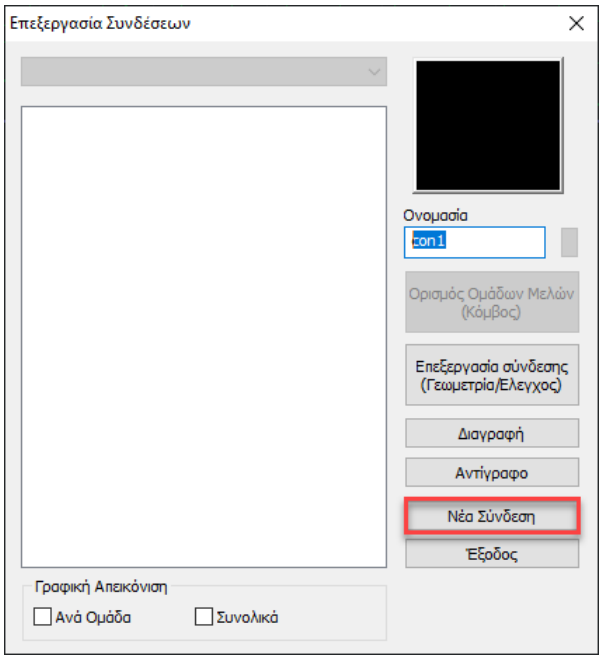
The last chapter of dimensioning for wooden structures is the dimensioning of the connections of the structure.
Select the command and the dialog box opens:



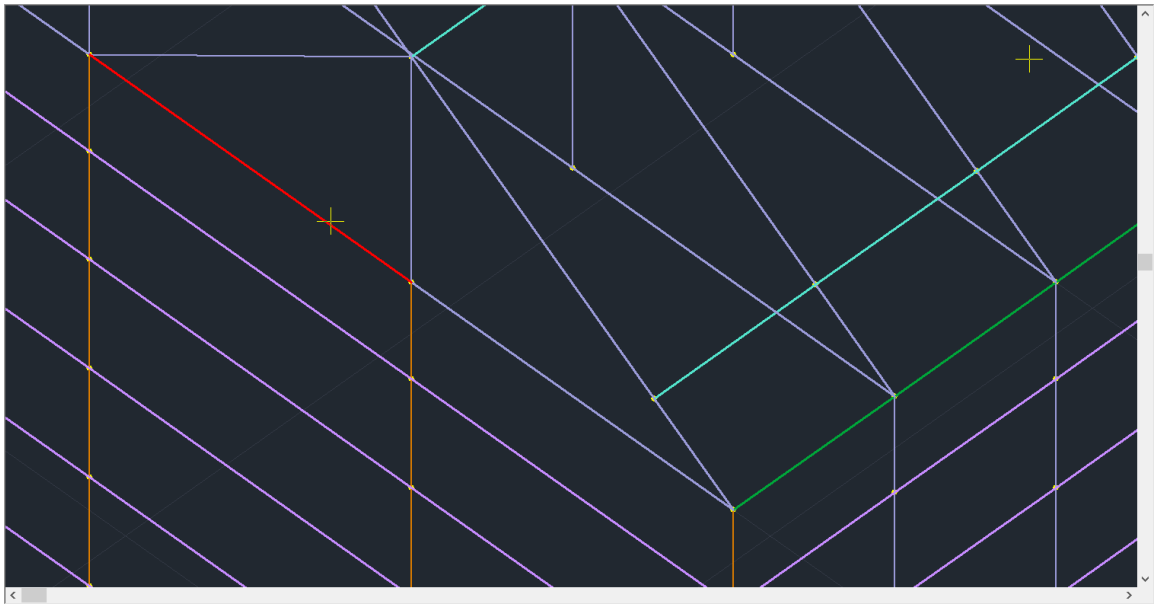
EXCEPTION: Before the first connection is defined the dialog box is empty, then it is filled with the created connections and in addition the possibility has been added to visually display on the vector, the connections that have been defined either by group or in total.



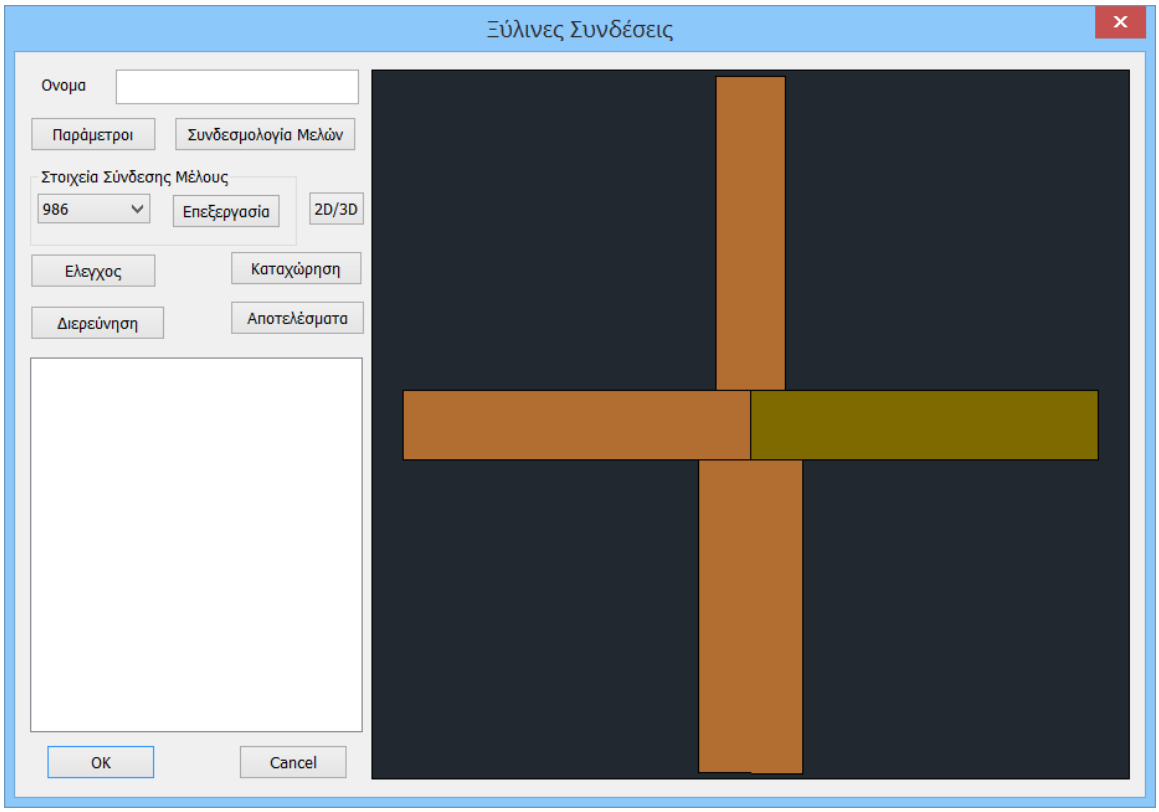
Select the command and in the dialog box that opens, specify a name (the name should be in Latin and there should be no spaces between the words) and select New Connection:



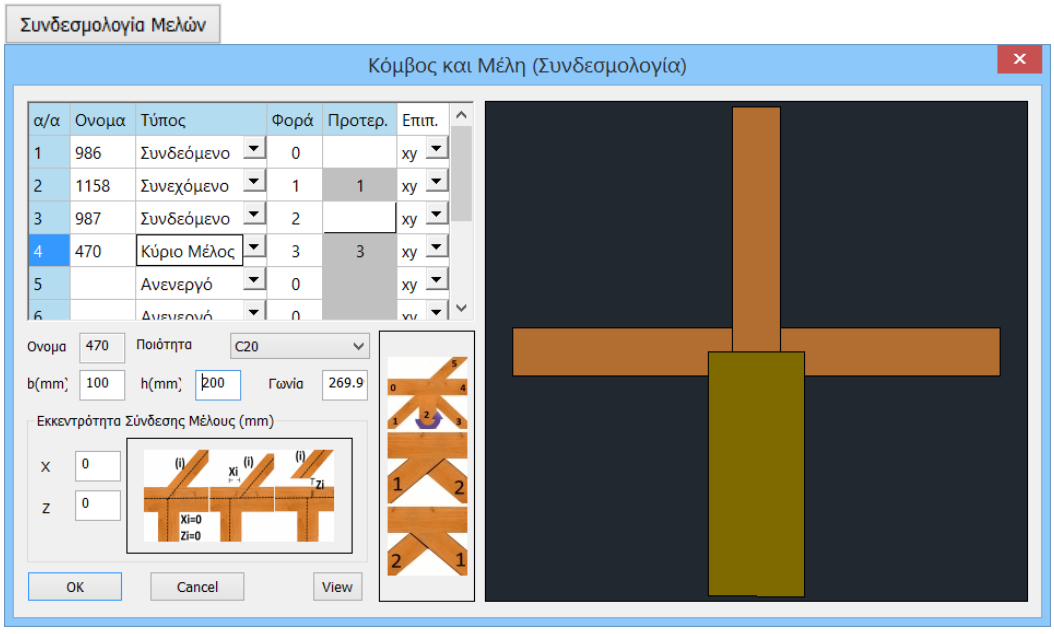
and show the connected ones in sequence.



Right-click to close the option and open the dialog box below:

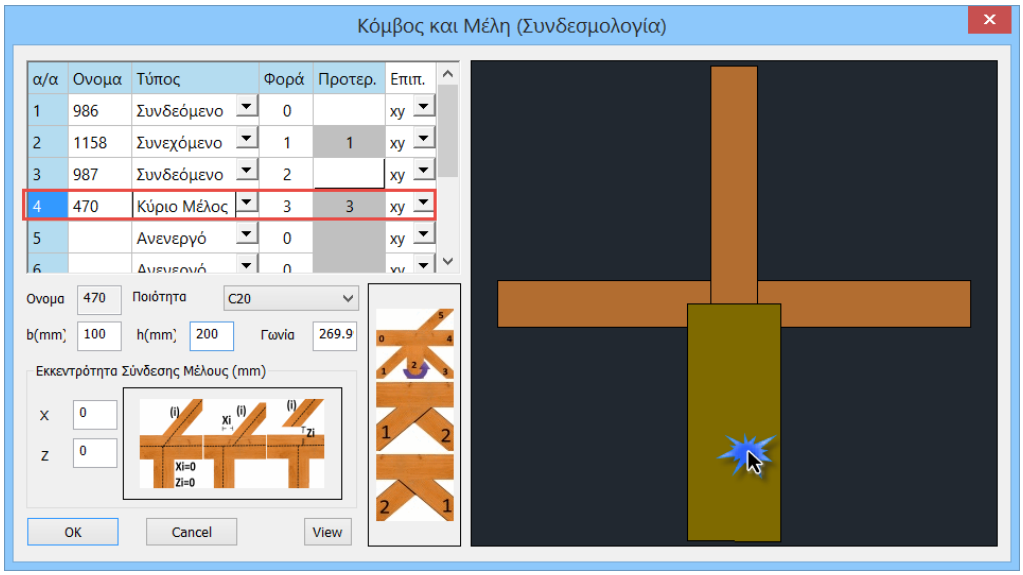


The right part of the window shows the connected members with b and h randomly given by the program. Through the command **Linking Members** the designer defines the actual dimensions of the members.
Give the connection a name and select the **Membership Link** command.

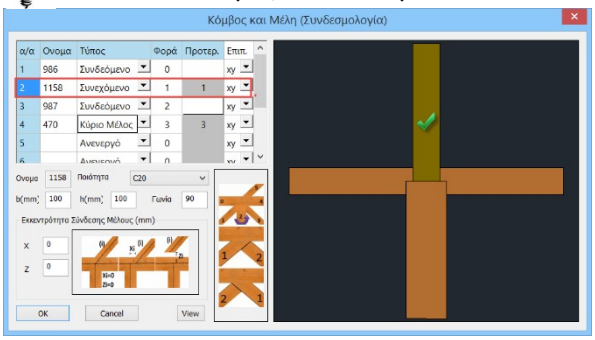


In the first field you set the **type** of membership.

Select graphically by left-clicking on the member to be designated as the Master Member (each connection has only one Master Member). The selected member is automatically highlighted in the list on the left.



For this example, the Principal Member is 470.



In case there is an associated member (e.g. 1158 of the example) this can be defined either as **Associated** or as **Conjoined**
All other members of the association are **Online**

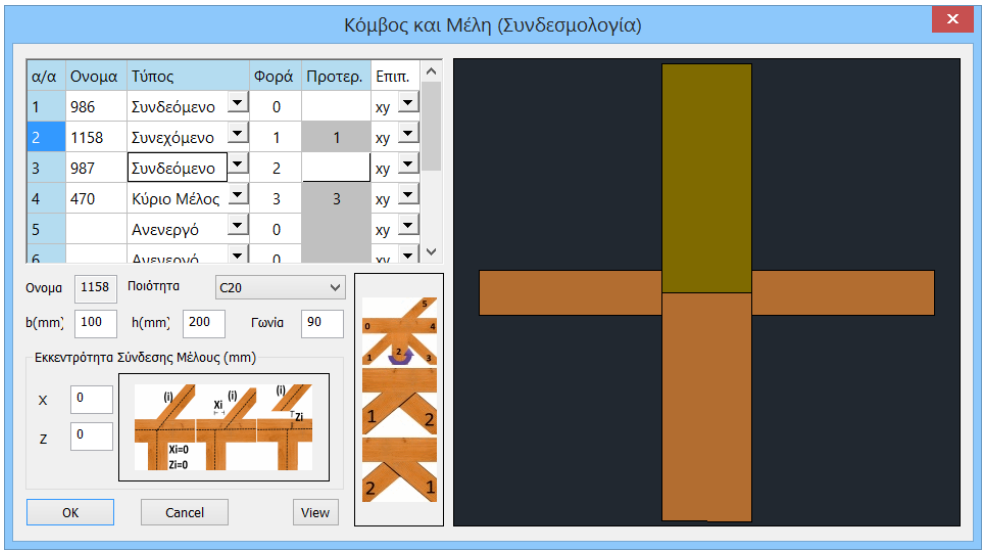
Definitions:

- **Main member**: can be any member of the association
- **Continuous**: a member that is a continuation of the Master member without interruption. It is a single member and cannot have dimensions different from the main member.
- **Connected**: is a member that is connected to other members and may have different dimensions from those it is connected to.

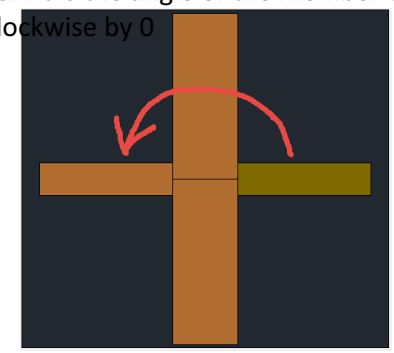
Therefore, you define the press of all members in the same way. The

next step is to set the **dimensions** of each member. You choose

from what list and set the values of **b** and **h.**
b= the thickness of the member (dimension perpendicular to the screen)
h= is the height of the cross-section (dimension at screen level)

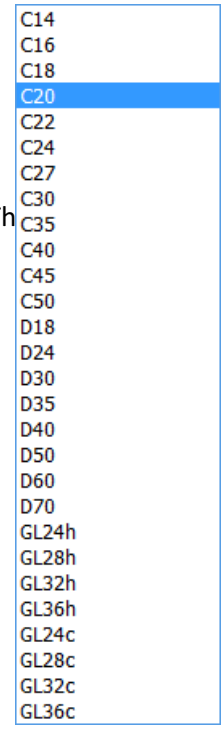


Corner: It is the angle of the member with respect to the connection. Angles are defined anticlockwise by 0



at +x (right of the link)

Quality: to set the quality of each member, select the member and its quality



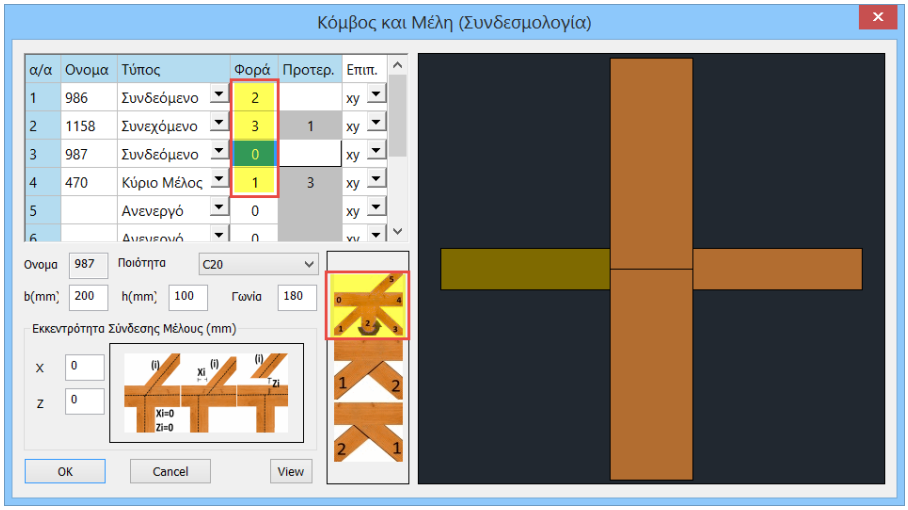
- **ATTENTION:**

The Principal Member and the Continuum cannot have different dimensions. Th
are the same element!

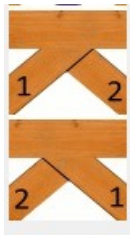
The View command displays the total link with the lengths of the members

Wear: The time of each member is defined according to the plan 

. Therefore, start by selecting the left member and setting its direction to 0 and continue by defining the direction of the other members of the link.



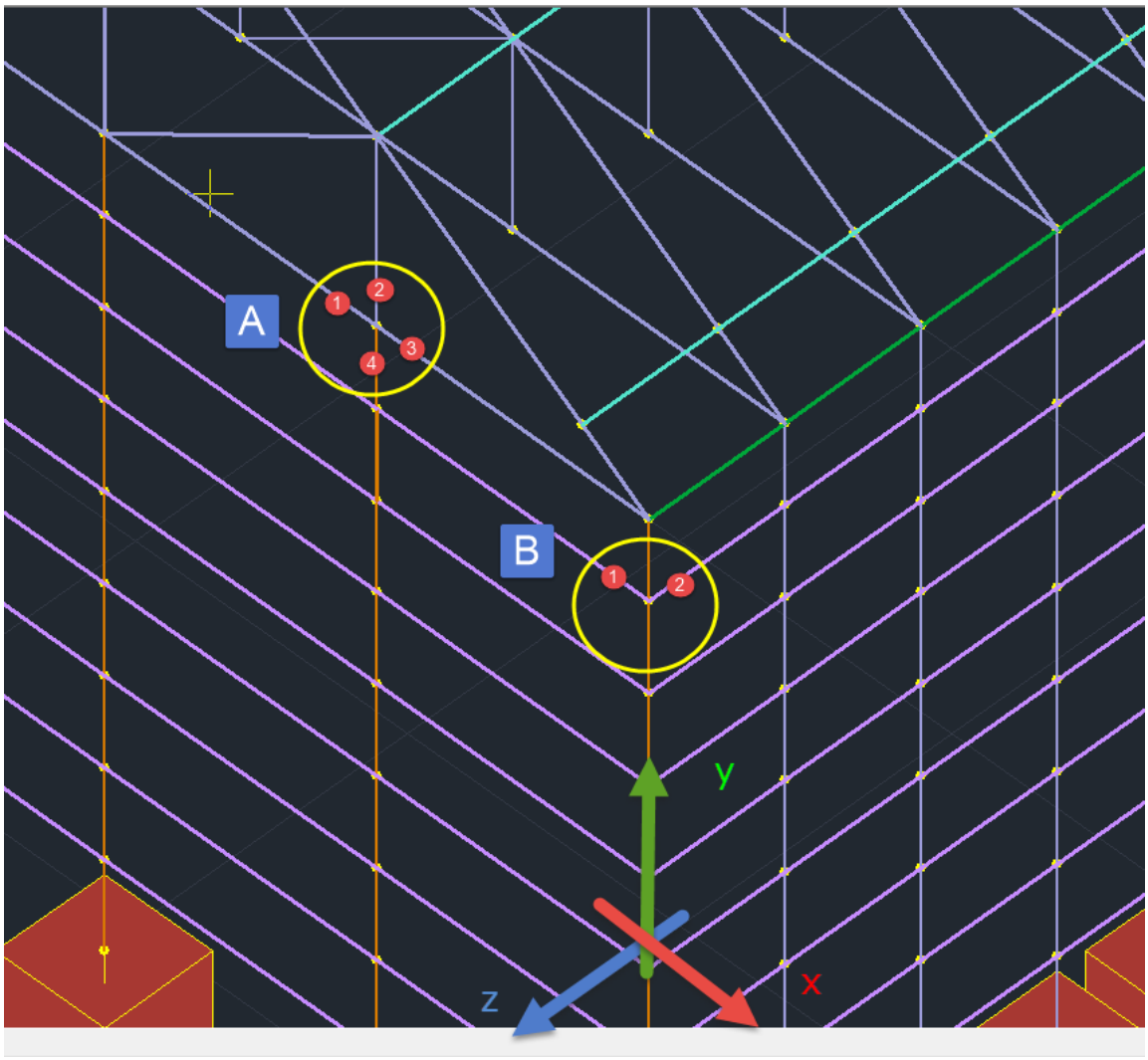
Priority: With the priority you define the Connected Member that "dominates" the connection. This is the "truncation" of a Connected Member that encounters another Connected Member.



In the priority column you set a number only for connected members.

α/α	Ονομα	Τύπος	Φορά	Προτερ.	Επιπ.
1	986	Συνδεόμενο	2	1	xy
2	1158	Συνεχόμενο	3		xy
3	987	Συνδεόμενο	0	2	xy
4	470	Κύριο Μέλος	1		xy
5		Ανενεργό	0		xy
6		Ανενεργό	0		xy

Levels:

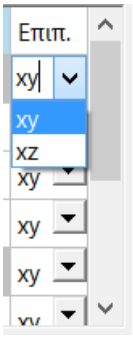


Connection A is in the XY plane which means that the metal connection plate will be inserted in this plane (vertical).
Connection B is in plane XZ and therefore members 1,2 will be connected with a horizontal metal plate.

Only members belonging to the same level can be connected to each other. Therefore, in connection B, for example, the members of the poles cannot be connected.

OBSERVATIONS:

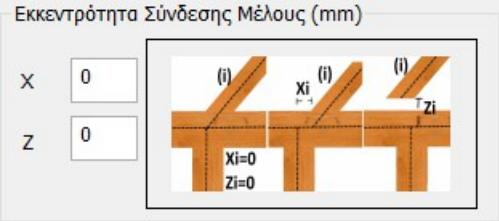
- The level of the connection defines the level of bending of the members to be taken into account according to the local axes of each member.
- Therefore, of the 6 intensive quantities (N,Mz,Vy,My,Vz,Mx) of each member, at the connection node, 3 of them, N,Mz,Vy in the xy plane, and N,My,Vz in the xz plane, will be taken into account.



The selection of the correct level of each member based on its local axes is defined in the Levels column.

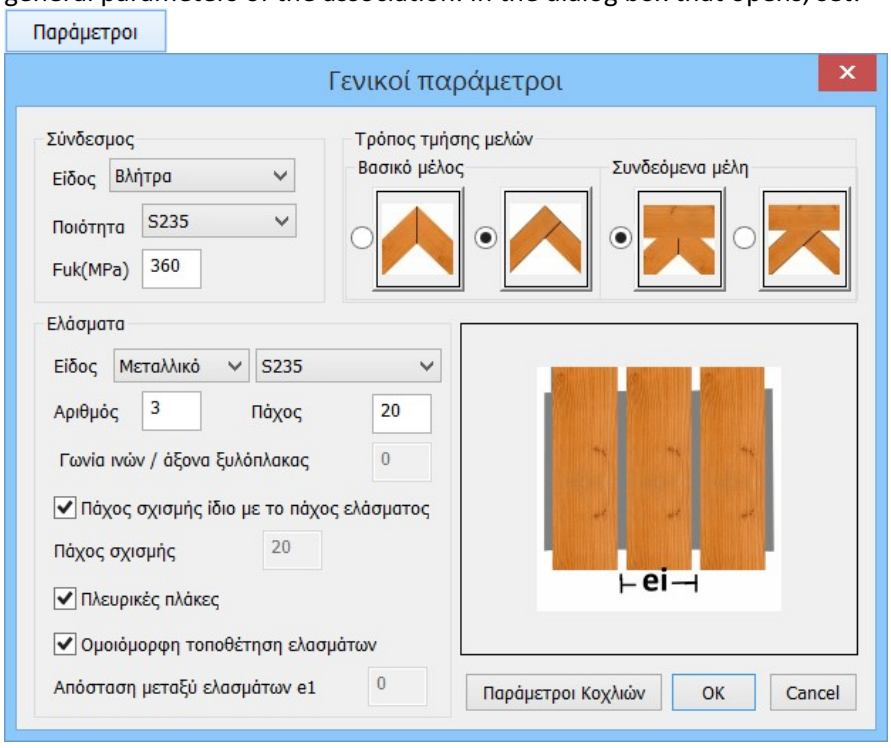
Member Connection Eligibility:

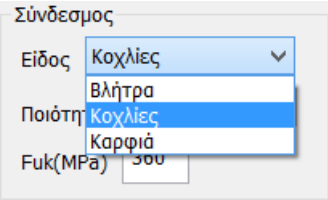
Through the eccentricity of a Connecting Member, the end of a connecting member can moved from the connecting node during eccentricity.
In this way, construction eccentricities are covered.



Select the member and according to the figure define the eccentricities by X and Z.

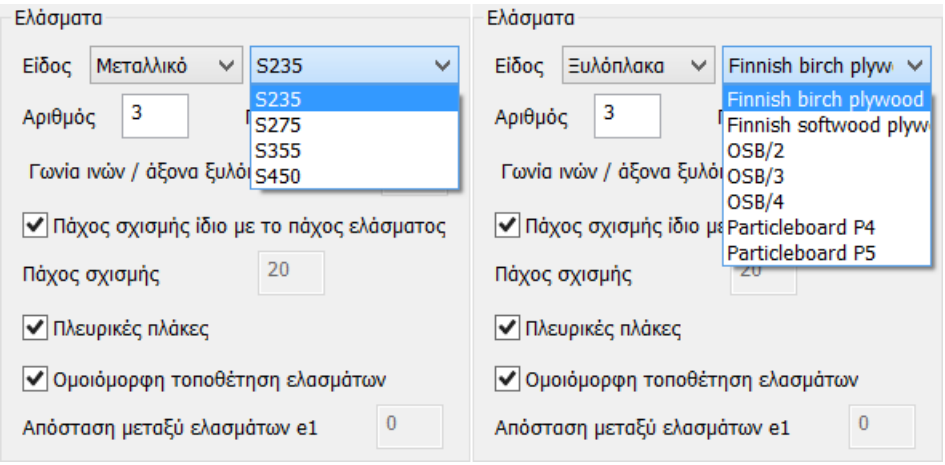
After completing the Membership Association, select the **Parameters** command to set the general parameters of the association. In the dialog box that opens, set:



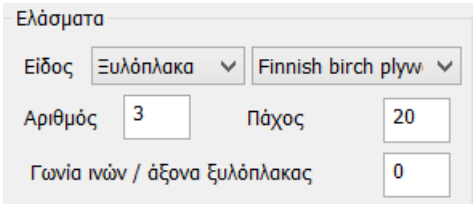


In the **Connector** field you define the **type** of connector by selecting from the list Bolts or Screws or Nails, the corresponding **Quality** and the value of the **Fuk** breakage limit is automatically updated. Alternatively, the designer can enter his own value for the breakage limit to be taken into account in the checks.

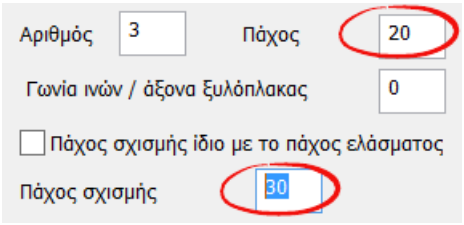
In the **Lamellae** field you set the parameters for the lamellae to be used in the connection by choosing between metal and wooden lamellae. In each case you select from the corresponding list the quality of the material, the *number* of plates and their *thickness.*

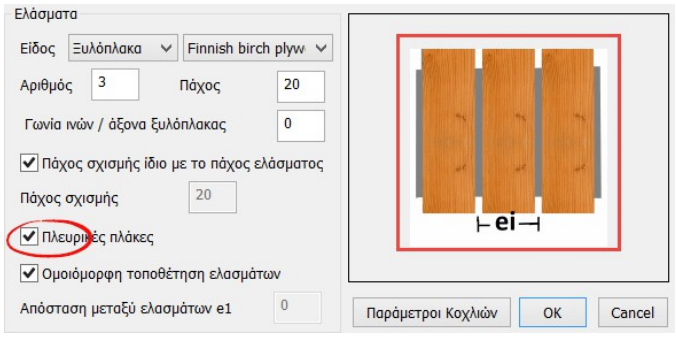


- In the case of the **Wood Slab**, the parameter *Fiber Angle/wood slab axis* is activated where the user defines the angle that the fibers of the wood slab have in relation to the axis of the main member.

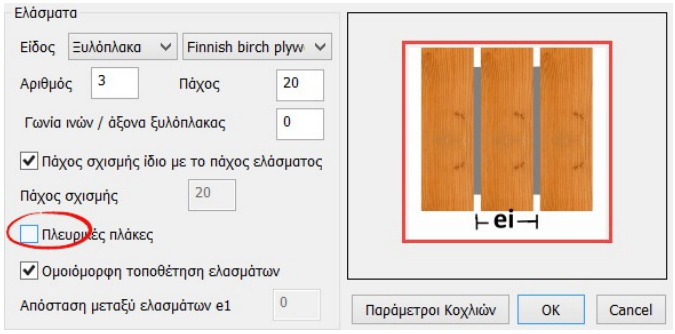


- In case the *Slit Thickness* is greater than the thickness of the laminate, uncheck the checkbox and set the Slit Thickness.

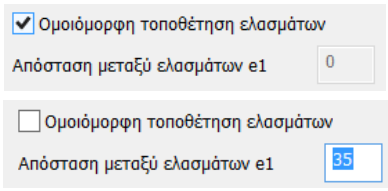




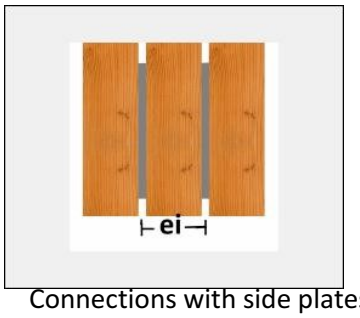
Checking the *Side Plates* checkbox activates the insertion of the side plates.



Otherwise there are only the intermediate plates.

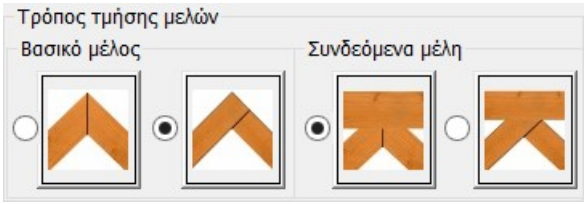


The placement of the laminates within the cross-section of the wooden member (by its thickness) can be *uniform*, i.e. dividing the cross-section into equal parts (active checkbox) or not.

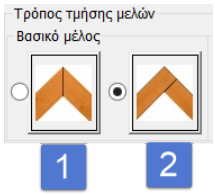


In the second case turn off the check and set the distance ei as defined in the figure.

- Connections with side plates require evenly spaced plates.

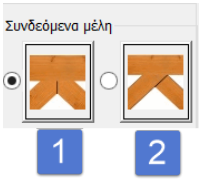


In the **Member segmentation mode** field you select the mode of segmentation of the member cross-sections.



The first 2 options involve trimming the Master member with the linked member having priority 1:

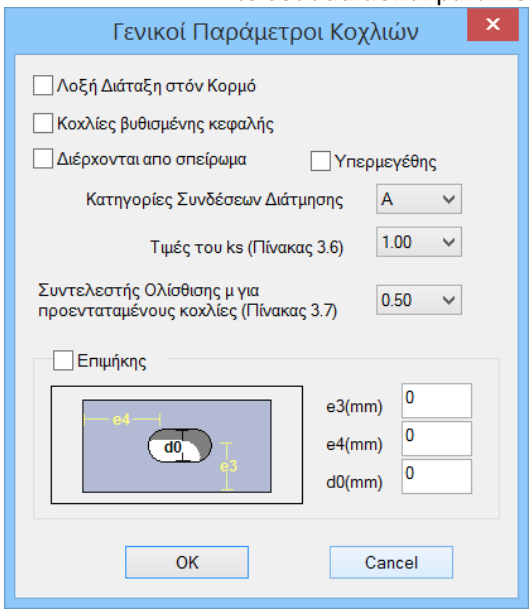
- the Main member is cut in half with the connected member
- the Principal Member shall prevail over the Associate Member.



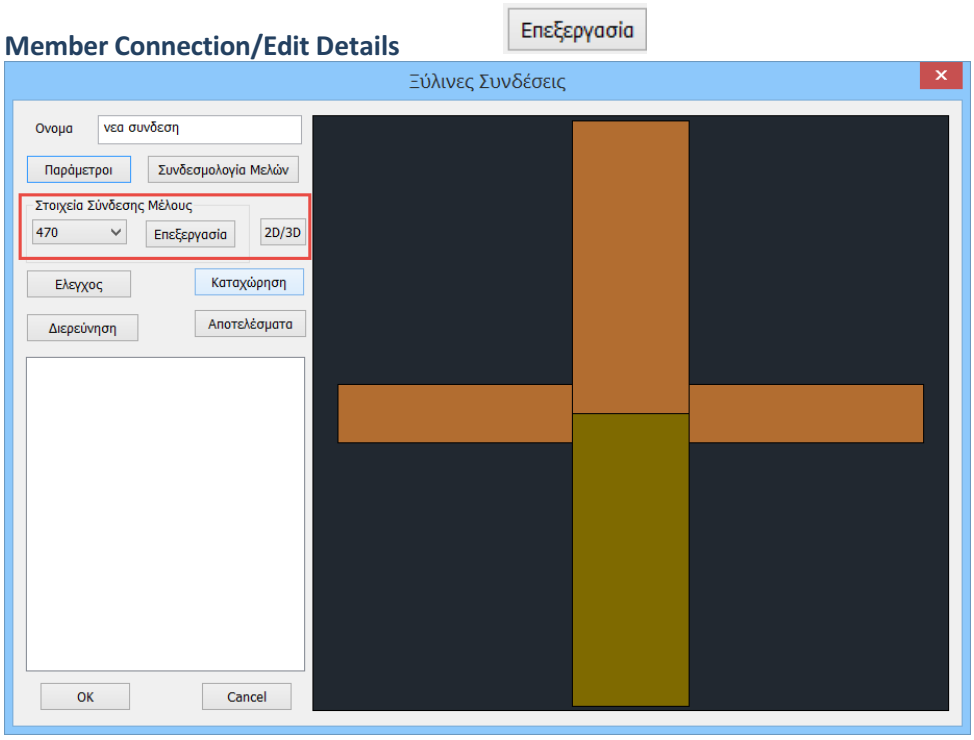
The other 2 options involve trimming the connected members between priority based:

- the connected members shall be cut in half
- the linked member with the highest priority prevails.

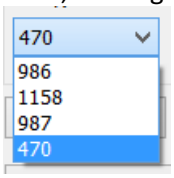
Παράμετροι Κοχλιών to set additional parameters related to bolts in a metal plate.



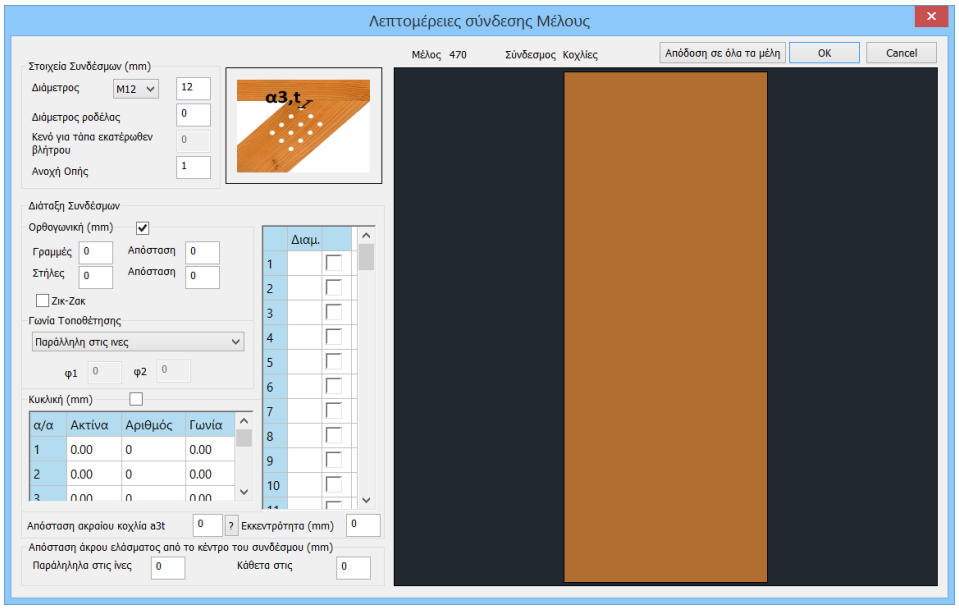
Member Connection/Edit Details

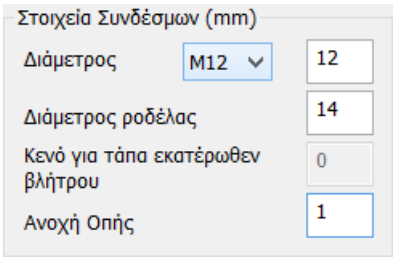


After linking the members and defining the parameters, the next step is to Edit the link.
You select the member, starting from the *Main*, or graphically, by pointing to it in the image on the left,

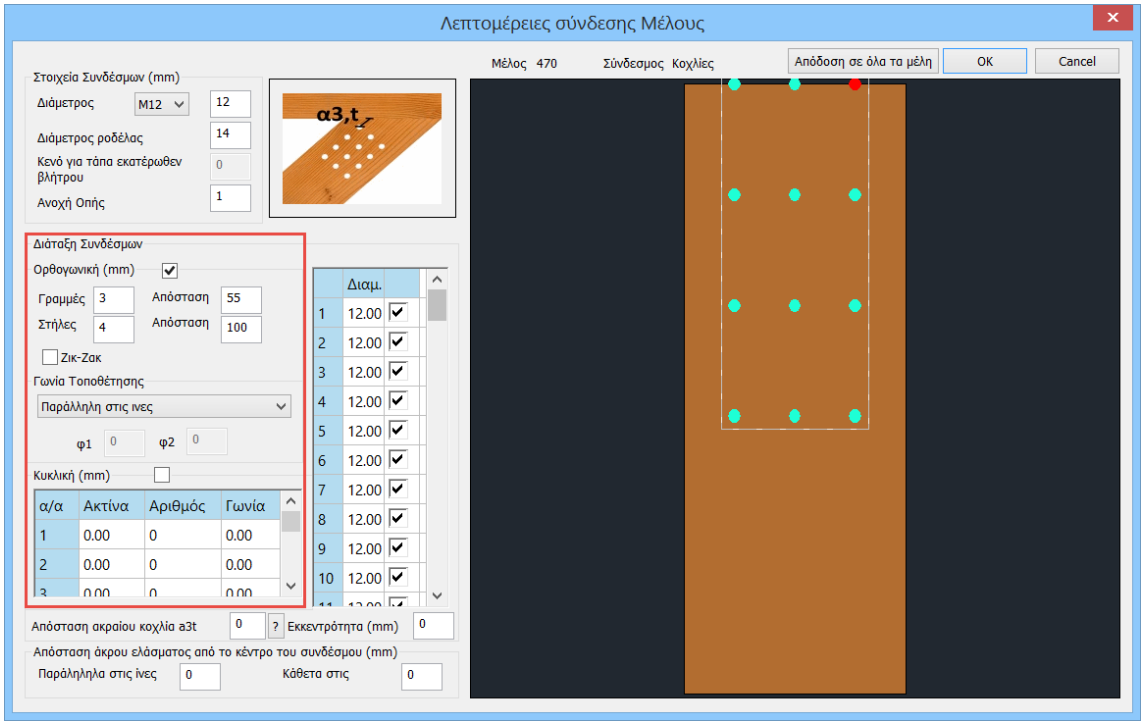


either from the list and the **Edit** command.





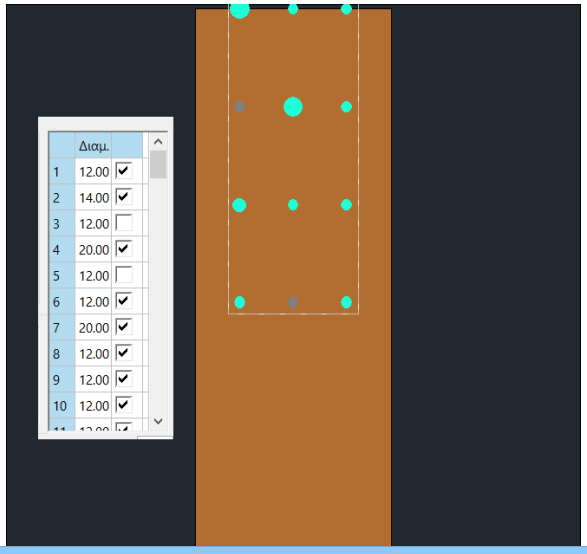
- Select the diameter of the screw (or set the diameter of the screw)
- Set diameter for the washer (only in case of screw)
- Set gap for plug (only in case of a screw)
- Give the tolerance of the hole in mm.



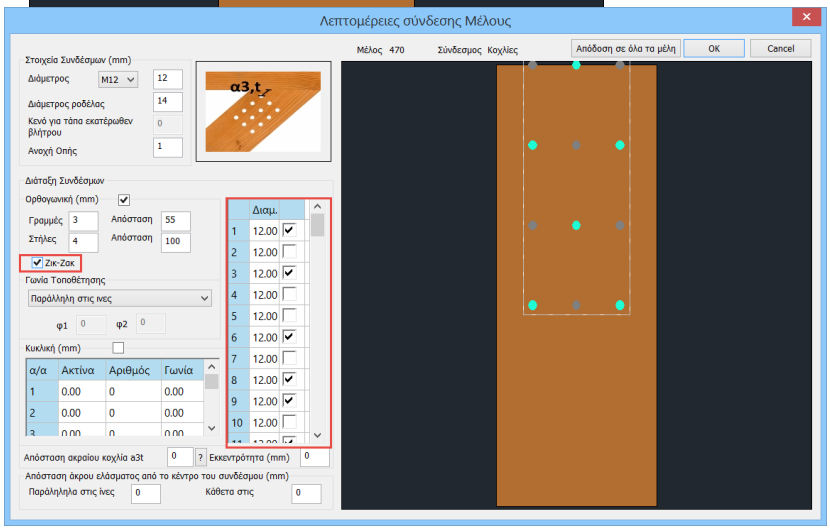
In the **Liaison Order** field

- With the *rectangular arrangement* active:

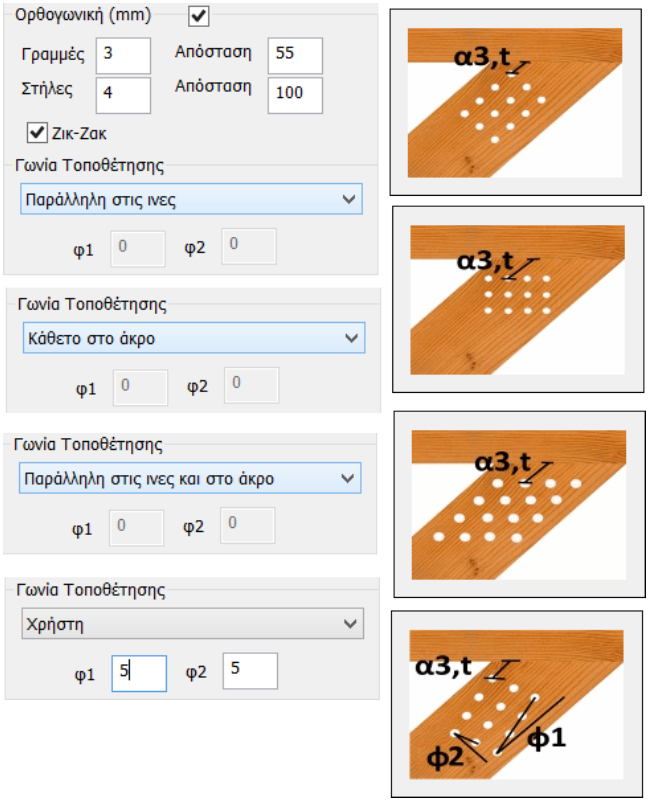
You define the number of Lines (parallel to the wood fibres) and Poles (perpendicular to the wood fibres) and the corresponding distances. The shape on the right is updated showing the links as well as the outline of the laminate.



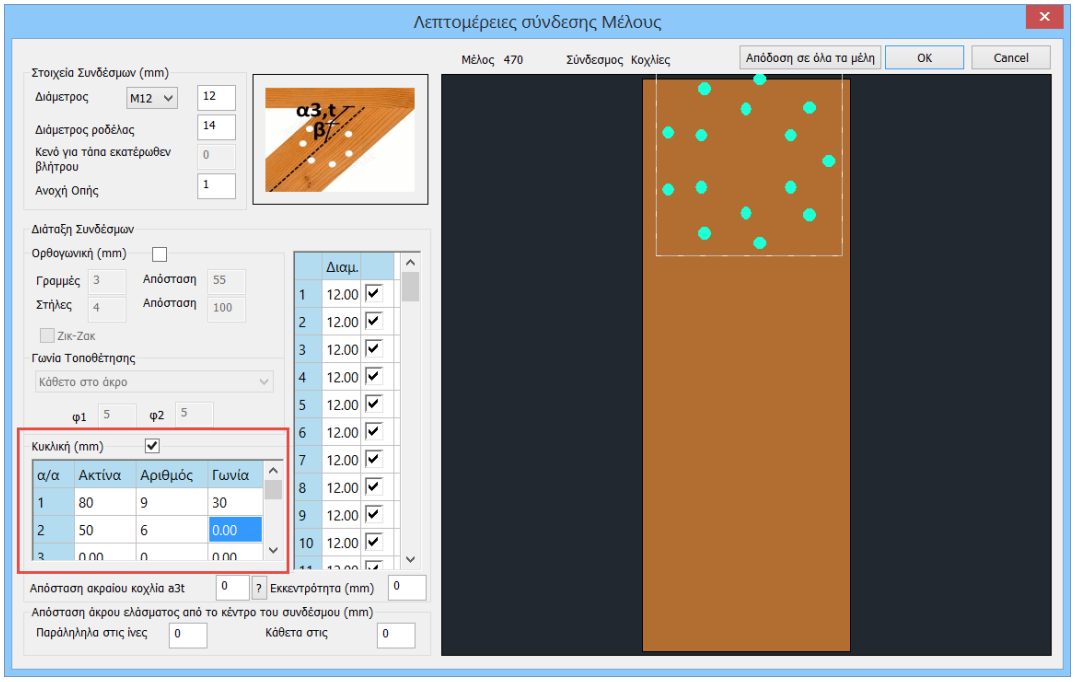
Through the column of links the user can modify the diameters by directly typing the new value of the selected link either graphically in the figure or in the column. It is also possible to exclude links by disabling the checks.



The Zig-Zag option excludes all intermediate links.



The **placement angle of** the links is selected from the list and displayed in the window.



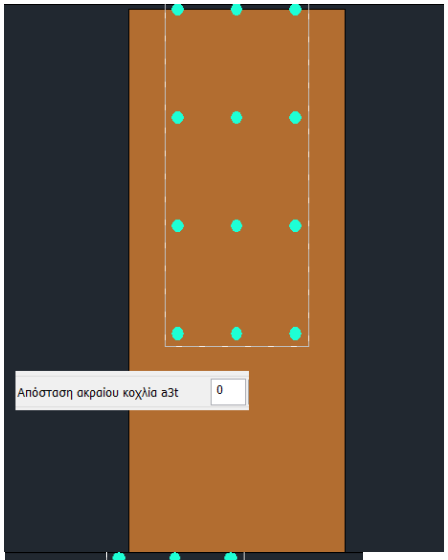
In the **Liaison Order** field

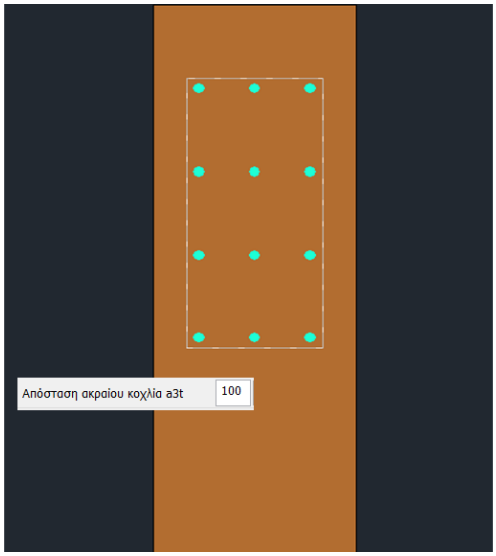
- With the *Circular Arrangement* active:

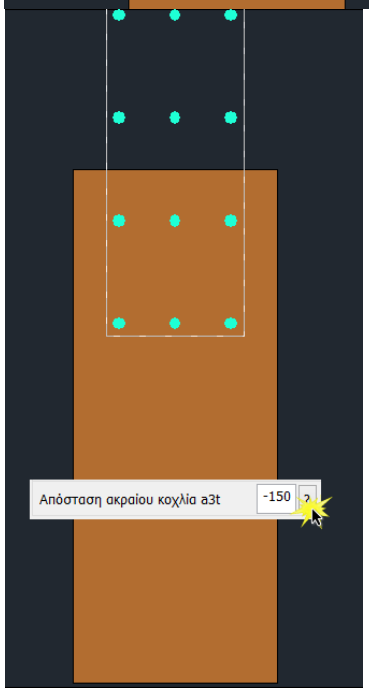
Complete the table by defining for each link circle the radius and the number of links. The angle rotates the corresponding circle from +x anticlockwise.

At the bottom of the remaining window you can choose to :

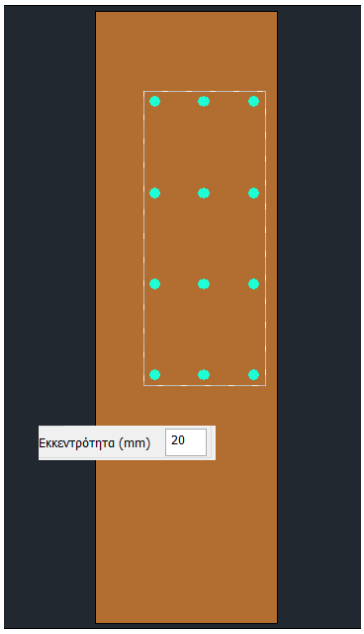
- Define the distance a3t (the distance from the end of the member the nearest link, parallel to the fibres of the member)





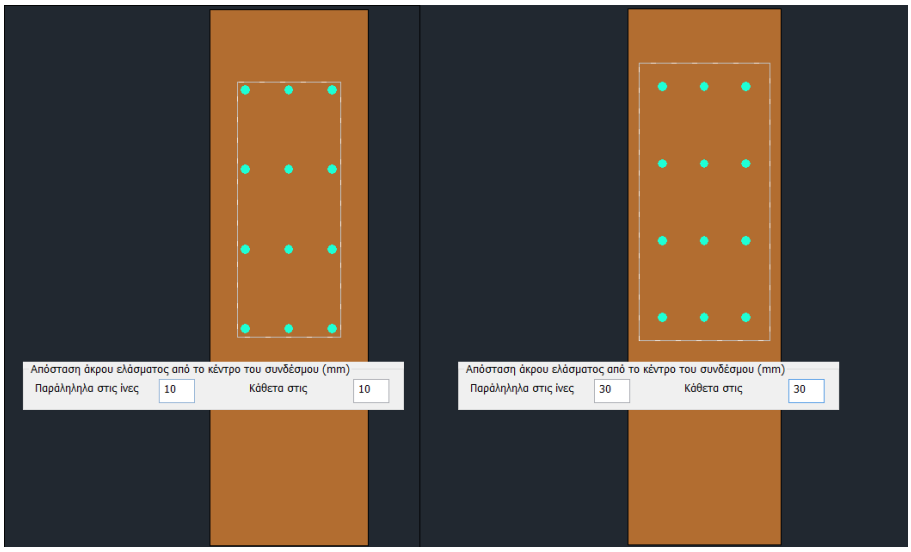


By pressing **?** the centre of the fastening is automatically moved to the end of the member.



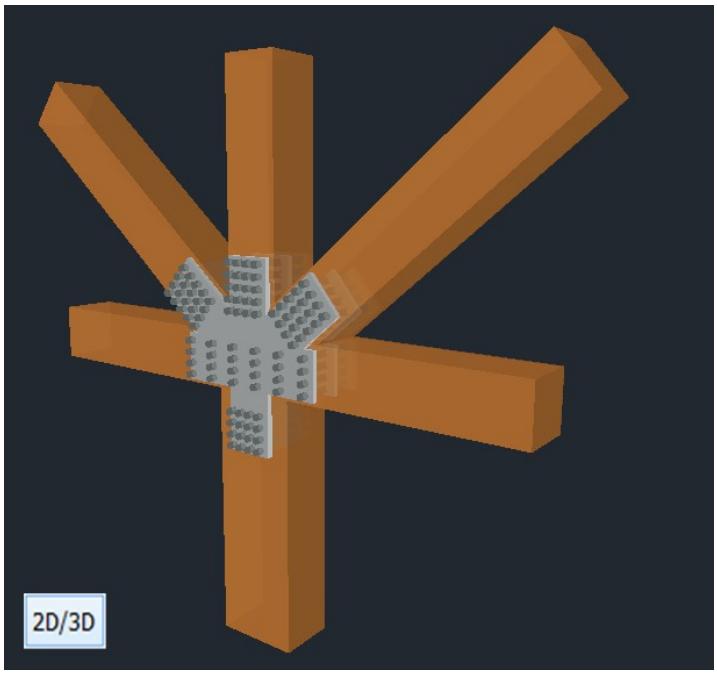
It carries the connectivity during eccentricity, perpendicular to the fibers of the member.

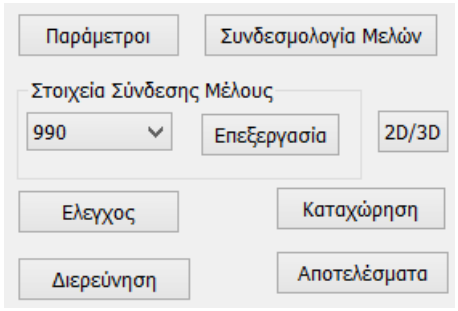
It modifies the distances perpendicular and parallel to the fibres of the two ends of the laminate from the centre of the ligament.

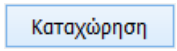


Απόδοση σε όλα τα μέλη by selecting this command, all the above member connection details are also attributed to the other members of the connection.
OK to save the options and close the window.

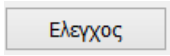
You can create complex links with a large number of members and display them in 3D with the 2D/3D option.



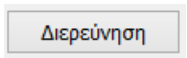




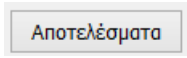
: saves the connection



: performs the required checks according to EC5 (for wood) EC3 (for steel)



: displays the results of the checks in detail



: displays the results of the checks in tabular form

CHAPTER 10C 'DIMENSIONING'