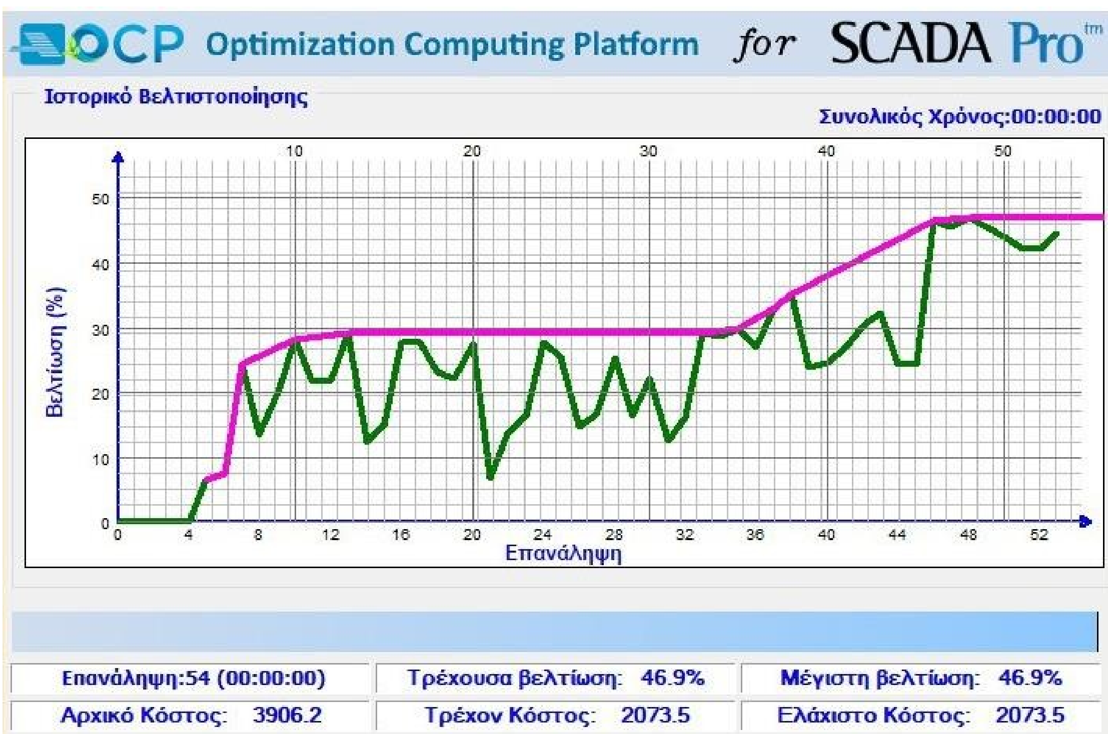
SCADA Pro 25™
Structural Analysis & Design

User's Manual
13. OPTIMIZATION



ACE HELLAS
INTEGRATED SOLUTIONS

CONTENTS

Scada Pro OCP

The new innovative SCADA Pro OCP software is a module of SCADA Pro and is a computational platform for the general optimization of realistic scale construction for civil engineering structural systems.

The main development objective of SCADA Pro OCP is to **minimize construction and material costs,** resulting in the required performance, reliability, quality and safety of the structural system within an innovative technological framework.

The SCADA Pro OCP module has:

- **Advanced and** easy-to-use command interface, making the construction optimization process a one-click affair.
- The possibility to **select multiple criteria** related to the cost of construction, such as construction costs, the cost of construction materials, environmental costs over the lifetime of the construction, etc.These criteria are used either as "Objectives" or "Constraints", which are taken into account in addition to those imposed by the regulations.
- Solving the problems by selecting from a list of state-of-the-art **deterministic and probabilistic numerical optimization algorithms** that can replace the traditional test and correction planning process by means of an optimized solution obtained quickly.
- Multiple options for defining the range of **design variables** and grouping them either at **the section level or at the component level.**
- **Comparison of the original design with the optimal one**.

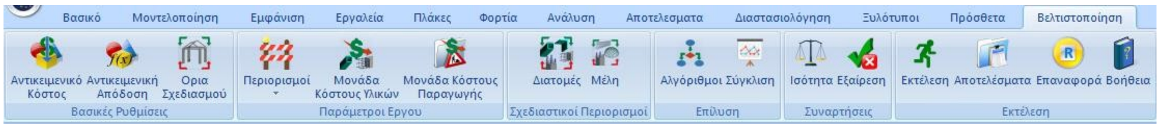
OBSERVATIONS:

The new innovative SCADA Pro OCP software, with the main objective of minimizing the cost of construction and material costs, offers us the possibility to re-examine our already adequate carrier, using smaller cross-sections, always consistent with the required performance, reliability, quality and safety of the structural system.

With SCADA Pro OCP you can size operators that have inefficiencies in order to resolve them. In these cases SCADA Pro OCP will seek to find the most cost-effective solution that offers adequacy to the operator!

Optimization

The "Optimization" section is about defining the parameters, executing and displaying the results of the optimization process. The commands are grouped into modules according to the type of function they perform.



As pointed out in the theoretical manual, the optimization process is performed in two phases:

- In the first phase the Basic Settings, the Project Parameters and Design Constraints are defined.
- In the second phase, the Optimisation Algorithm is defined and the Iterative Computational Process is performed.

There are also some additional functions and features using which the design variables can be further specialised.

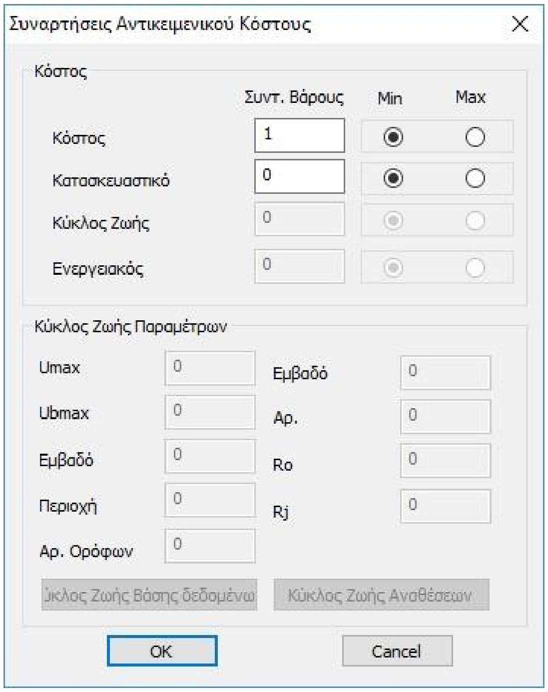
1.1 Basic Settings

The first command subsection includes the following 3 commands:

- Objective Cost
- Objective Performance
- Design Boundaries

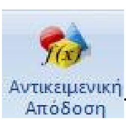
1.1.1 Objective Cost

Using this command allows you to select the objective function and/or a combination of objective functions on which to base the optimization process.
These functions are defined in the dialog box that appears:



where the active functions are Material Cost and Construction Cost.
The value of the Severity Coefficient takes values from 0 to 1 and determines whether the criterion will work towards minimum or maximum Optimization.
The sum of the gravity coefficients in the two fields must be unity.

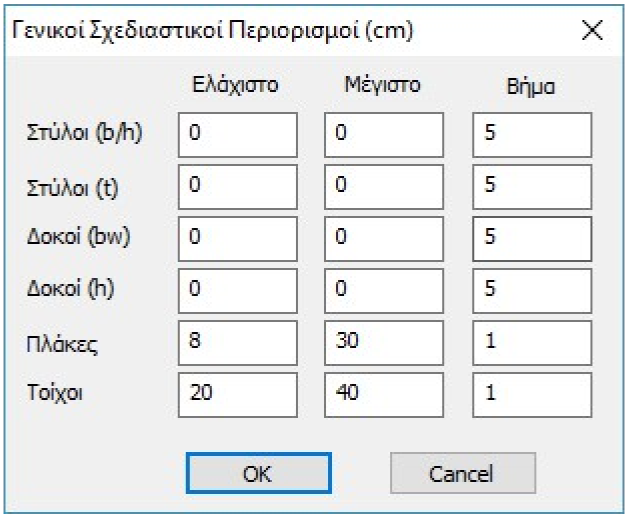
1.1.2 Objective Performance



The options of this command will be enabled in a later version of the program.

1.1.3 Design Boundaries

So using the "**Design Constraints**" command displays the following dialog box


Γενικοί Σχεδιαστικοί Περιορισμοί (cm)

	Ελάχιστο	Μέγιστο	Βήμα
Στύλοι (b/h)	0	0	5
Στύλοι (t)	0	0	5
Δοκοί (bw)	0	0	5
Δοκοί (h)	0	0	5
Πλάκες	8	30	1
Τοίχοι	20	40	1

OK Cancel

where the general Minimum and Maximum dimensional limits and the Step of change of these dimensions defined for each type of structural element.

ATTENTION!

- The above dialogue box only applies to reinforced concrete elements.
- The limits are defined for each type of structural element (Columns, Beams, Slabs and Walls).
- For poles, two "categories" of dimensions are defined:
  The first one (b/h) which concerns all large dimensions, regardless of the type of cross-section (rectangular, Gamma, Tau, etc.).
  The second dimension (t) refers to the corresponding small dimensions, mainly thicknesses.
- For the beams, the limits of their two basic dimensions defined, i.e. width (bw) and height (h).

The value limits for the thickness of the plates are then set. This includes conventional plates as well as those simulated with finite surface elements.

Finally, in the "Walls" section, the boundaries of the vertical elements simulated with finite surface elements are defined.

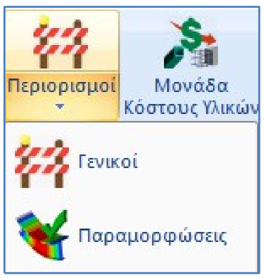
⚠ OBSERVATION:

The value 0 in the maximum and minimum limit fields means that the program takes as lower limit the original value of the element's dimension decreased by 30% and as upper limit again the original value increased by 30%. This variable applies only to the dimensions of beams and columns.

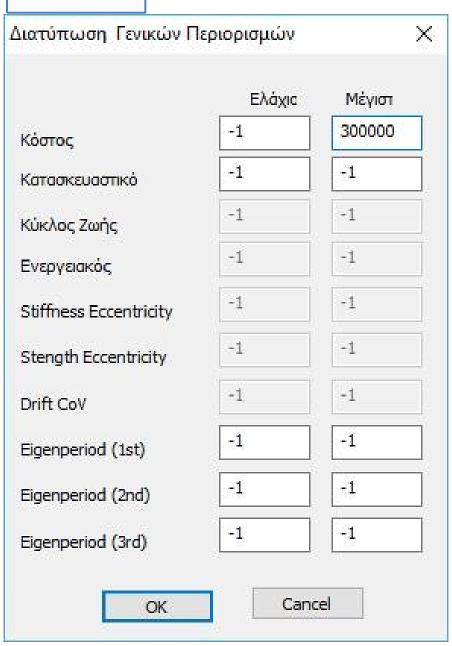
1.2 Project parameters

The next subsection deals with the project parameters and includes constraints imposed by the designer and the definition of material and production cost units.

1.2.1 Restrictions



Using the '**General Restrictions**' command displays the following dialog box:

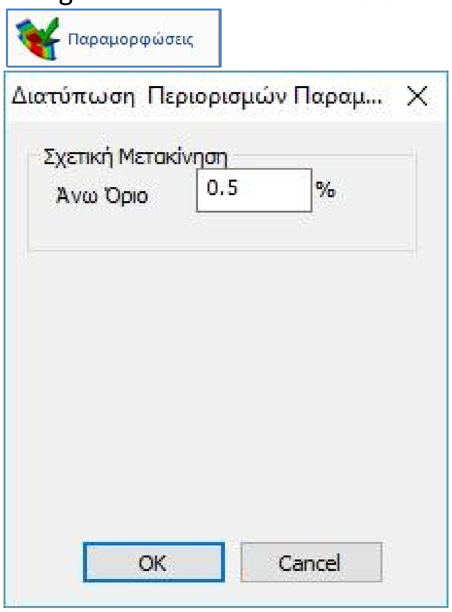


where you can specify a minimum and a maximum material cost and the construction cost of your carrier, as well as a minimum and maximum limit for the eigenmodes of the first three eigenmodes.

OBSERVATIONS:

- During the execution of the optimization process, the above General Constraints will be taken into account, in addition to the other geometric constraints.
- A value of -1 in the above fields means that the corresponding restriction does not apply.

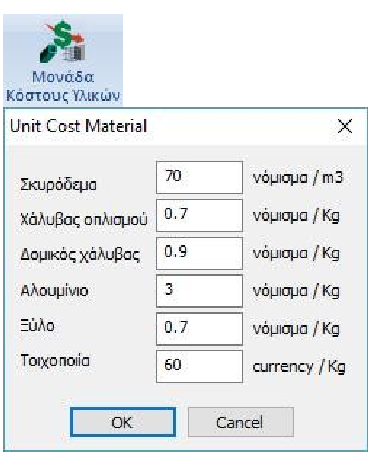
Using the next command "**Deformations**", in the dialog box that appears



specify an upper limit to the relative movement (drift) of the floors of the carrier. The resulting optimal solution will also obey this constraint.

1.2.2 Unit Cost of Materials

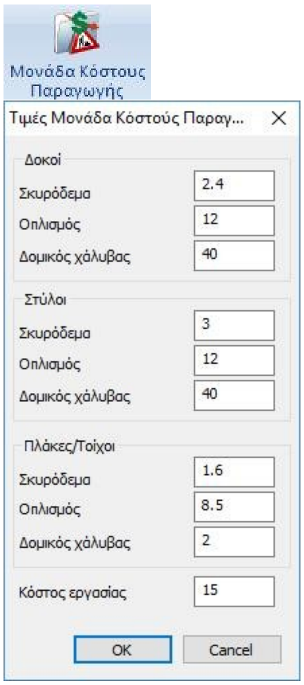
Using this command allows you to specify the cost per unit of material for the different materials supported by the program. The materials are shown in the following dialog box



These values are used to calculate the total cost of the construction materials.

1.2.3 Unit Cost of Production

By using the above command you can specify the following



the labour hours of production (hours/unit of production) per building element and per material. In the last field "Labour Cost" you enter the labour cost per hour.

1.3 Design Constraints

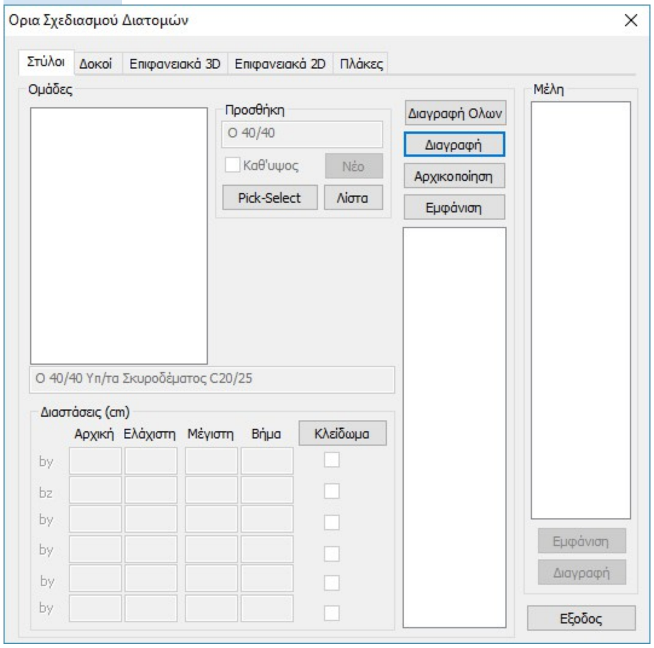
Design Constraints in the Optimization Process.

The way of imposing design constraints in the optimization process (OCP) of structures analyzed and designed in Scada Pro, follows a logical hierarchy that starts from general - total constraints applied to each category of structural elements (columns, beams, plates), continues to a more specific level of selection of cross-sections with the structural elements they include, to reach the level of grouping of individual linear or surface elements. These constraints concern either minimum and maximum dimension limits and their step of variation, or the "locking" of dimension or dimensions, i.e. to keep them (the dimensions) unchanged regardless of whether they belong to a type of cross-section or to specific structural elements.
More specifically, the first level of defining the boundaries within which the dimensions of the building elements will take values is done using the "Design Boundaries" command which belongs to the Basic Settings subsection.

The next two commands make up the Design Constraints subsection.

1.3.1 Cross sections

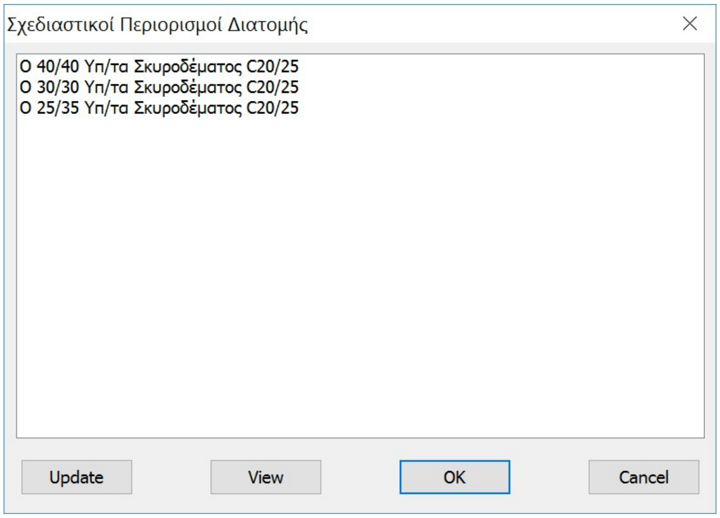
The first command "Sections" is about defining boundaries and restrictions at the Section level. Selecting this command displays the following dialogue box



The import and processing of cross-sections is done by type of structural element.

1) So in the first section **"Pillars"**, the way of inserting the cross-sections can be done two ways:

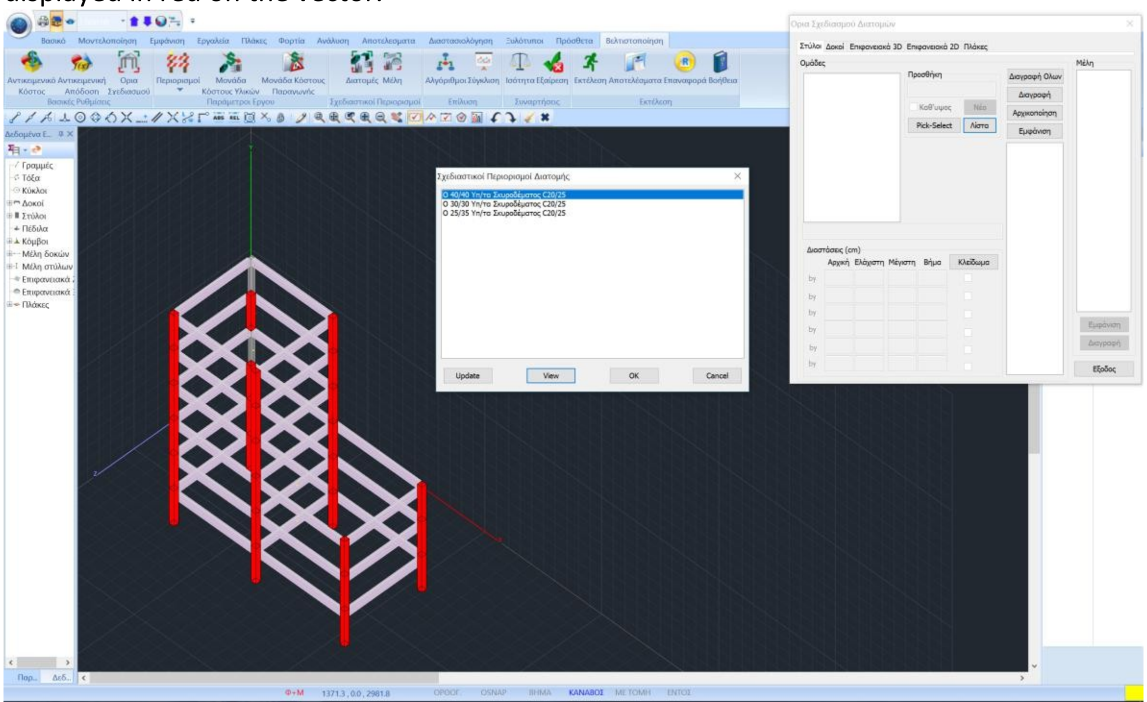
With a choice of List and graphics.
The graphical selection is made with the "Pick-Select" button.
Click on the "List" button to display the following dialog box



where all the different cross-sections included in the carrier are shown. The criteria for differentiating the cross-sections are:

- The type of cross-section
- To Layer
- The Quality of the Material

The selection is made for one or more cross-sections. After selecting the cross-section by pressing the "View" button, the structural elements that have this cross-section are displayed in red on the vector.



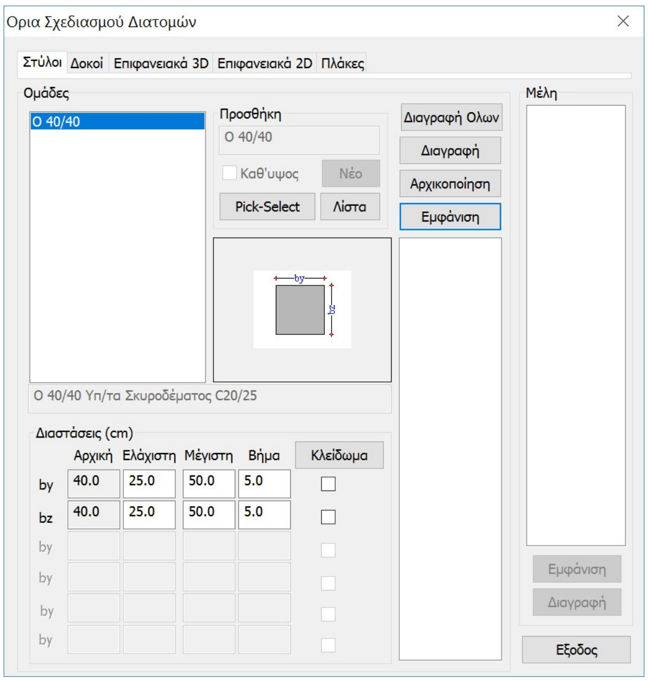
By selecting the "Update" button, the list of cross-sections is automatically updated if changes have been made to the vector after the first display of this list.

By pressing the "OK" button the selected cross-section or cross-sections are inserted in the "Groups" field.

The other way to select the cross-sections is by pressing the "Pick-Select" button and then selecting a physical cross-section of a pole with the mouse. Then this cross-section is also entered in the "Groups" field.

ATTENTION!

The option here applies only to physical cross-sections of poles, therefore the display of physical cross-sections must be activated in the 3D model



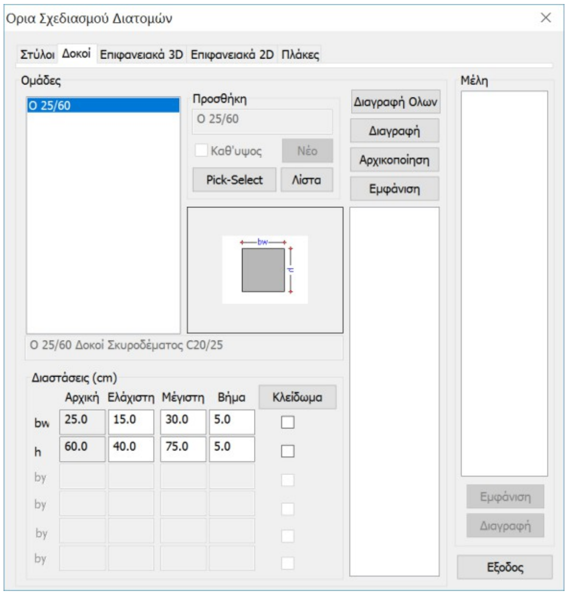
By selecting the corresponding group, the dimensions of the cross-section are displayed in the "Dimensions" field according to the graph. More specifically, the first column contains the initial dimensions, the second the minimum dimension limit, the third the maximum dimension limit, and the fourth the step of variation.

The "Lock" column locks the dimension in order for it to remain unchanged at its original value. By pressing the "Lock" button all dimensions are checked, i.e. the whole cross-section is locked. The default values in the boundary and Step columns are those defined in the General Design Parameters and which all elements initially obey.

By using the "Delete All" button, all the cross sections that have been defined are deleted, while by using the "Delete" button, the selected cross section is deleted. Using the "Initialize" button, the original limits are restored to the dimensions of the cross-section.

Finally, by selecting the "Show" button, the elements that have the specific cross-section are displayed in red.

2) For the "**Beams**" section

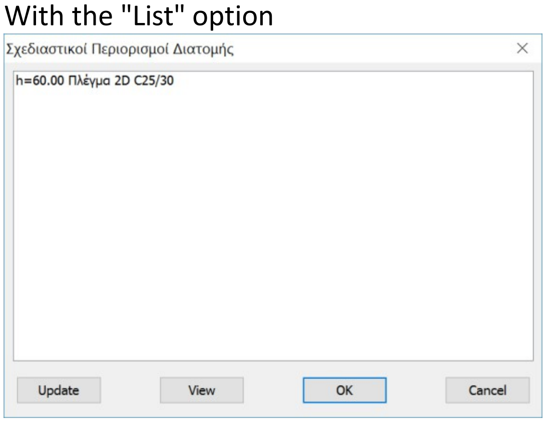


exactly the same as those of the pillar module apply.

3) The next two sections deal with finite surface element cross sections **"Surface 3D"** and **"Surface 2D"**. The mode of operation is the same in both cases.

So by selecting the "3D Surfaces" section you can add surface cross sections either from the List or Graphics.

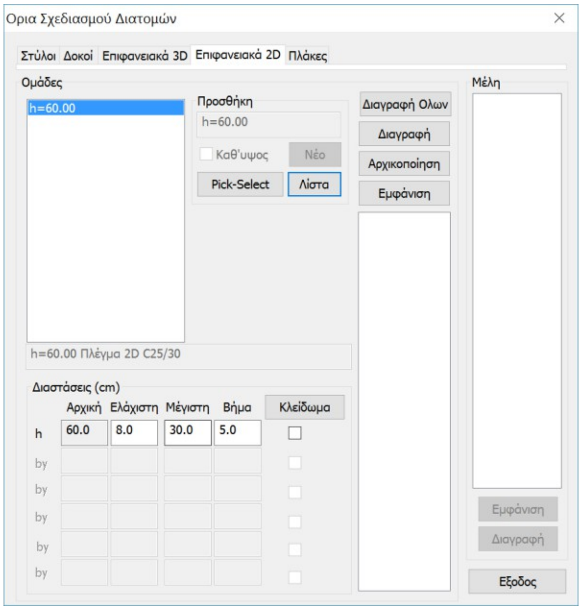
With the "List" option



the list of different cross-sections of the surface is displayed. The criteria for categorising the cross-sections are as follows:

- The thickness
- The material
- The Layer
- If it is horizontal or vertical By

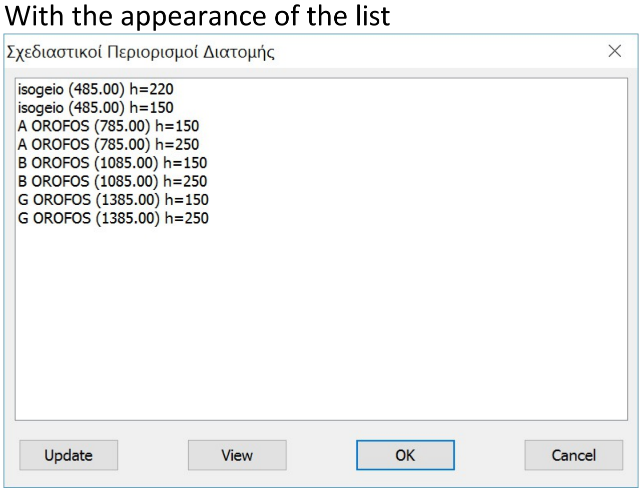
entering the cross-section



the initial thickness, the lower and upper limit and the step of change are displayed in the dimension section. Finally there is an option to lock the specified thickness.

4) The last section **"Slabs"** concerns conventional slabs. The choice here can be made in two ways.

With the appearance of the list

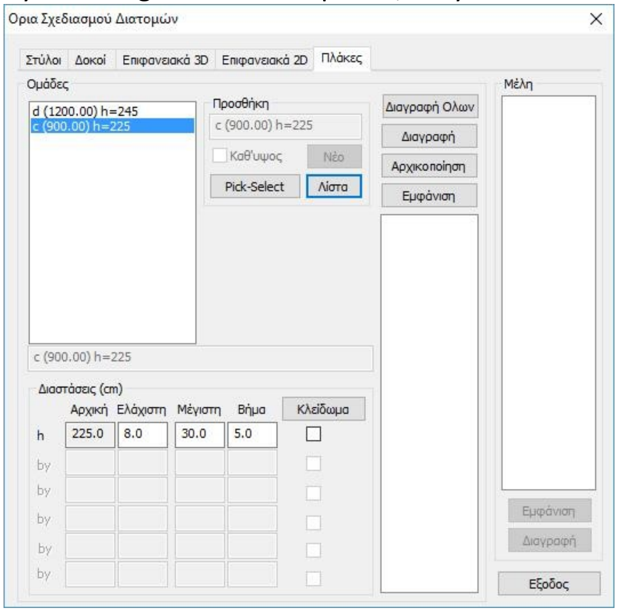


where it includes all plates of the construction. The classification of cross-sections is based on two criteria:

- The floor
- The different thickness

In the list the name of the floor is given, then in brackets the elevation of the level and finally the thickness h of the slab.

By selecting one or more plates, they are inserted into the group box.



The selection of the plate or plates can also be done with the "Pick-Select" button. For selection you need to rotate the vector in 2D.

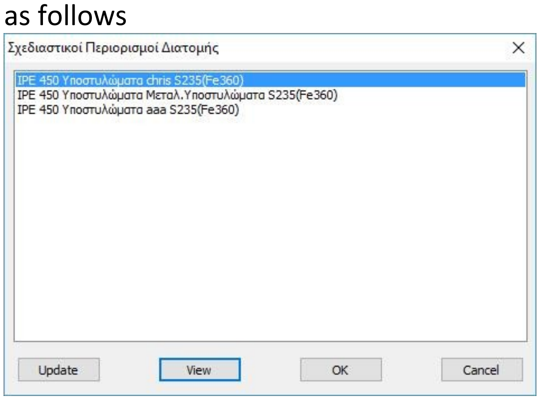
Then, by selecting from the "Groups" field the plate you wish to edit, in the "Dimensions" section the initial thickness, the minimum and maximum thickness as well as the option to lock the thickness of the specific plate are displayed.

Metallic cross sections

For metallic sections, the same applies as for reinforced concrete sections, as far as the definition of groups is concerned.

EXAMPLE

For example, for metal poles the corresponding list that appears using the "List" command is as follows

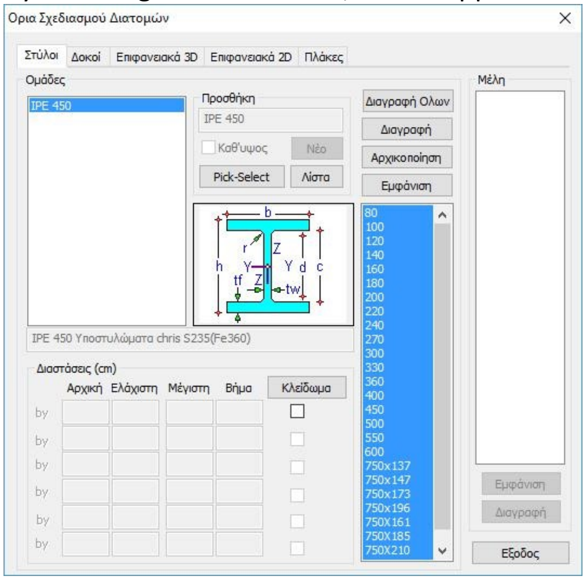


and includes all types of cross-sections included in the carrier. The differentiation is also made here on the basis of the following criteria:

- Type of cross-section
- Layer owned by

- Quality of Material

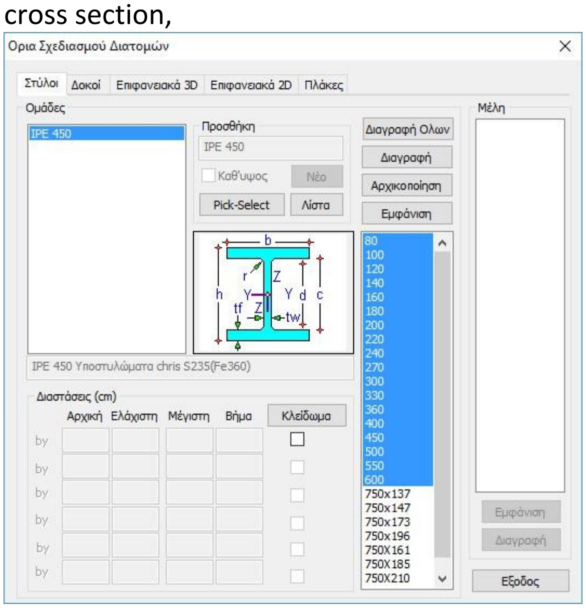
By selecting a cross-section, it now appears in the groups field.



The image of the cross-section with its dimensions is also displayed, as well as the list of cross-sections of the specific type of cross-section. The blue colour in the cross-section table means that the cross-section can be changed in the full dimensional range of the specific cross-section type.

EXAMPLE

For example, in the image above, the IPE450 cross-section has been selected. The full range of IPE cross-section dimensions available in the library has been made blue which means that the cross-section can be changed over the full range shown in the table. Here you can also either select a smaller range by clicking, with the Shift key held down, on the first and last cross section,



or by selecting compared cross-sections by holding down the Ctrl key.
Finally, the following commands apply to all modules and apply to all sections:

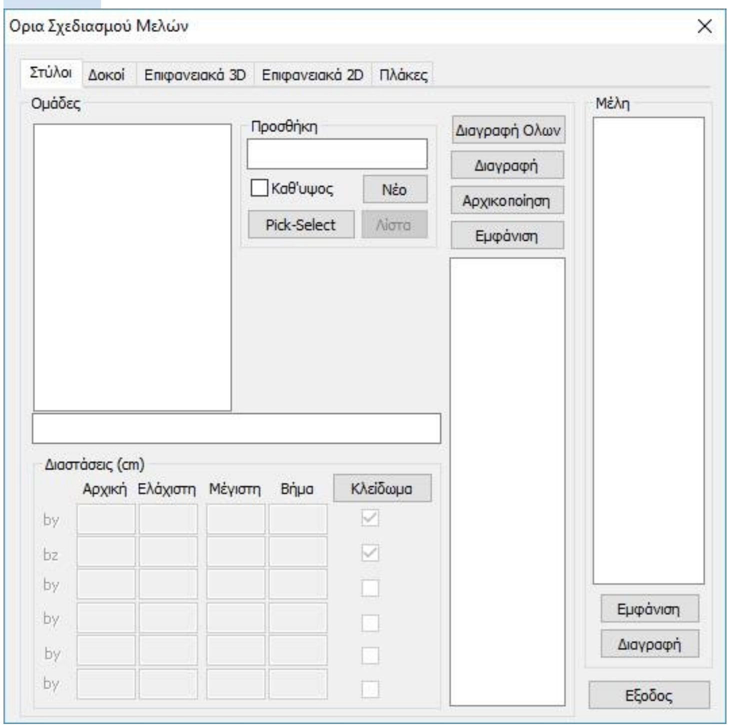
- The "Delete All" command deletes all groups that have already been entered.
- The "Delete" command deletes the specific group you have selected.
- The "Initialize" command restores the limits to their original values as specified in the General Parameters.
- Finally, the "Show" command displays the selected items in red.

Finally, it is worth noting that by using the "Pick-Select" command, if you select a different item type from the one in the section you are in, the application will automatically go to the corresponding item type.

1.3.2 Members

The next command to define a group or groups of members in order to set restrictions at the level and individual member level.
By selecting the "**Members**" button

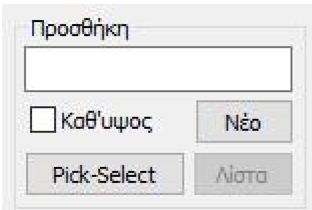
the following dialog box appears



where you can create groups of elements for which you want to impose specific design limits or lock dimensions.
The structure of the dialog box is the same as that of the cross-sections, i.e. divided into sections according to the type of element.

The logic in all modules is the same:
You start by defining a group or groups of members that must have the same cross-section.

1) In the "**Pillars**" section you start by defining the name of the group you are going to create. The group can contain one or more individual members which must have the same initial dimensions and will obey specific lower and upper bounds.



After entering the name you press the "New" button to create the group and with the "Pick-Select" option you graphically select the members you want to join this group. The cross-section of the member of the first column you show will determine the cross-section of the group you create.

EXAMPLE

For example, if the cross-section of the first pole is 40/40 all subsequent poles must have this cross-section and the same material quality and belong to the same layer. You can choose either the physical cross-section or the mathematical member. The element type must be the same as the corresponding module you are in.

OBSERVATION

⚠ If you do not enter a name for your group and simply press "New" the program first puts English question marks "???" as the name and then names the group based on the cross-section of the first element you select. Of course you can modify the name of the group.

So by selecting the items these appear in the corresponding list

Here you can delete one or more items by selecting them (one at a time) and pressing the "Delete" button. You can also display them in the vector by pressing the "Show" button.

⚠ Defining the elements of a group can be done either all at once the first time, or by adding elements to an existing group afterwards. You select the existing group from the list and use the "Pick-Select" command to select the elements you want to add.

With the option ☐ Καθ'υψος when checked, all the poles of a column row can be automatically selected by graphically selecting any of them. This procedure allows only one element to be selected.

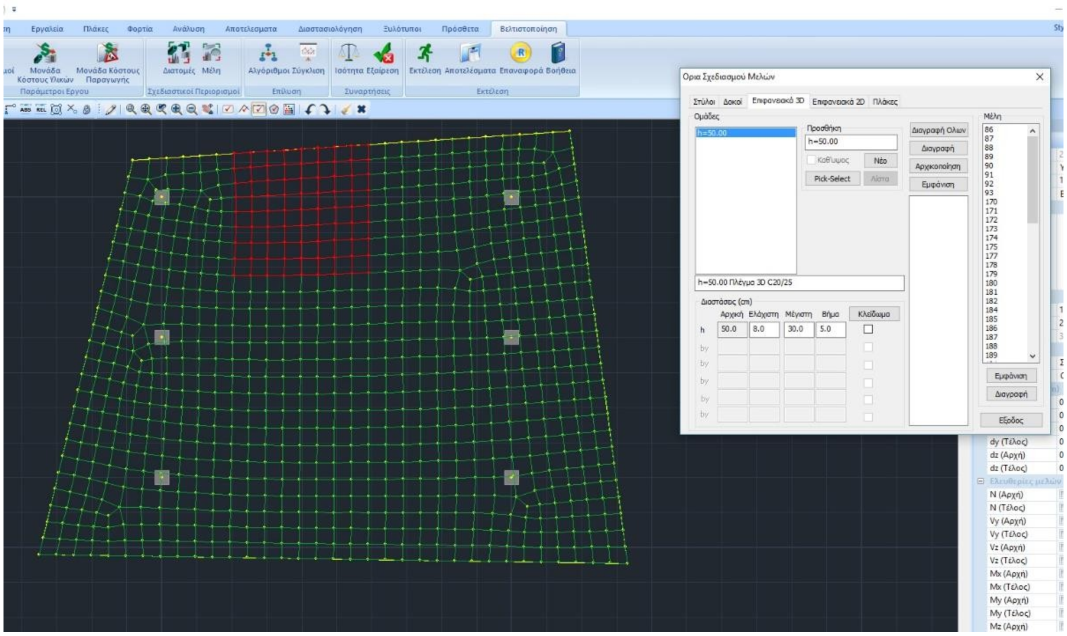
For the dimensional domain

apply the same as described above for the cross-section dialogue box.
Finally, the following commands apply to all modules and apply to all sections:

- The "Delete All" command deletes all groups that have already been entered.
- The "Delete" command deletes the specific group you have selected.
- The "Initialize" command restores the limits to their original values as specified in the General Parameters.
- Finally, the "Show" command displays the selected group items in red.

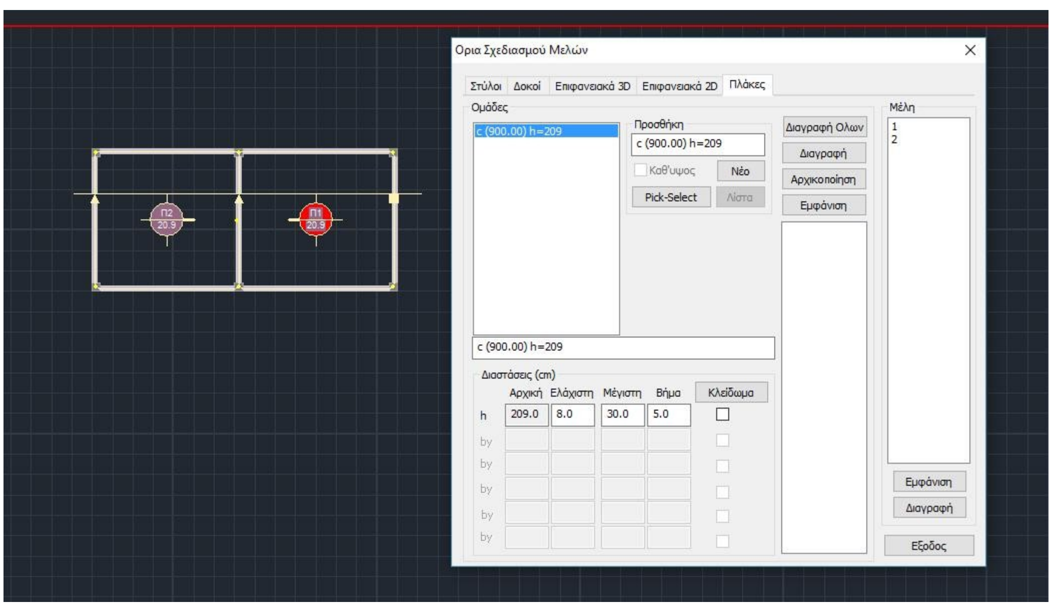
2) For the "**Beams**" section, exactly the same applies as previously mentioned for the poles.

3) For the next two sections **2D** and **3D Surface** the following applies:
The group or groups you create can consist of one or more surface elements.
It is recalled that in the surface the only parameter that can be determined is the thickness.
So you define a new group and with the graphical selection tools (individual, windowed, etc.) you select the surfaces that will make up the group you create.



The surface elements can belong to different groups or subgroups but they must have the same initial thickness, the same material quality and belong to the same layer. Upon selection they appear in the list of members.

In the slabs section you create a group where you include slabs that have the same thickness and belong to the same floor.

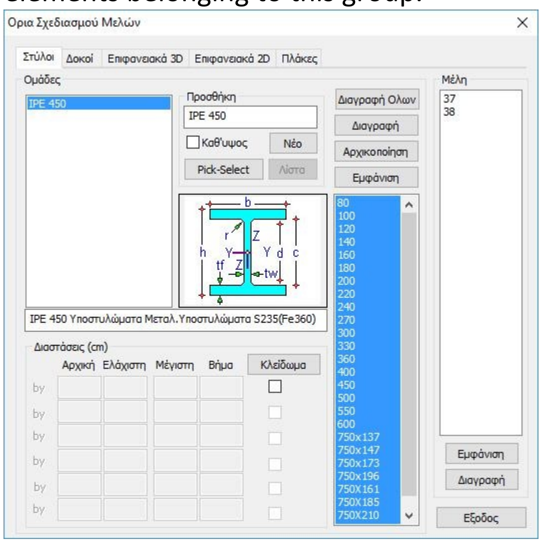


EXAMPLE

For example, the image above shows the creation of a group of 209 mm thick slabs belonging to level c with an elevation of 900.00 cm and includes two slabs (1,2) as shown in the list of members.
Finally, note that in order to display the plates and make their selection graphically, you need to go to the corresponding section of the plates.

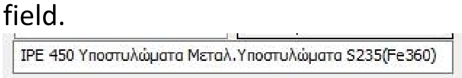
Metallic cross sections

As far as the Steel Sections are concerned, creation of groups is done for both beams and columns, in the same way as for reinforced concrete sections.
The selection of the first element determines the cross-section, material and layer of elements belonging to this group.



EXAMPLE

For example, in the image above a group was created containing the members of pillars 37 and 38 which are of IPE 450 cross-section, belong to the Metal layer. Subcolumns and their material is s235(Fe360). All this information is also shown in the line above the dimension field.



So all of the above tools and commands give you the ability and flexibility to specify design constraints on as many members and in any way you wish.

1.4 Resolution

Two commands are included in this section:

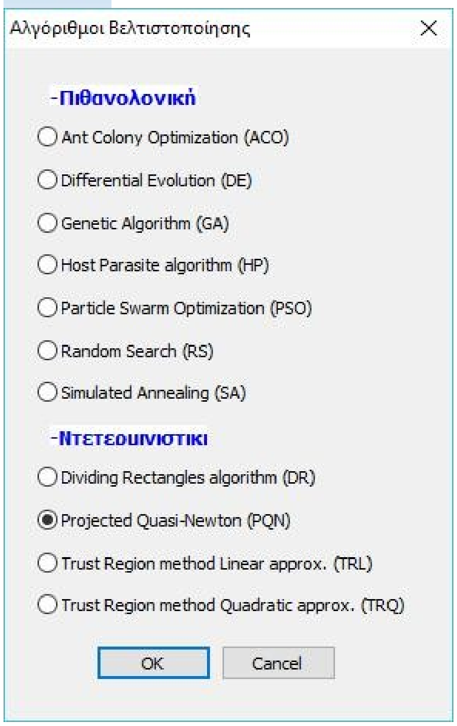
Algorithms, where you select the Algorithm to be used for the optimization process

Convergence, where you specify the convergence criteria to be used as well as the analysis and sizing scenarios.

1.4.1 Algorithms

By selecting the "Algorithms" command

The following dialog box appears

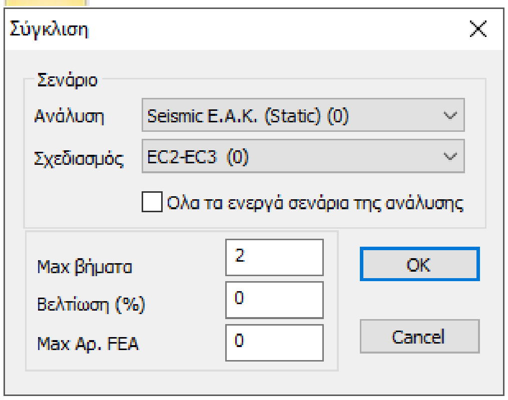


where you can select the optimisation algorithm from two categories

- Possibles
- Determinist

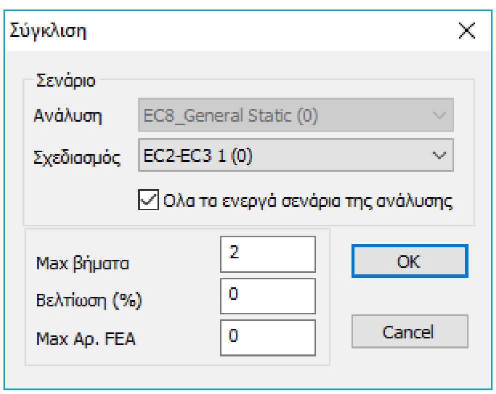
1.4.2 Convergence

Selecting this command displays the following dialog box



Built into the program the ability to run sequential analysis scripts when running OCP.

In the corresponding dialog box where we select the analysis and sizing scenario to be used in OCP, the option "All active analysis scenarios" was added. When this option is checked, the "Analysis" field is disabled and OCP sequentially runs all ACTIVE analysis scenarios. An active scenario is one that is not cancelled (no asterisk).



Two observations:

- If we check this option and in order to save time, it is advisable to cancel all the analysis scenarios that we have not used in the file of combinations we have created and on the basis of which the checks and sizing are done.
- The program automatically disregards in the automatic process all analysis scenarios related to the EIA as well as the cancelled ones.

In the manual selection of a script when this script is cancelled or a CAN script, the procedure is not executed.

Where in the "Scenario" field you can specify the Analysis and Sizing scenarios to be used in the optimization.
In the next section there are two convergence criteria:
Max Steps : Here you can specify the maximum number of steps (iteration cycles) where there will be no improvement.
Maj. No. Steps : Set the maximum number of repetitions
Improvement (%) : Set the minimum percentage of improvement which is considered as a lower limit in order to achieve convergence.

1.5 Settings

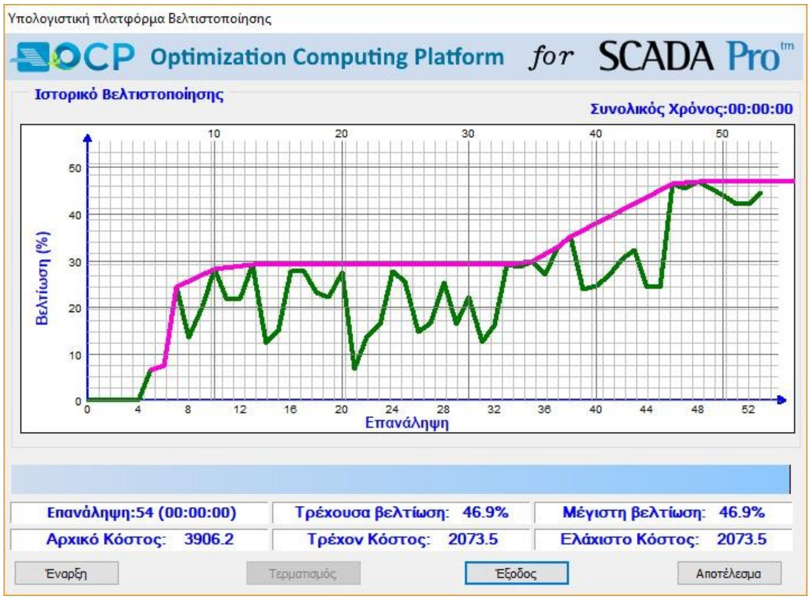
The two commands in this section will be activated in a later version of the program

1.6 Execution

The last section contains commands concerning the execution of the optimization process and its results.

1.6.1 Execution

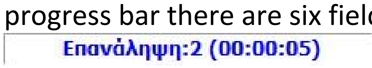
Using this command displays the following dialog box



Pressing the "Start" button starts the iterative optimization process.
At the top of the dialog box the graph of the optimization history is displayed where the horizontal axis is the number of iterations and the vertical axis is the percentage of improvement of the vector.

Below the graph is the progress bar

where messages describe the successive stages of the process. Below the progress bar there are six fields which are:



The number indicates the number of repetitions so far and in parentheses the time the convergent repetition has been running.

⚠ **OBSERVATION:**

At iteration numbered 0 is the upper boundary beam, i.e. the beam with largest cross-sections.
At iteration numbered 1 is the lower boundary beam, i.e. the beam with smallest cross-sections.
In iteration number 2 is the vector as originally designed by the designer and is considered as the reference vector.

The percentage described in the current improvement refers to the improvement of the current iteration vector over the original (reference) vector.

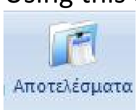
The percentage described in the maximum improvement refers to the improvement of the best operator achieved so far in any of the previous steps compared to the original operator (reference operator).

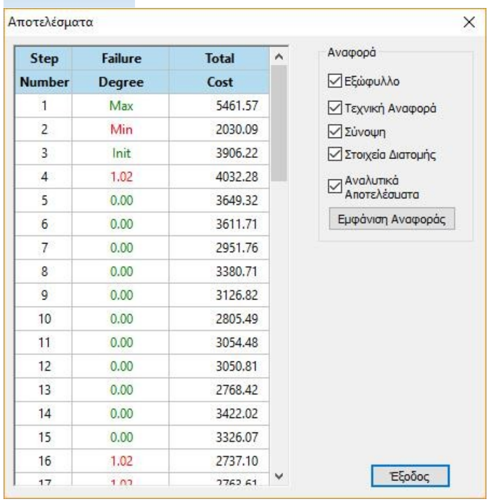
Αρχικό Κόστος: 2606.9 The amount shown here refers to the total construction cost of the entity in step 2 (reference entity).
Τρέχον Κόστος: 20866.5 The amount shown here refers to the total cost of constructing the carrier in the current iteration step.
Ελάχιστο Κόστος: 2594.6 The amount shown here is the total cost of building the most optimised carrier achieved so far.

The optimization process is completed as soon as convergence is reached or terminated if you press the Τερματισμός.

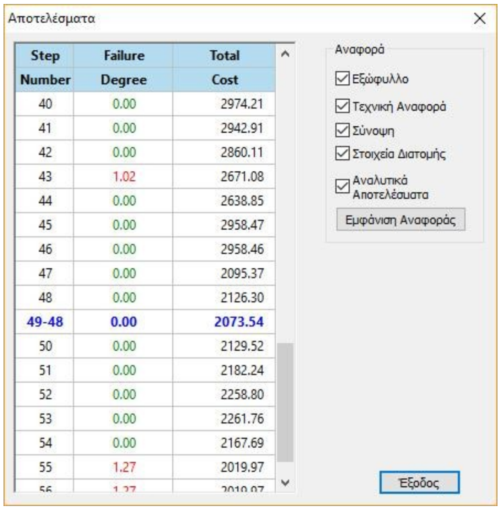
1.6.2 Results

Using this command displays the following dialog box



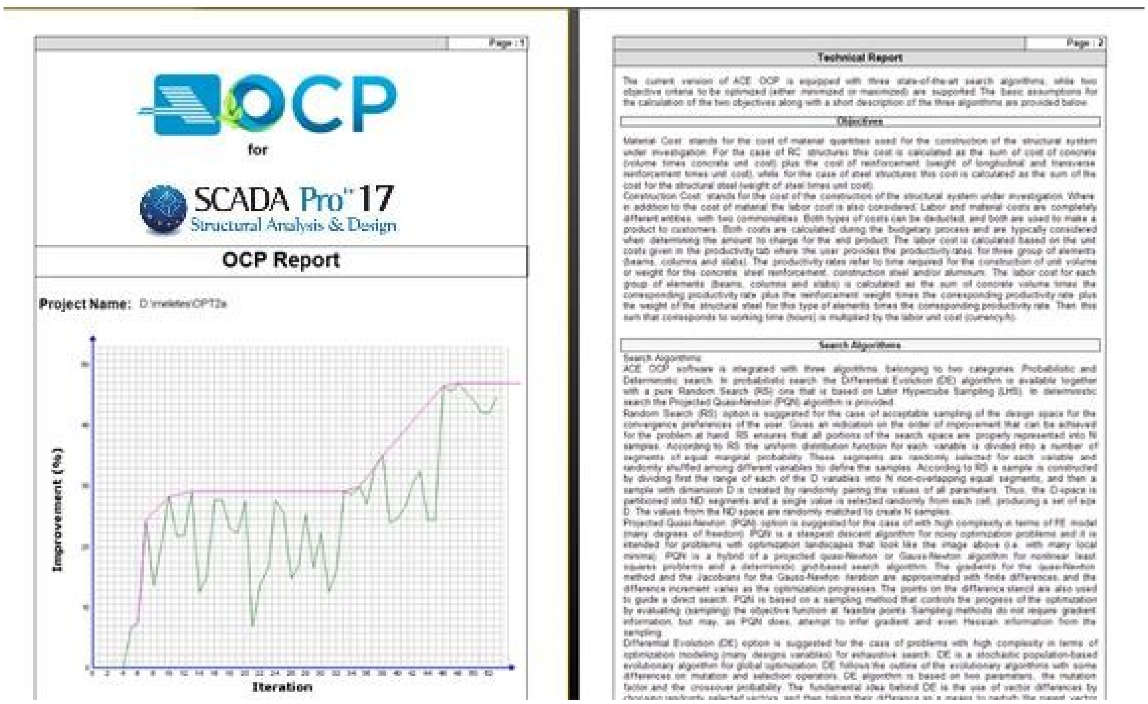


where the left side shows 3 columns with the optimization steps. The first column contains the serial number of the step, the next one the penalty degree of each step. The step with the optimal solution obviously has a penalty degree of 0 and is indicated in blue.



Step Number	Failure Degree	Total Cost
40	0.00	2974.21
41	0.00	2942.91
42	0.00	2860.11
43	1.02	2671.08
44	0.00	2638.85
45	0.00	2958.47
46	0.00	2958.46
47	0.00	2095.37
48	0.00	2126.30
49-48	**0.00**	**2073.54**
50	0.00	2129.52
51	0.00	2182.24
52	0.00	2258.80
53	0.00	2261.76
54	0.00	2167.69
55	1.27	2019.97

On the right side you select which parts you want to include in the print job. By selecting the Εμφάνιση Αναφοράς button, the report issue of the report is displayed.



The printout is a complete and detailed report that includes the theoretical background, the parameters, the convergence criteria, the final cost of the construction, the percentage of improvement achieved and the changes in the cross-sections of all structural elements.

1.6.3 Reset

Using this command deletes the procedure you may have run earlier and all the optimization parameters are returned to their original, default values.

OBSERVATION:

Within the study folder there is the OCP subfolder and within it RUNFILES is created which contains all files in the respective steps of the optimization (but without the analysis and sizing).

If after the optimization process you save the study, then the vector to be plastered will be the optimized one.

To open the original vector, select the file of the 2nd iteration and run the analysis and sizing again.