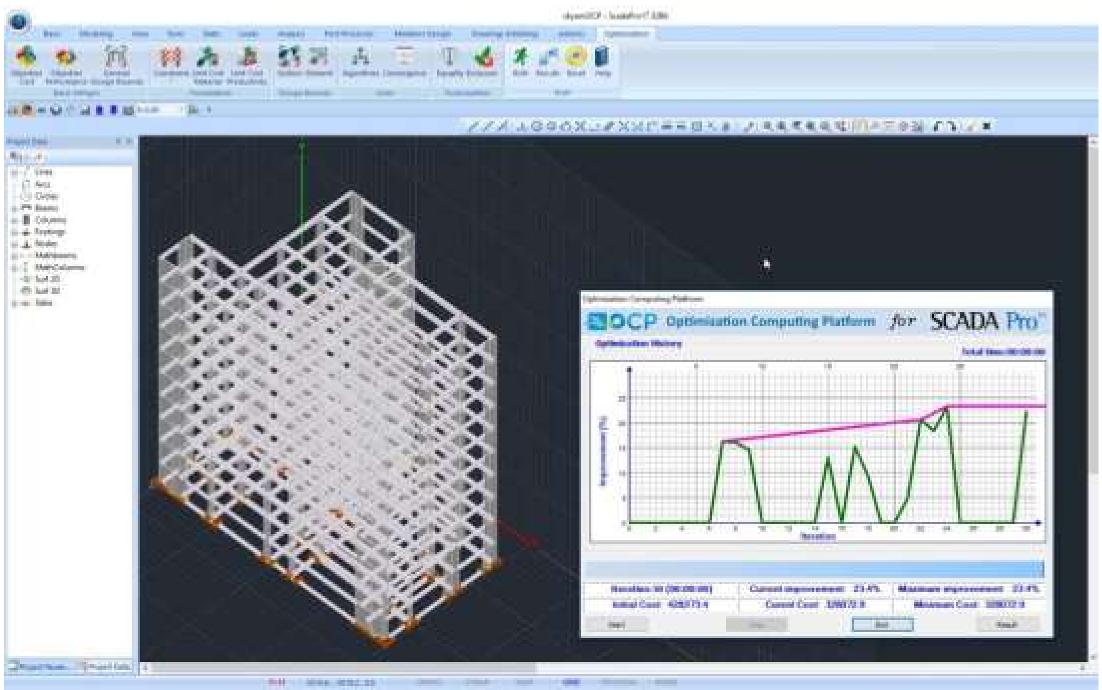
SCADA Pro tm

Structural Analysis & Design

Example 10

Optimisation of Reinforced Concrete Construction



ACE HELLAS

INTEGRATED SOLUTIONS

EXAMPLE 10: 'OPTIMISATION'

CONTENTS

EXAMPLE 10: 'OPTIMISATION'

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 ability to **select multiple criteria** related to the cost of construction, such as construction cost, cost of construction materials, environmental cost 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**.

Before Optimisation

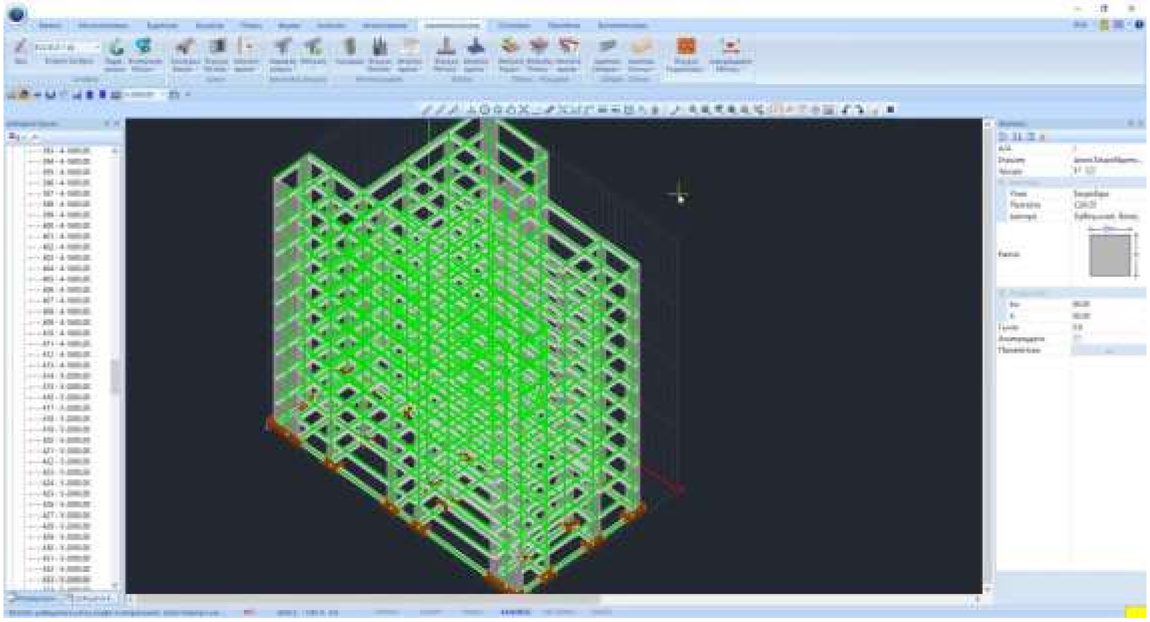
The process of optimising a carrier requires that it has been modelled and dimensioned beforehand.
So we start the setup of the vector as usual.

The specific example concerns a reinforced concrete structure (C20/25 and S400s), ten storeys and a foundation with footings and connecting beams.
The modeling of the vector was done using a dwg file and then with the help of Scada commands some modifications were made to the members of the individual floors.

The plates were inserted automatically. Then the loads were set and plate loads were assigned to the members.

After the vector setup was completed, a dynamic analysis of the Eurocode was performed and the combinations were created.
The last step was its dimensioning with a Eurocode scenario.

EXAMPLE 10: 'OPTIMISATION'



The aim in each study is to ensure that all data are adequate for all the controls required by the regulation. This example shows adequacy in both beams, columns and walls, as well as slabs.

Having arrived at such a result, the study could be considered complete and we could proceed to the printing, the wood type and finally its implementation.

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.

OBSERVATION:

With SCADA Pro OCP you can size operators that have deficiencies 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!

EXAMPLE 10: 'OPTIMISATION'

1. Optimization

So after the design is complete, and before the print part, open the "Optimization" section. "Optimization" 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 Variables 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.

[redacted] Basic Settings

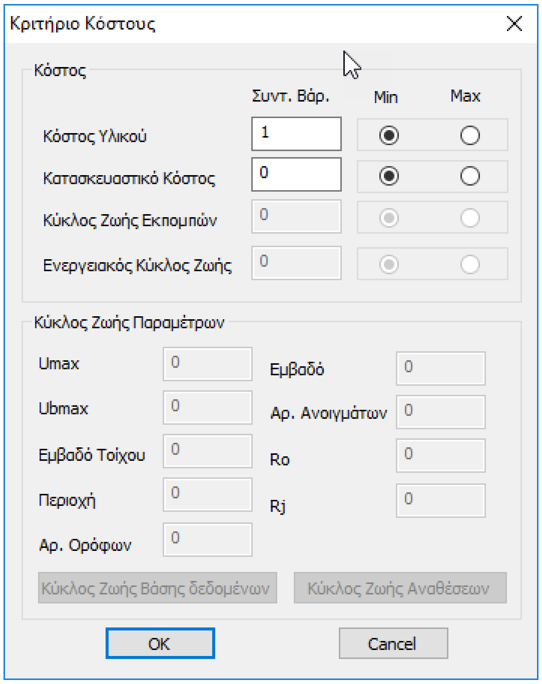
The first command subsection includes the following 3 commands:

- 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 form the criterion target of the optimization.
These functions are defined in the dialog box that appears:

EXAMPLE 10: 'OPTIMISATION'



where the active functions are **Material Cost** and **Construction Cost**.
The value of the Weighting Coefficient takes values from 0 to 1 and determines the percentage of participation of each criterion in the optimization. The sum of the weighting coefficients of the criteria that you want to participate as the final optimization objective should be equal to one.
Next to the weighting factor field there is a "Min" and "Max" indicator for each criterion to indicate whether the optimization goal is to minimize or maximize the criterion of our choice.

- For this example, the cost criterion was exclusively the **Cost of Materials.**

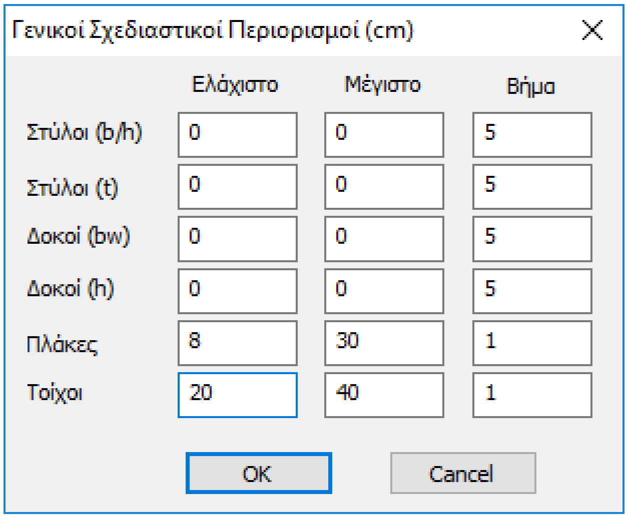
1.1.2 Objective Performance

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

1.1.3 Design Boundaries

Using the "**Design Boundaries**" command displays the following dialog box:

EXAMPLE 10: 'OPTIMISATION'



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 example the dimension of the thickness of the leg of a Gamma cross-section.
- 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 limits of the thickness dimension 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 automatically takes as lower limit the original value of the dimension of the element **reduced by 30%** and as upper limit again the original value **increased by 30%.** This variable applies only to the dimensions of beams and columns.

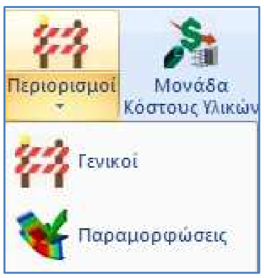
For this example, we leave the value 0 so that the program can automatically determine the limits of dimensional variation by varying the original dimensions by 30%.

EXAMPLE 10: 'OPTIMISATION'

1.2 Project parameters

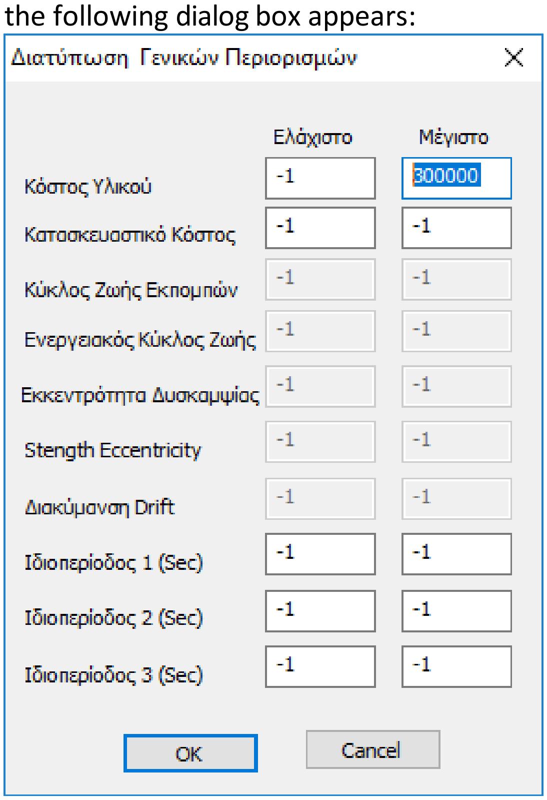
The next subsection deals with the project parameters and includes constraints imposed by the designer and the definition of material and production cost units.

1.2.1 Restrictions



Using the "**General Restrictions**" command

the following dialog box appears:



In this form you can define the quantities to be used as additional constraints of the problem by setting a minimum and a maximum value of the material cost, as well as the construction cost of your carrier, and a minimum and maximum limit for the eigenperiods of the first three eigenmodes.

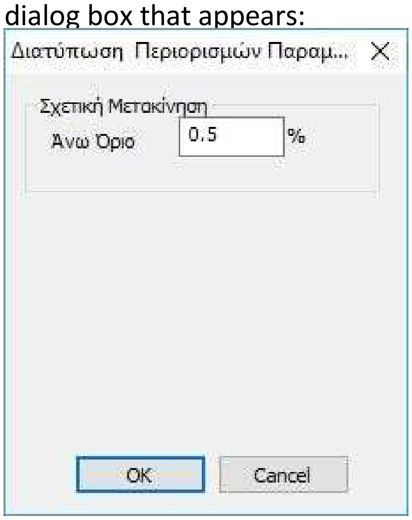
OBSERVATIONS:

- When performing the optimization process, in addition to the restrictions imposed by the regulatory provisions of the Regulation you have already selected in the sizing, the above General Restrictions will be taken into account.
- A value of -1 in the above fields means that the corresponding constraint is inactive and does not participate in the optimization process.

For this example we define the maximum Material Cost as 300.000 (in currency of the place where the construction will take place) and we do not set any other restriction, i.e. we leave the values -1 in the other fields. This means that the resulting final construction will have the minimum possible material cost, whose value will be less than 300.000€ (Greek currency) and will satisfy the controls of the regulation you have already selected in the dimensioning phase.

Using the next command "**Deformations**", and in the

dialog box that appears:

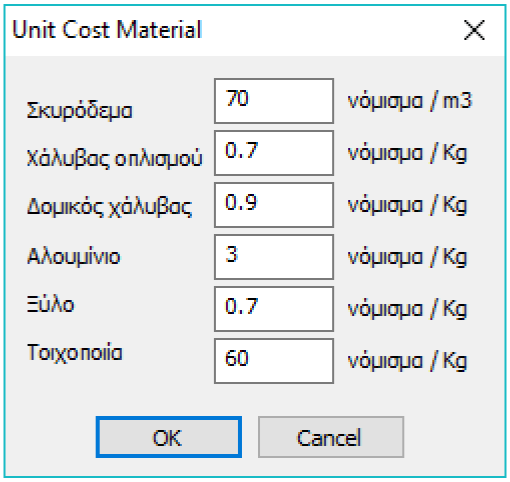


specify an upper limit to the relative movement (drift) of the floors of the carrier. The resulting optimal solution will also obey this constraint.

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 dialog box below:

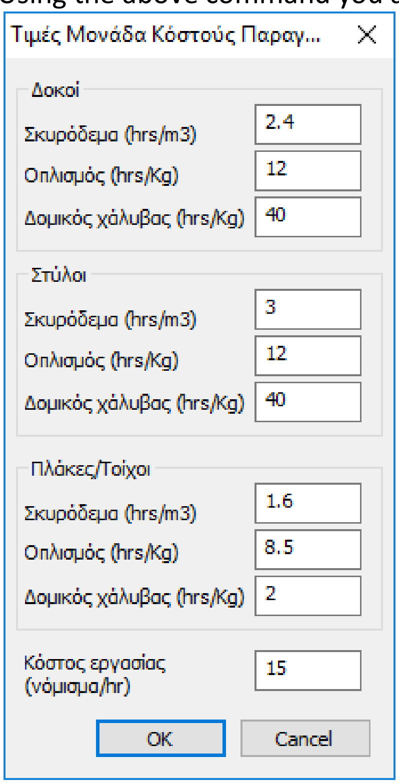
EXAMPLE 10: 'OPTIMISATION'



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

1.2.3 Unit Cost of Production

Using the above command you are given the option to specify:



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 are the limits of design variables that you can set per section type or per structural element. The design variables are the sizes of the structure that change during optimization, i.e. the dimensions of the structural elements, and the limits are the range of values in which each dimension will take values. The way of imposing the limits of the design variables in optimization process (OCP) structures analyzed and designed in Scada Pro, results in grouping the structural elements following a logical hierarchy:

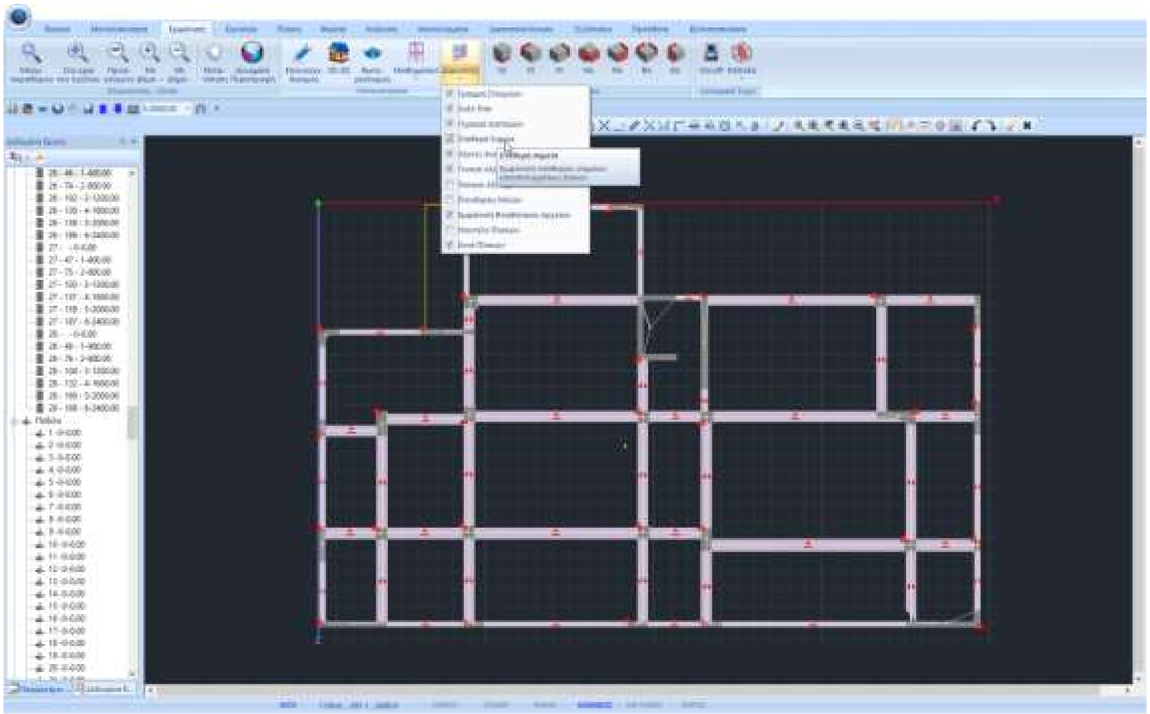
- starts from **general - total constraints** applicable to each type of structural element (columns, beams, slabs)
- continues at a more specific level **per cross-section** with the structural elements they include, grouping elements of the same type, e.g. beams, by type of cross-section,
- to arrive at the definition of the limits **per structural element** and at the level of grouping of individual linear or surface elements, grouping elements of the same type and cross-section per structural element.

These restrictions concern either minimum and maximum dimension limits and their step of change, 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.

⚠ **IMPORTANT OBSERVATION:**

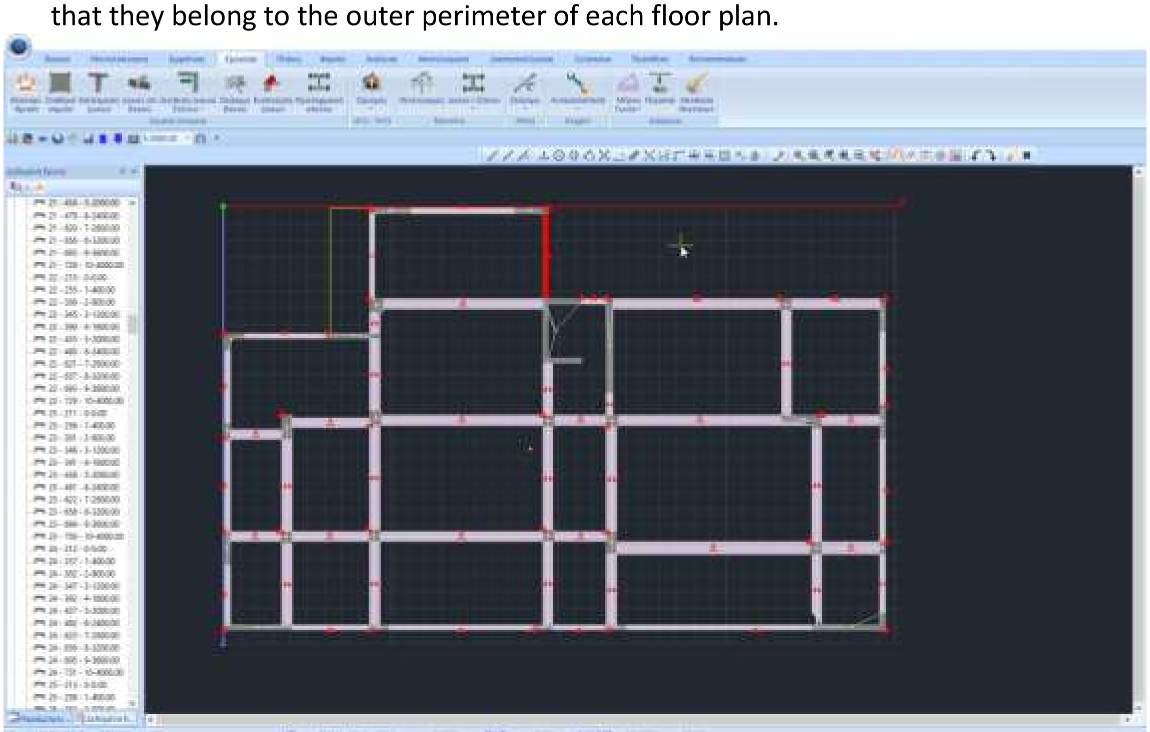
It is important to clarify that all the variations of the cross-sections during the Optimization are done keeping always constant their Fixed Points and the corresponding Fixed Passages of the beams.
This means that before starting the optimisation process, it is necessary to adjust the Fixed Points of the cross-sections in such a way that the floor plans, outer boundaries and passages of columns and beams are not deformed.
Through the "Show" section and the "Switches" command, activate the Fixed Points and display the study floor plans one by one to locate the fixed points and change them if necessary.

EXAMPLE 10: 'OPTIMISATION'



With the "Fixed Points" command in the "Tools" section you can modify the fixed point of the post and the fixed pass of the beam.

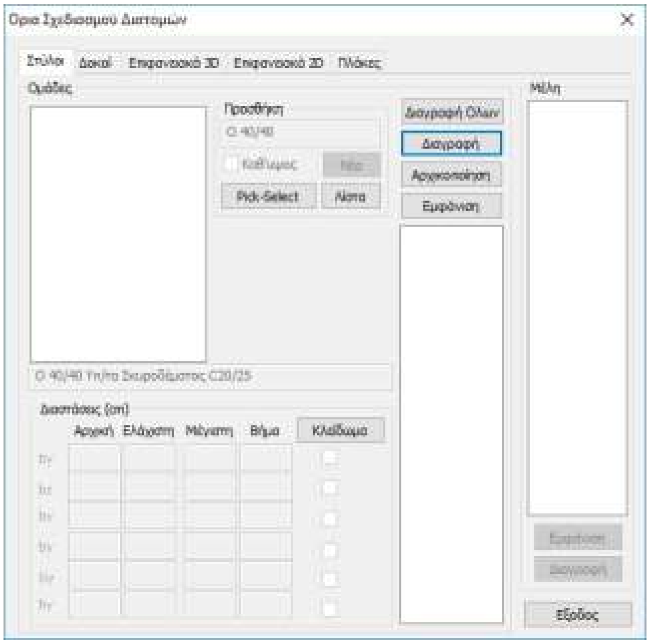
- In this example, the fixed points of the perimeter columns and beams were changed so that they belong to the outer perimeter of each floor plan.



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 dialog box:

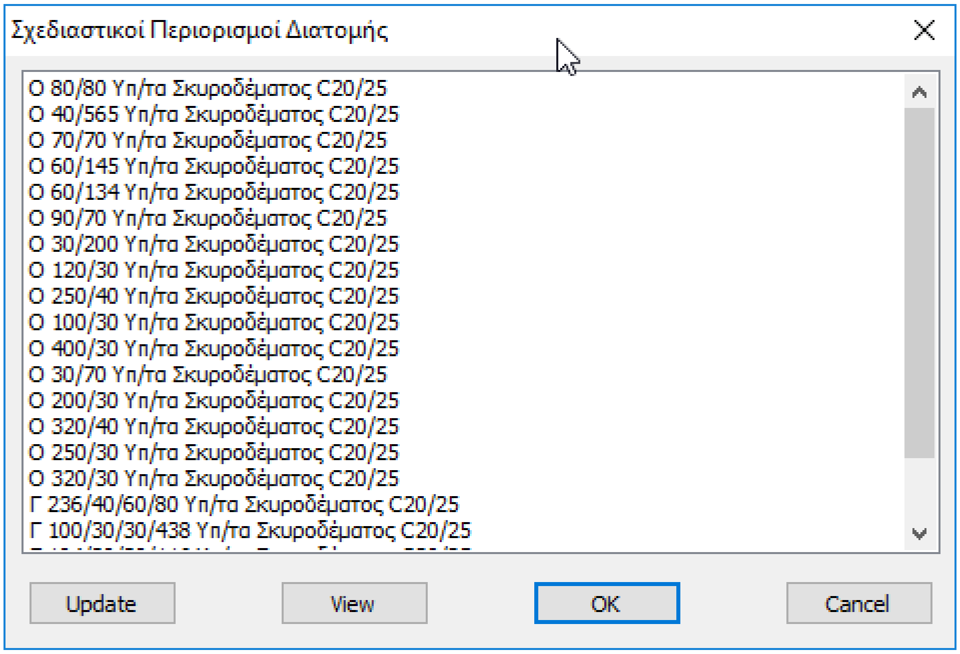


The import and processing of cross-sections is done by type of structural element.

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.
The "List" button displays the following dialog box:

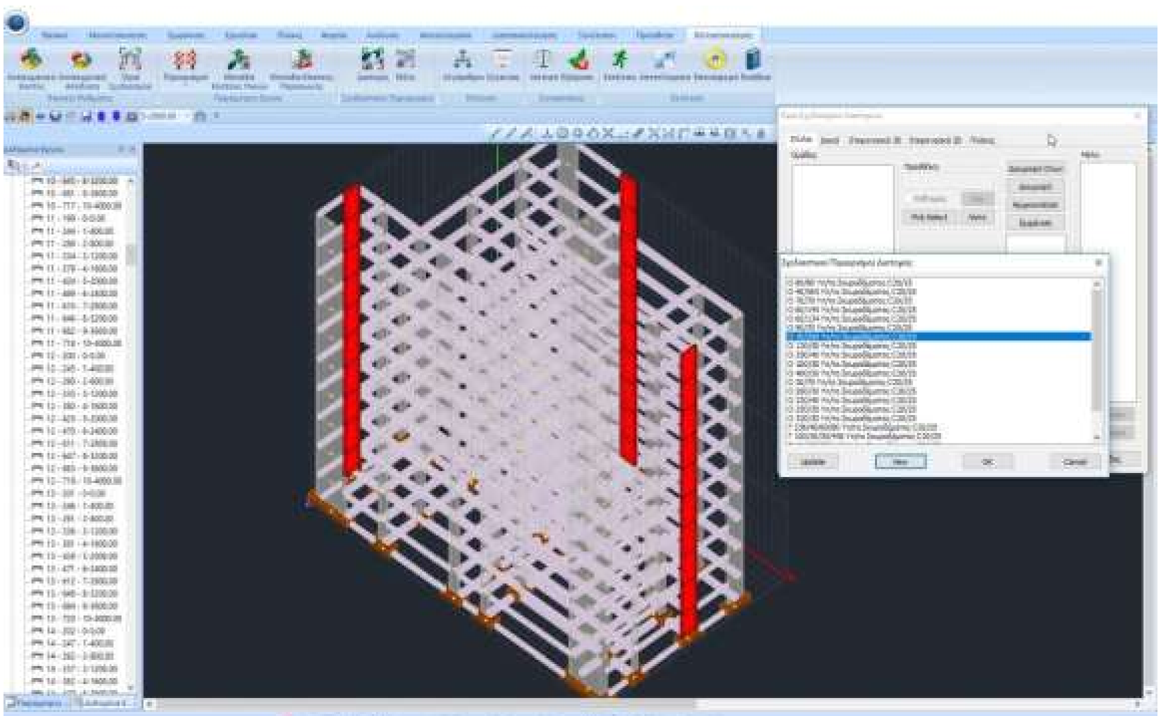
EXAMPLE 10: 'OPTIMISATION'



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

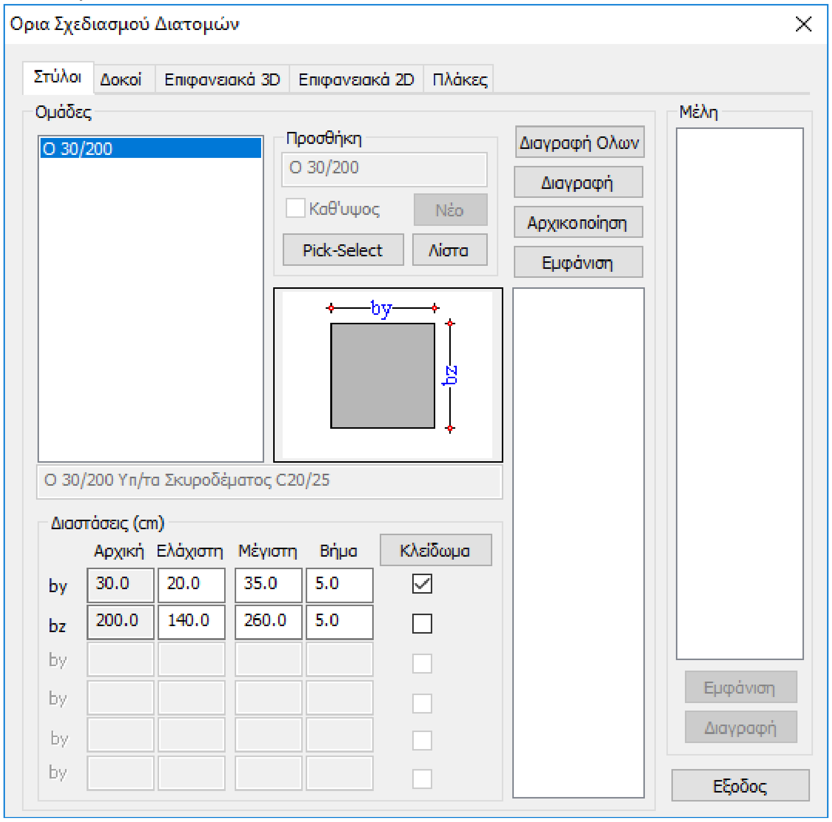
The selection is made for one or more cross-sections. After selecting the cross-section by pressing the "View" button, the structural elements with this cross-section are displayed in red on the vector.



EXAMPLE 10: 'OPTIMISATION'

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.



For this example we choose the 30X200 walls.

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.

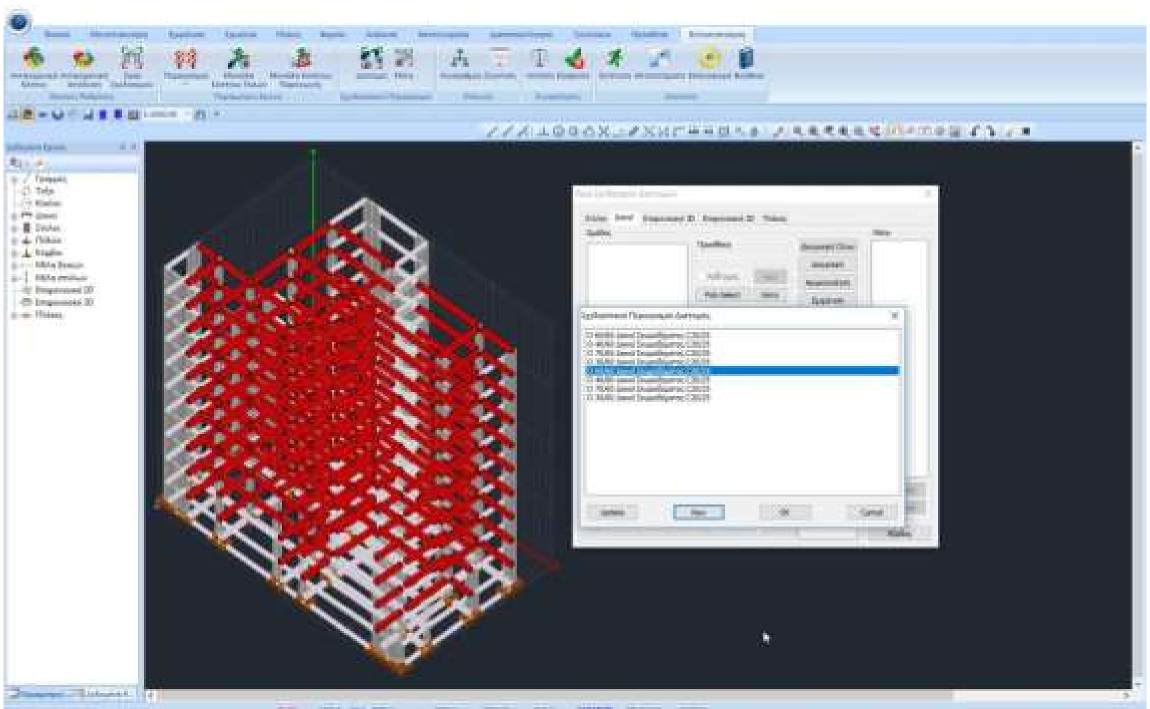
The "Lock" column locks this dimension to remain unchanged and fixed at its original value. Pressing the "Lock" button checks all the dimensions, i.e. the whole cross-section is locked. The default values in the boundary and Step columns are those defined in the General Design Parameters, which are initially obeyed by all elements.

- For this example we lock the dimension 30cm.

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.
With 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.

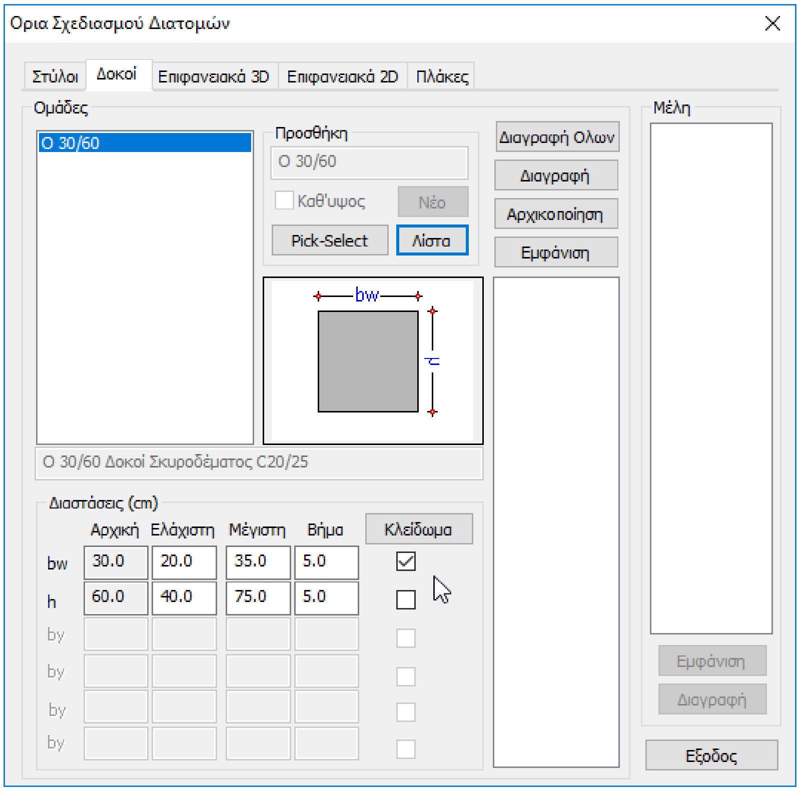
2) For the "**Beams**" section



exactly the same applies as for the pillar module.

- For this example and for the 30X60 beams we lock the width 30 cm.

EXAMPLE 10: 'OPTIMISATION'



3) The next two sections deal with finite surface element cross sections **"Surface 3D"** and **"Surface 2D"**.

This example does not contain surface elements and is therefore omitted. The mode of operation is the same in both cases and is explained in detail in the respective user manual.

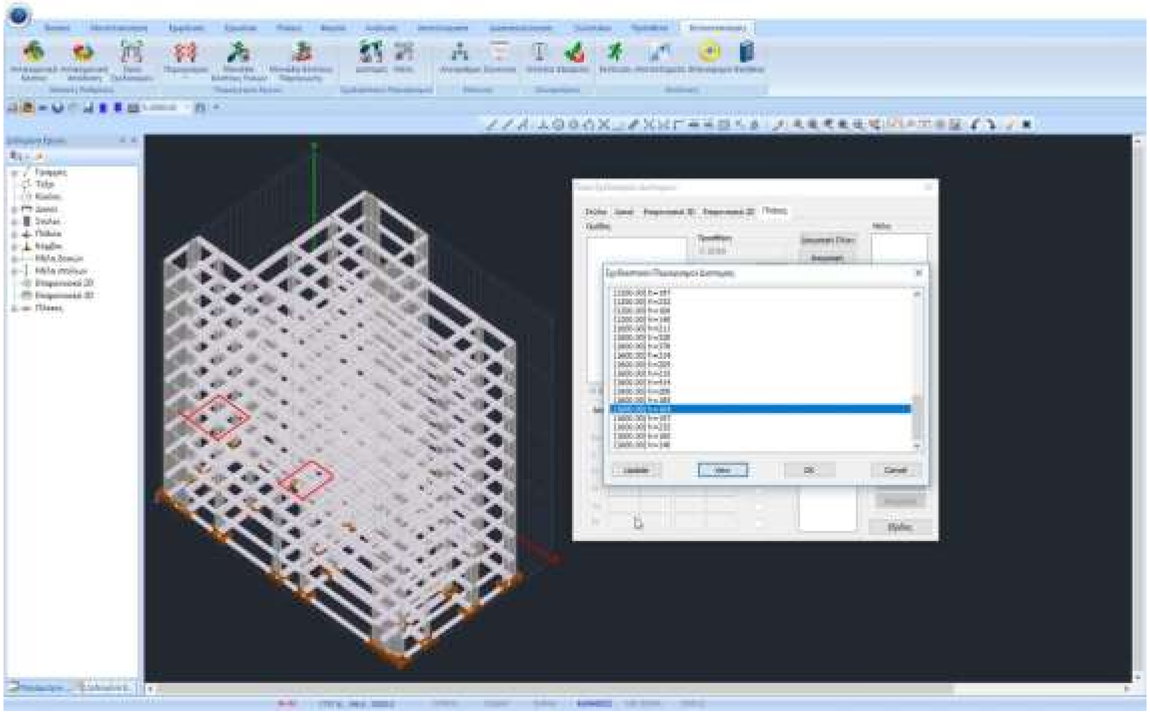
4) The last section **"Slabs"** concerns conventional slabs. The choice here can be made in two ways.

With the appearance of the list, where it includes all the plates of the construction. The categorization of the cross-sections is based on two criteria:

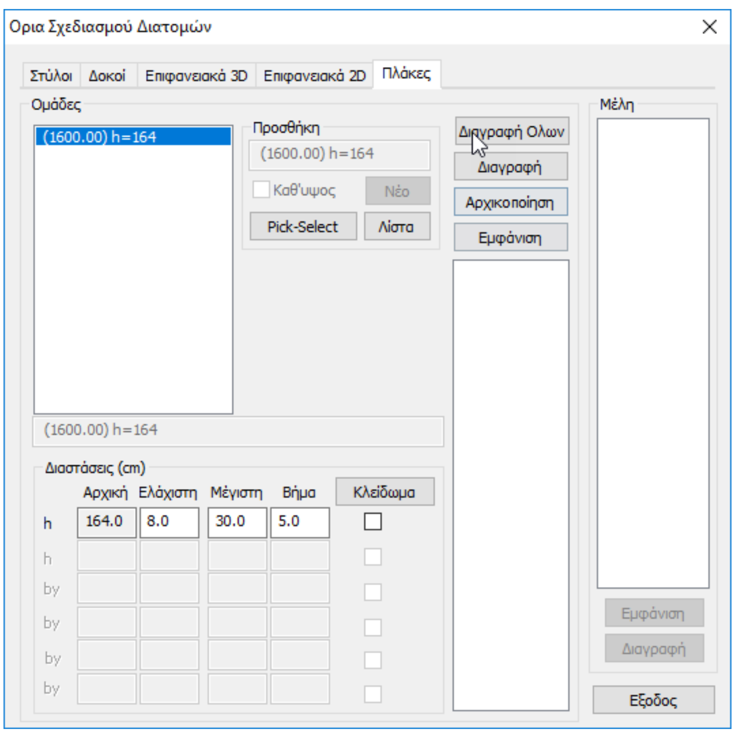
- The floor
- The different thickness

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.

EXAMPLE 10: 'OPTIMISATION'



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.

- For this example we do not impose any additional constraint on the plates.

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

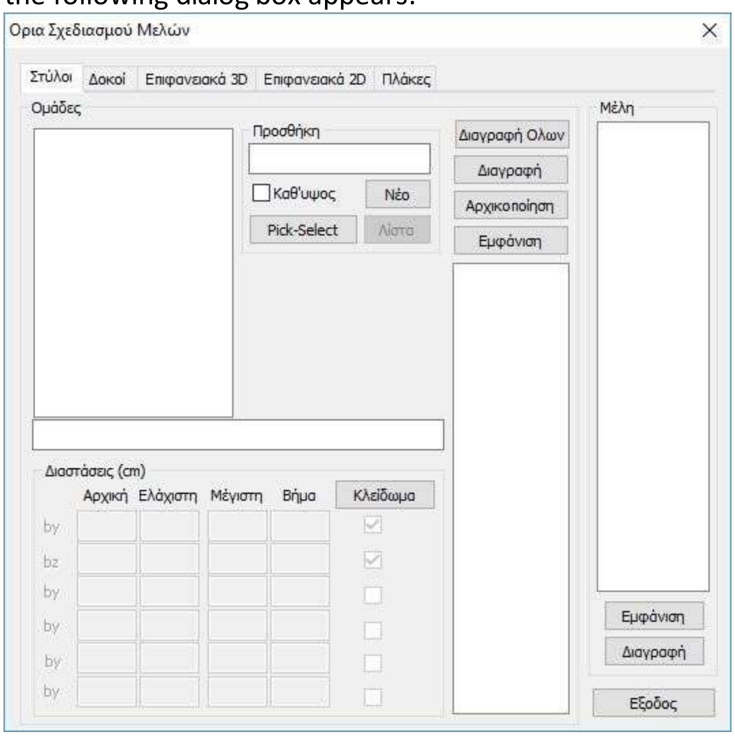
- This example does not contain metallic cross-sections and is therefore omitted. The mode of operation is explained in detail in the relevant user manual.

1.3.2 Members

The next instruction is 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.

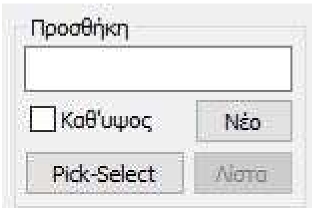
The structure of the dialog box is the same as that of the cross-sections, i.e. divided into sections according to the type of element.

EXAMPLE 10: 'OPTIMISATION'

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.

For example, if the cross-section of the first pole is 200/30 all subsequent poles must have this cross-section and the same material quality and belong to the same layer.

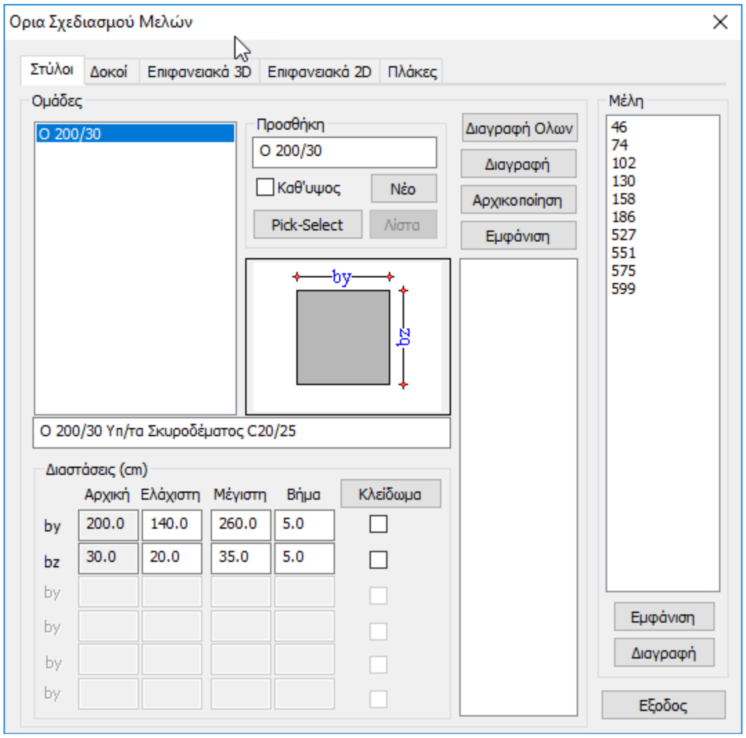
You can choose either the physical cross-section or the mathematical member. The element type must be the same as the corresponding module you are in.

OBSERVATION

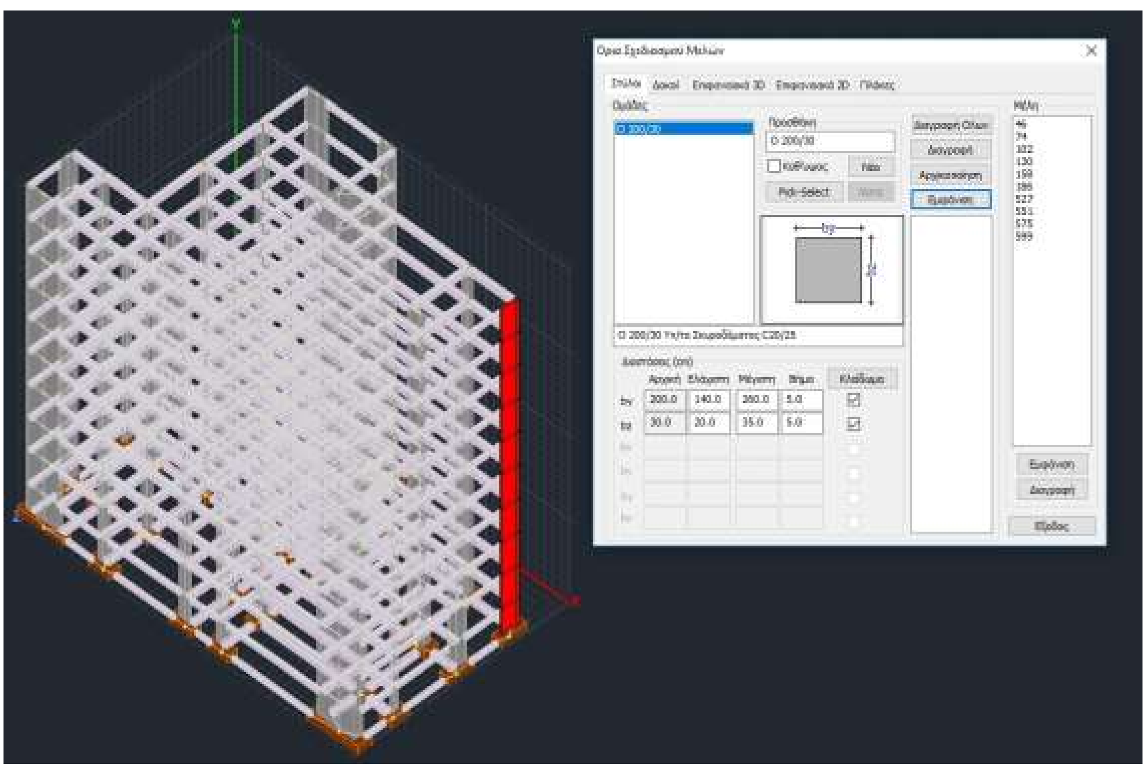
⚠ If you do not enter a name for your group and simply press "New" the program first puts English question marks "???" as the name and then names the group based on the cross-section of the first element you select. Of course you can modify the name of the group.

By selecting the items, they appear in the corresponding list:

EXAMPLE 10: 'OPTIMISATION'



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.



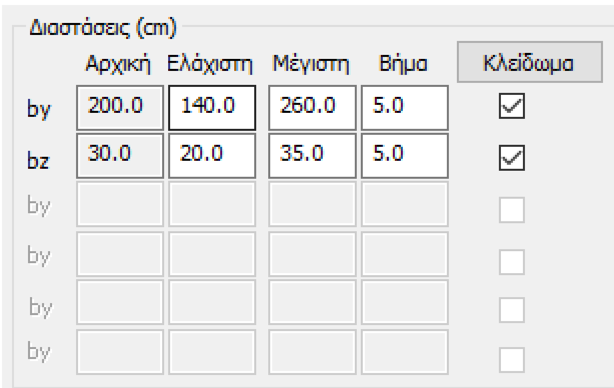
EXAMPLE 10: 'OPTIMISATION'

⚠ 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 

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.

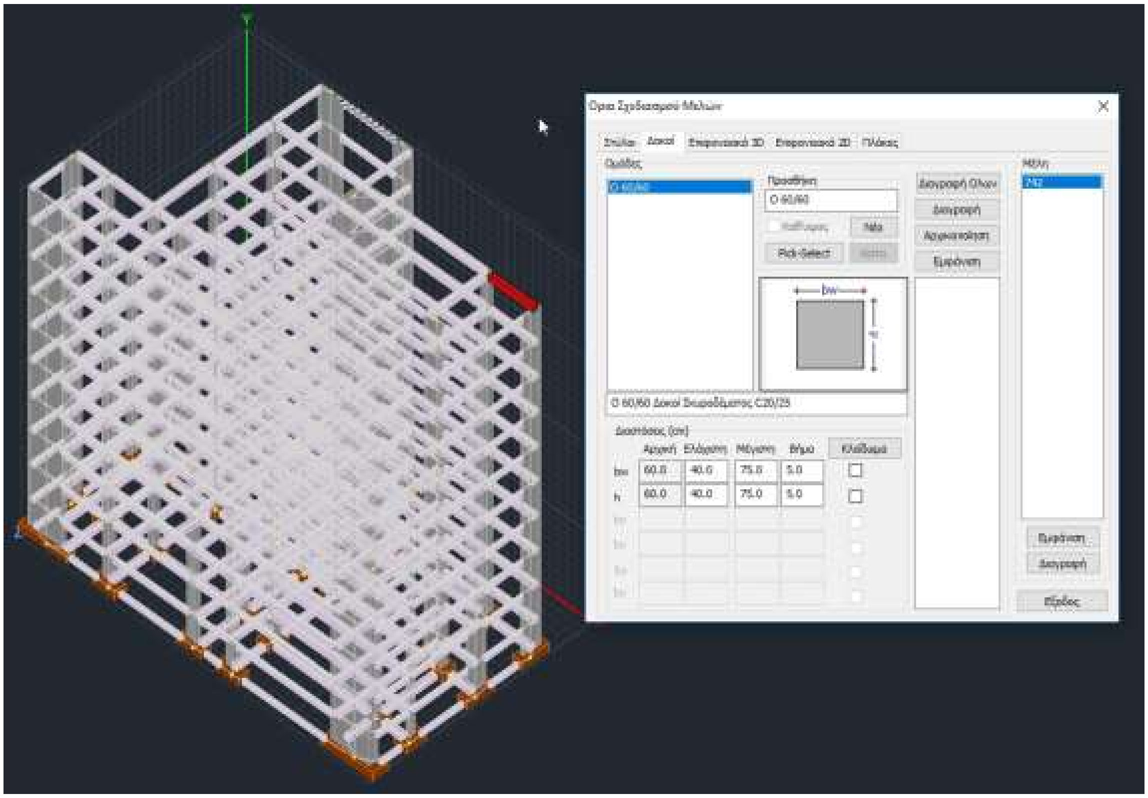
For this example it was chosen to lock both dimensions of whole group of 200X30 poles.

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 in red the elements of selected group.

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

EXAMPLE 10: 'OPTIMISATION'

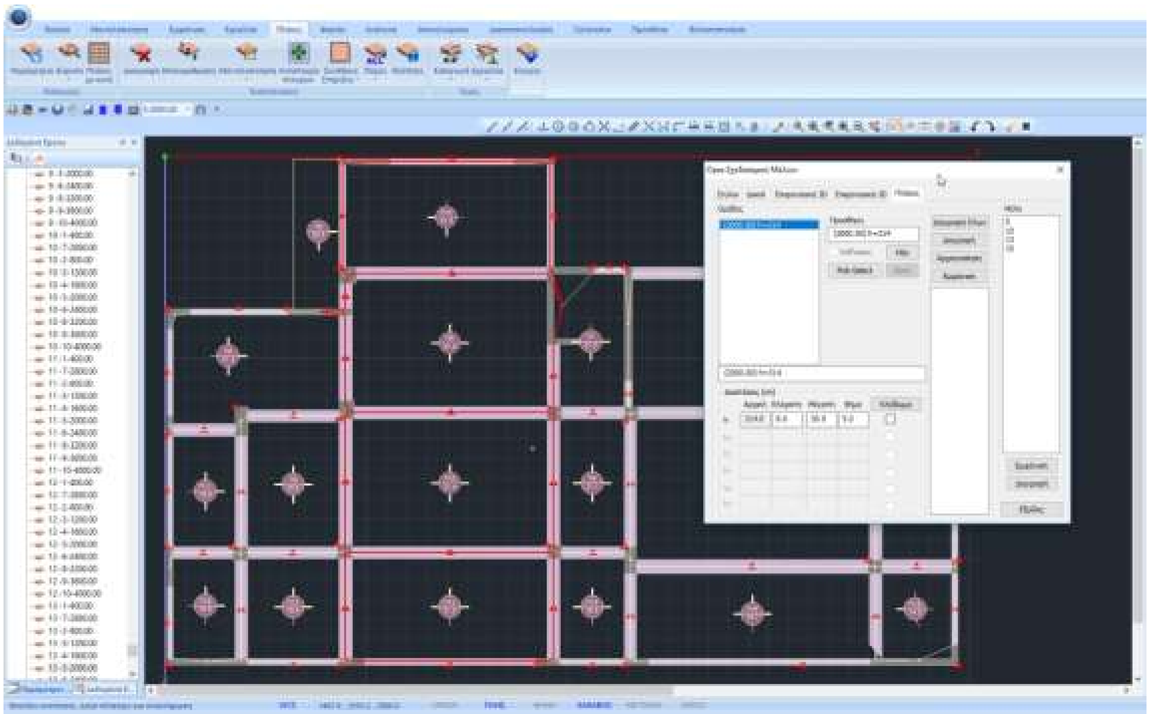


- For this example, no design limits are defined for the beams.

3) For the next two sections **2D** and **3D Surface** detailed description can be found in the relevant user manual.

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

EXAMPLE 10: 'OPTIMISATION'



In this example, a group of slabs with a thickness of 314 mm belonging to level 5 with an elevation of 2000.00 cm was created and includes 4 slabs, as shown in the list of members.

Finally, note that in order to display the plates and make their selection graphically, you must go to the corresponding Plates section.

So all the above tools and commands give you the ability and flexibility to set the limits of the design variables to as many members and in any way you wish.

EXAMPLE 10: 'OPTIMISATION'

1.4 Resolution

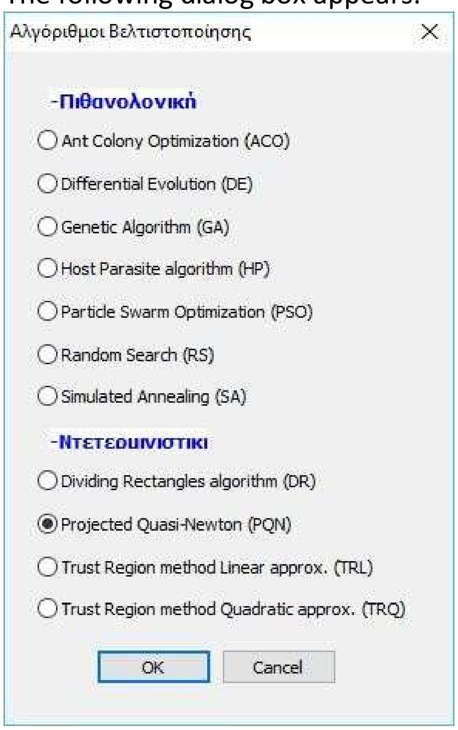
Two commands are included in this section:

- 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:

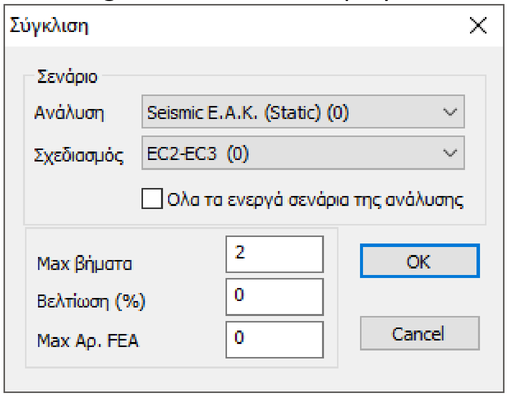


where you can select the optimisation algorithm from two categories

- Possibles
- Determinist

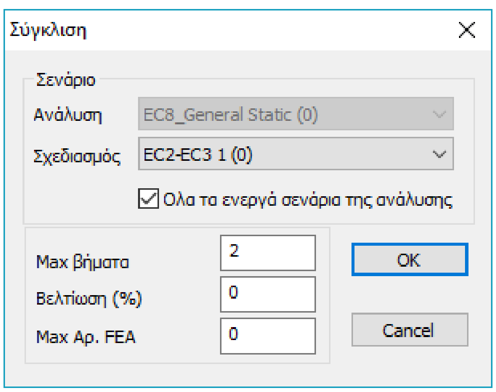
1.4.2 Convergence

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.

EXAMPLE 10: 'OPTIMISATION'

In the next section there are two convergence criteria:
Max Steps : Here you can indicate the maximum number of steps (cycles of repetitions) where there will be no improvement.
Improvement (%) : Set the minimum percentage of improvement which is considered as a lower limit in order to achieve convergence.
Max FEA No. : Set the maximum number of repetitions

1.5 Settings

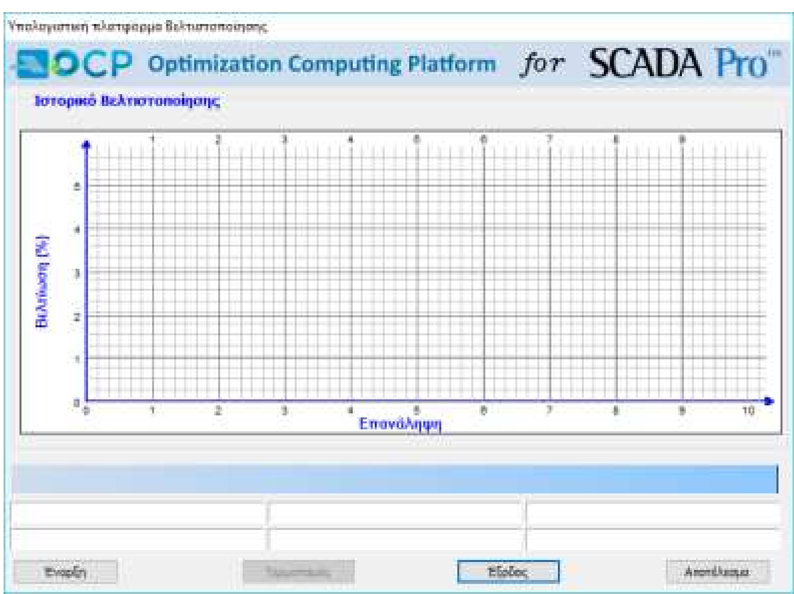
The two commands in this section will be activated in a later version of the program

1.6 Execution

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

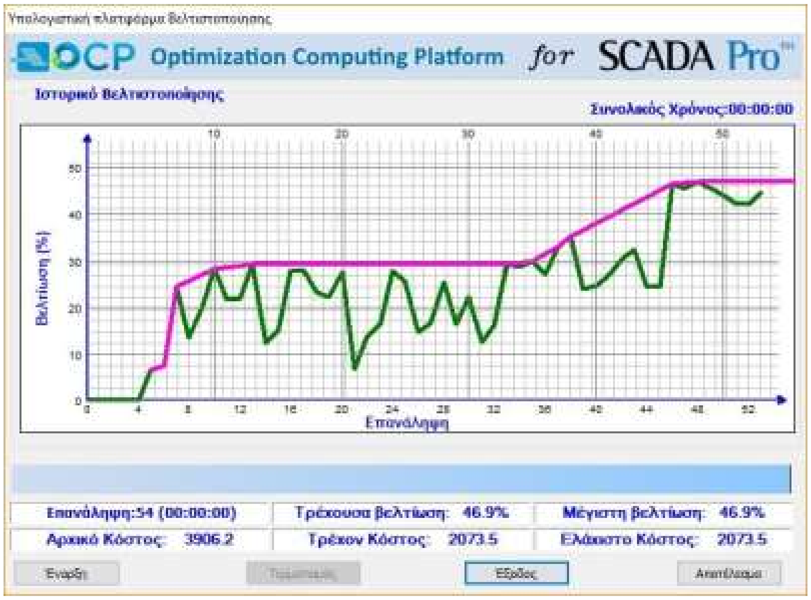
1.6.1 Execution

Using this command displays the following dialog box



EXAMPLE 10: 'OPTIMISATION'

Pressing the "Start" button starts the iterative optimization process and the following graph is generated and the fields are filled with the corresponding information.



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

Διαστασιολόγηση Στύλων

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

Επανάληψη:2 (00:00:05) 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.

Τρέχουσα βελτίωση: 0.0% The percentage described in the current improvement refers to the improvement of the current iteration vector over the original (reference) vector.

Μέγιστη βελτίωση: 0.5% 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).

EXAMPLE 10: 'OPTIMISATION'

Αρχικό Κόστος: 2606.9 The amount shown here refers to the total construction cost of the entity in step 2 (reference entity).

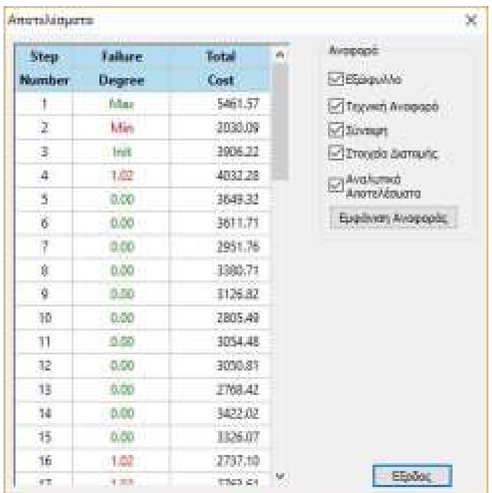
Τρέχον Κόστος: 20866.5 The amount shown here refers to the total cost of constructing the carrier in the current iteration step.

Ελάχιστο Κόστος: 2594.6 The amount shown here is the total cost of building the most optimised carrier achieved so far.

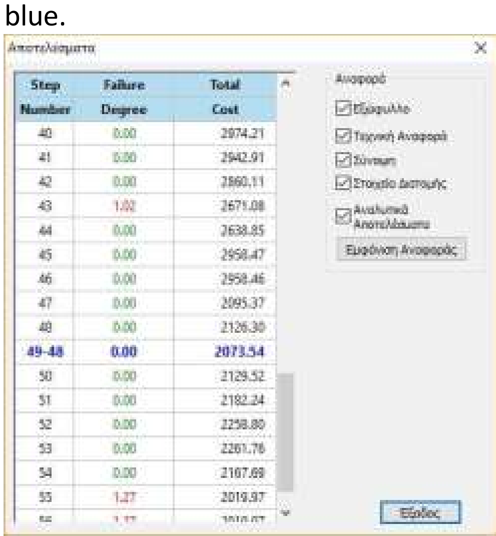
The optimization process is completed as soon as convergence is reached or terminated if you press the Τερματισμός.

1.6.2 Results

Using this command displays the following dialog box:

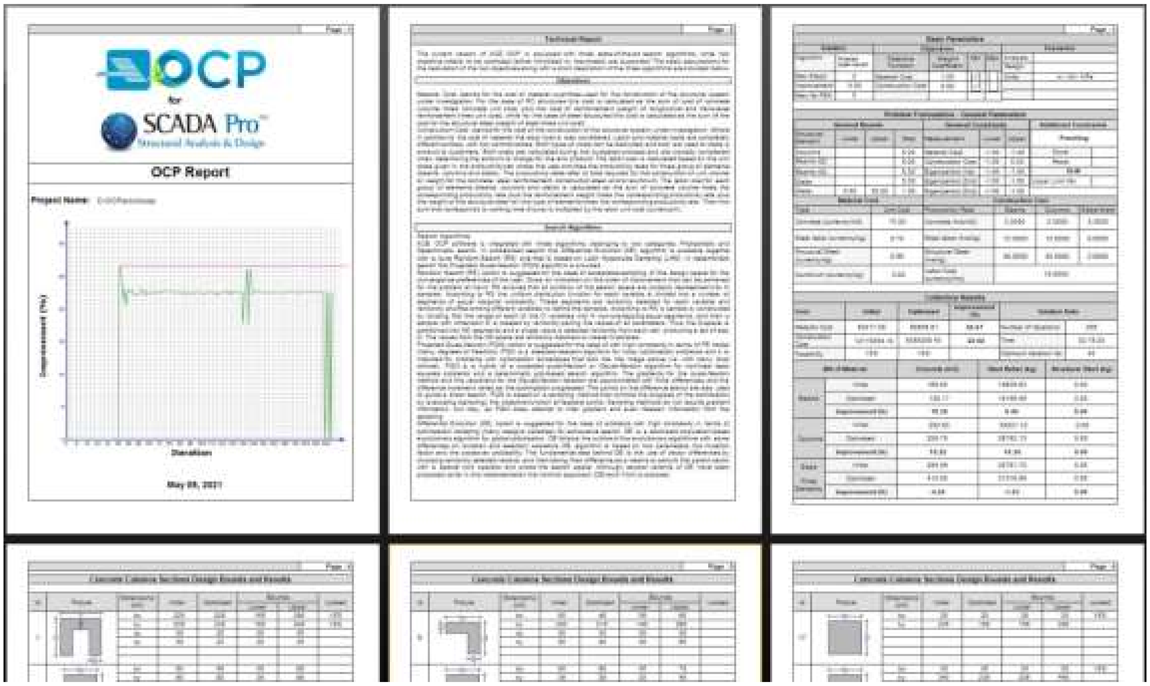


where the left side shows 3 columns with the optimization steps. The first column contains the serial number of the step, the next one the degree of constraint violation of each step. The step with the optimal solution obviously has a penalty degree of 0 and is indicated in blue.



EXAMPLE 10: 'OPTIMISATION'

On the right side you select which parts you want to include in the print job. By selecting the button [Εμφάνιση Αναφοράς] the 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 process you may have run earlier and all the optimization parameters are returned to their original, default values.