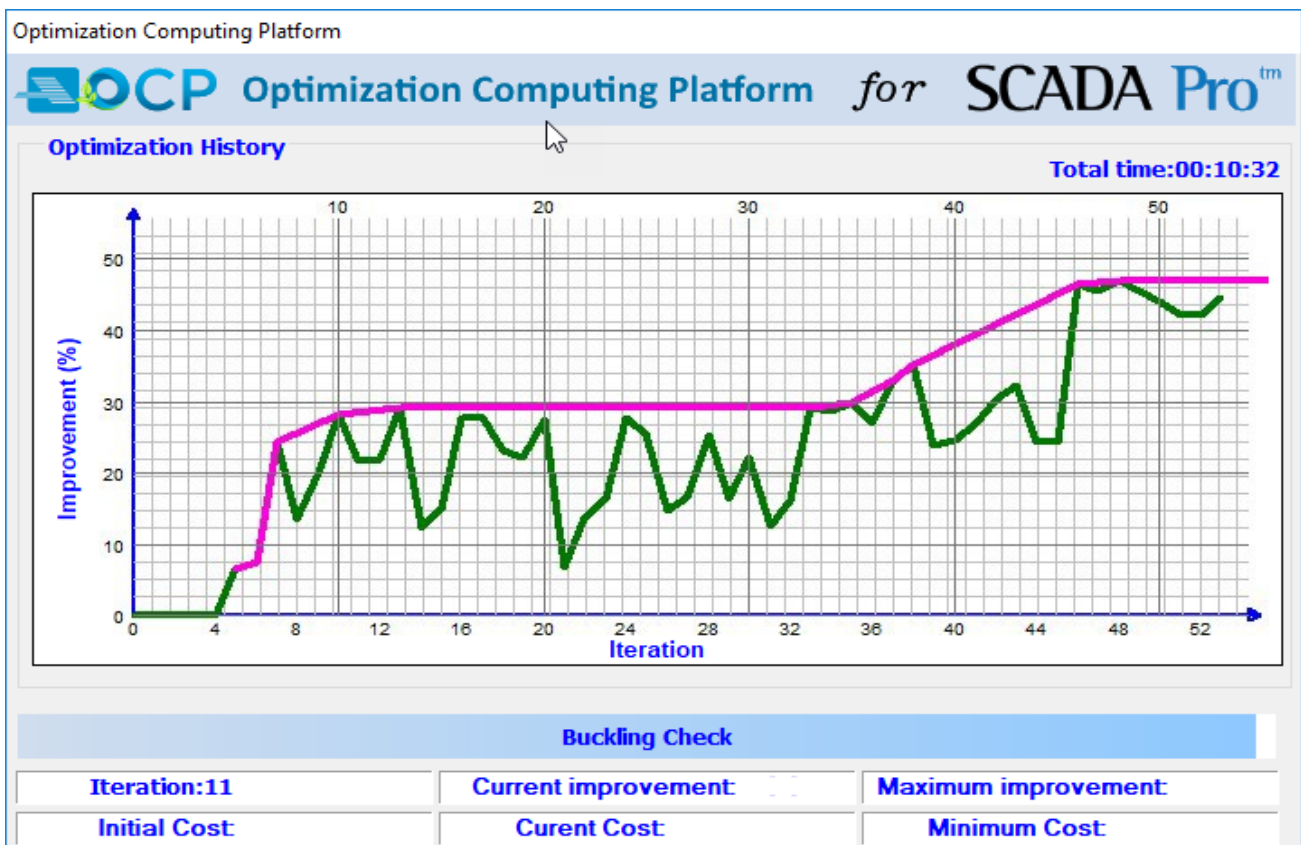




SCADA Protm
Structural Analysis & Design

User's manual

12.OPTIMIZATION



CONTENTS

I.	SCADA PRO OCP	3
1.	OPTIMIZATION	3
1.1	BASIC SETTINGS	4
1.1.1	COST OBJECTIVE FUNCTION.....	4
1.1.2	PERFORMANCE OBJECTIVE FUNCTION	5
1.1.3	DESIGN BOUNDS	5
1.2	PROJECT PARAMETERS	6
1.2.1	CONSTRAINTS.....	6
1.2.2	MATERIAL COST UNIT	8
1.2.3	PRODUCTIVITY COST UNIT.....	8
1.3	DESIGN BOUNDS	9
1.4	SOLUTION.....	21
1.5	FUNCTIONALITIES.....	23
1.6	RUN	23

I. SCADA Pro OCP

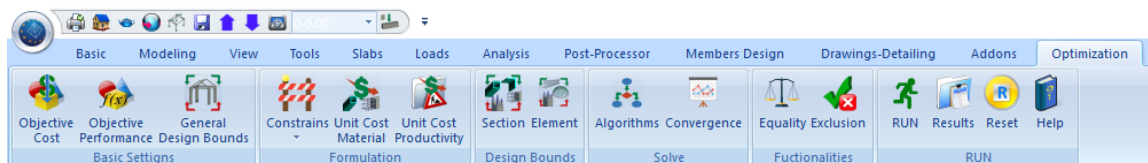
SCADA Pro OCP is a new module of SCADA Pro that represents an advanced, real-world, optimum design computing platform for real-world civil engineering structures. The main goal for the implementation of SCADA Pro OCP module is to **reduce construction and material cost** consistent with the required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

SCADA Pro OCP module provides:

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

1. Optimization

This section is about defining the parameters, performing and displaying the results of the optimization process. Commands are grouped into sections according to their type of use.

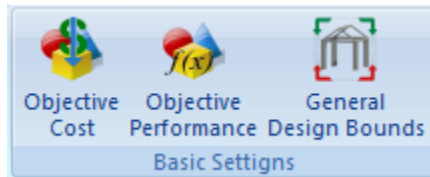


As it was pointed out in the theoretical manual, the optimization process is carried out in two phases:

- The first phase defines the basic settings, project parameters, and design constraints.
- The second phase specifies the optimization algorithm and runs the iterative computational process.

There are also some additional features and capabilities, where can be useful for further grouping and definition of the design variables.

1.1 Basic Settings

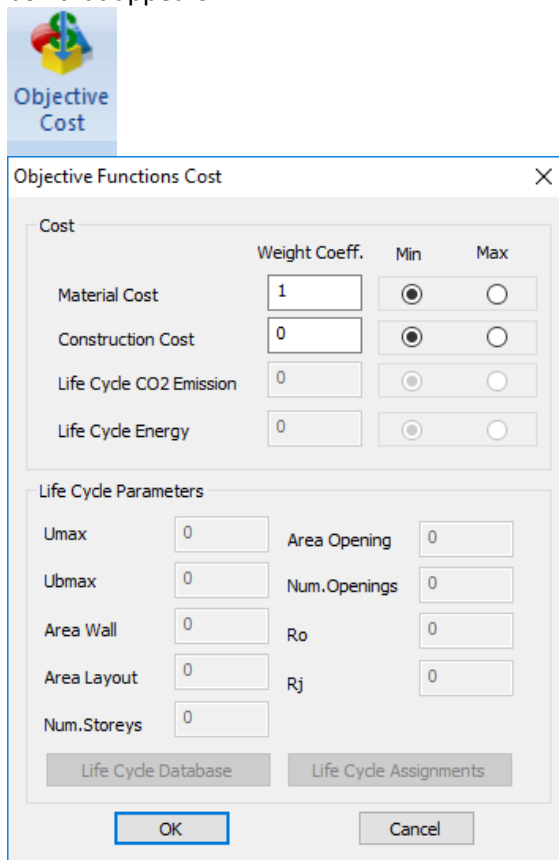


The first command subsection includes the following three commands:

- Cost Objective Function
- Performance Objective Function
- Design Bounds

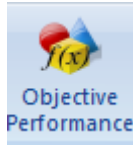
1.1.1 Cost Objective Function

Using this command allows you to select the objective function or a combination of objective functions to perform the optimization process. The definition of these functions is in the dialog box that appears:



Where active objective functions are the Cost of Materials and the Construction Cost. The value of the Weight Coefficient gets values from 0 to 1 and it is indicative of the participation percentage of each function to the final objective function. “Min” and “Max” variables define the selected criterion will be either minimized or maximized during the optimization process. The sum of weight coefficients in the two fields must be a unit.

1.1.2 Performance Objective Function



The options of this command will be available in a future version of the program.

1.1.3 Design Bounds

By using the command "Design Constraints", the following dialog box appears,



General Design Bounds (cm)			
	Lower	Upper	Step
Columns (b/h)	0	0	5
Columns (t)	0	0	5
Beams (bw)	0	0	5
Beams (h)	0	0	5
Slabs	8	30	1
Walls	20	40	1

OK Cancel

where the general minimum and maximum bounds of the dimensions and the step of changing these dimensions during the optimization process are determined for each type of structural element.

ATTENTION!

- The above dialog box options are only for reinforced concrete elements.
- Bounds are determined for each type of structural element (Columns, Beams, Slabs, and Walls).
- For the columns, the two "categories" are:
 - (i) The first (b / h) for all major dimensions, regardless of the type of cross-section (Rectangular, L, Tee etc.).
 - (ii) The second dimension (t) for the corresponding minor dimensions, mainly the thickness.
- For the beams, the bounds of their two basic dimensions are defined, namely width (bw) and height (h).

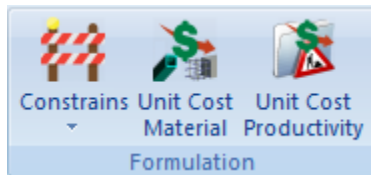
The thickness bounds for the slabs are then determined. It includes solid slabs as well as those simulated with finite surface elements.

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

NOTE:

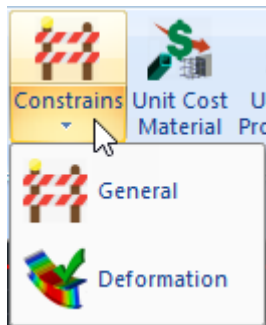
The value 0 in the fields of the maximum and the minimum bound means that the program receives as lower bound the initial value of the dimension of the structural element reduced by 30% and as upper bound the initial value increased by 30%. This variable applies only to the dimensions of beams and columns.

1.2 Project Parameters



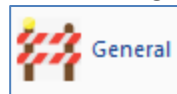
The next subsection is about the project parameters and includes constraints imposed by the designer as well as the units of the material and productivity cost.

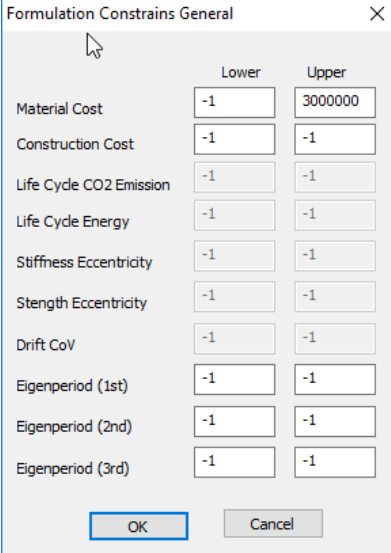
1.2.1 Constraints



Using the "General Constraints"

the following dialog box appears,





	Lower	Upper
Material Cost	-1	3000000
Construction Cost	-1	-1
Life Cycle CO2 Emission	-1	-1
Life Cycle Energy	-1	-1
Stiffness Eccentricity	-1	-1
Stength Eccentricity	-1	-1
Drift CoV	-1	-1
Eigenperiod (1st)	-1	-1
Eigenperiod (2nd)	-1	-1
Eigenperiod (3rd)	-1	-1

OK Cancel

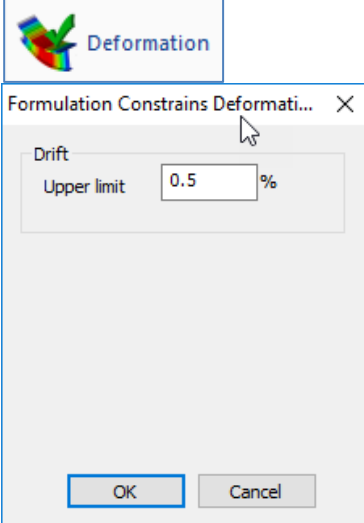
where you can specify a minimum and maximum cost of materials and the construction cost of your structure as well as a minimum and maximum bound for 1st (fundamental), 2nd and 3rd the periods of the structure. In the optimization process will be taken into account the above general constraints in addition to the other geometric constraints (design bounds).

NOTE:

- ⚠ In the optimization process will be taken into account the above general constraints in addition to the other geometric constraints (design bounds).
- ⚠ The “-1” value in the above fields stands for the inactive limit.

Using the following tab "Deformations"

the following dialog box appears,



Deformation

Formulation Constrains Deformati... X

Drift

Upper limit 0.5 %

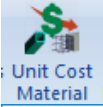
OK Cancel

You set an upper bound value of the drift of the floors of the structure. The best solution that will emerge will also satisfy this limitation.

1.2.2 Material Cost Unit

Using this command allows you to specify the cost per unit of the different materials supported by the software.

The supported materials are depicted in the dialog box below



Unit Cost Material

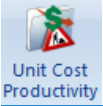
Concrete	70	currency / m3
Steel rebar	0.7	currency / Kg
Structural Steel	0.9	currency / Kg
Aluminium	3	currency / Kg
Wood	0.7	currency / Kg
Masonry	60	currency / Kg

OK Cancel

These values are included in the calculation of the total material cost of the structure.

1.2.3 Productivity Cost Unit

By using the above command it is possible to specify



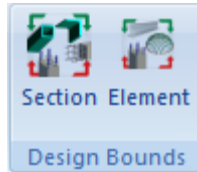
Unit Cost Productivity Rates

Beams	
Concrete (hrs/m3)	2.4
Rebar (hrs/Kg)	12
Structural Steel (hrs/Kg)	40
Columns	
Concrete (hrs/m3)	3
Rebar (hrs/Kg)	12
Structural Steel (hrs/Kg)	40
Slabs/Walls	
Concrete (hrs/m3)	1.6
Rebar (hrs/Kg)	8.5
Structural Steel (hrs/Kg)	2
Labor cost (currency/hrs)	15

OK Cancel

The hours spend for the production of each type of structural element (hours/unit of production) per material. In the last field "Labor Cost", you enter the value of the labor cost per hour.

1.3 Design Bounds



Design Bounds in the Optimization Process

The method followed for the definition of the design bounds on the optimization procedure of structures analyzed and designed in SCADA Pro follows a logical hierarchy. It starts from general constraints applied to each type of structural elements (columns, beams, slabs), continues in a more detailed definition of design bounds per cross-section and continues with the definition of design bounds per structural elements resulting in a grouping of individual linear or surface elements. These design bounds settings include minimum and maximum dimensional limits and their step change, the "locking" of a dimension, i.e. they remain unchanged whether they belong to one type of cross-section or certain structural elements.

More precisely, the first level of definition of the range of the cross-section dimensions, within which the values of the dimensions of the structural elements obtained, is applied using the Design Bounds command, which is part of the Basic Settings subsection.

The next two commands are the Design Constraints subsection

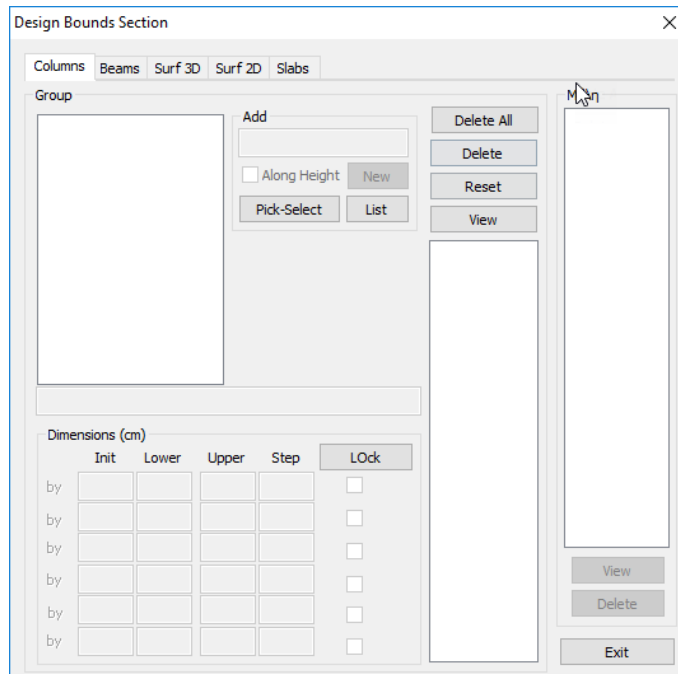
The following two commands are subsections of the category Design Bounds.

1.3.1 Cross-sections

The first "Cross-sections" command refers to the definition of bounds and limitations per cross-section type.

When selecting the command, the following dialog box appears.





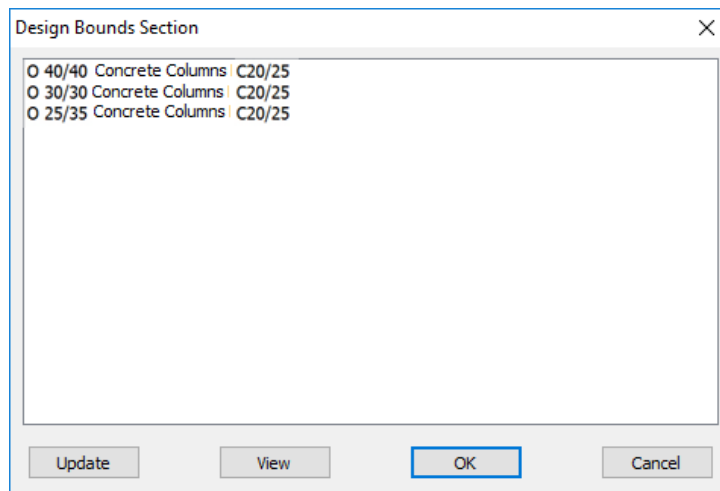
The insertion and processing of cross-sections are applied per type of structural element.

1) In the first tab “**Columns**”, the way of inserting the cross-sections can be done in two ways:

With a selection from List and or with graphically.

The graphical selection is done with the "Pick-Select" button.

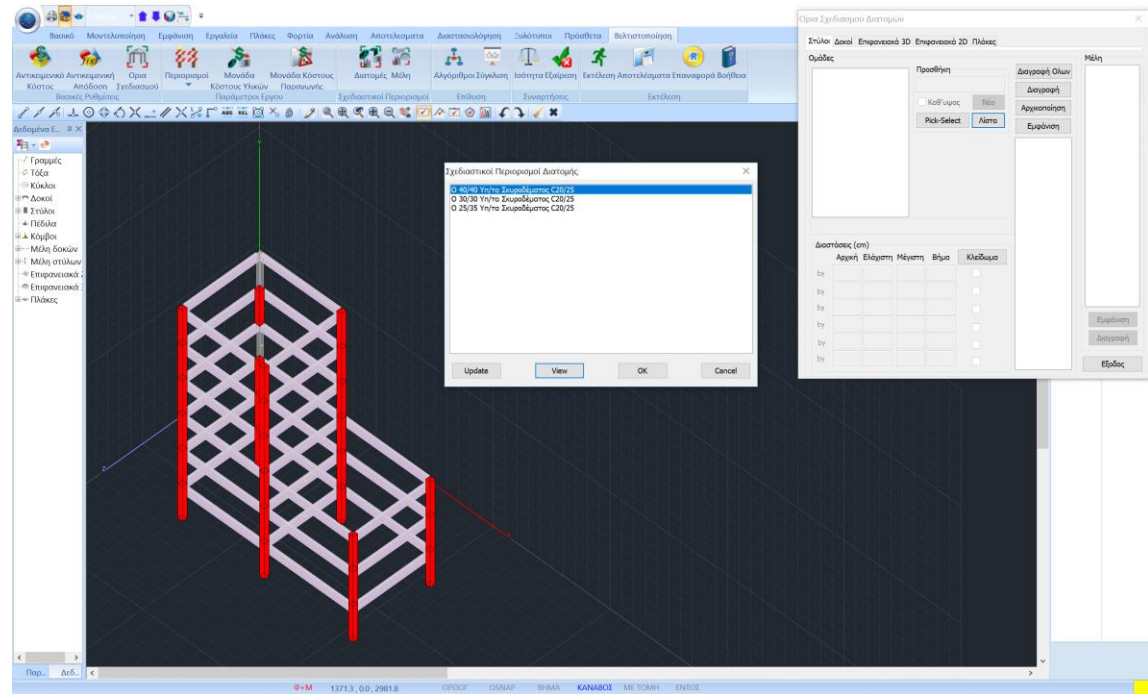
Clicking the "List" button, the following dialog box appears



where all types of cross-sections included in the structure are displayed. The criteria for dividing the cross-sections are:

- Type of the cross-section
- Layer
- Material (Type & Class)

The selection is available for one or more cross-sections. After selecting the cross-section by pressing the "View" button, the structural elements with this cross-section appear in the model with red color.



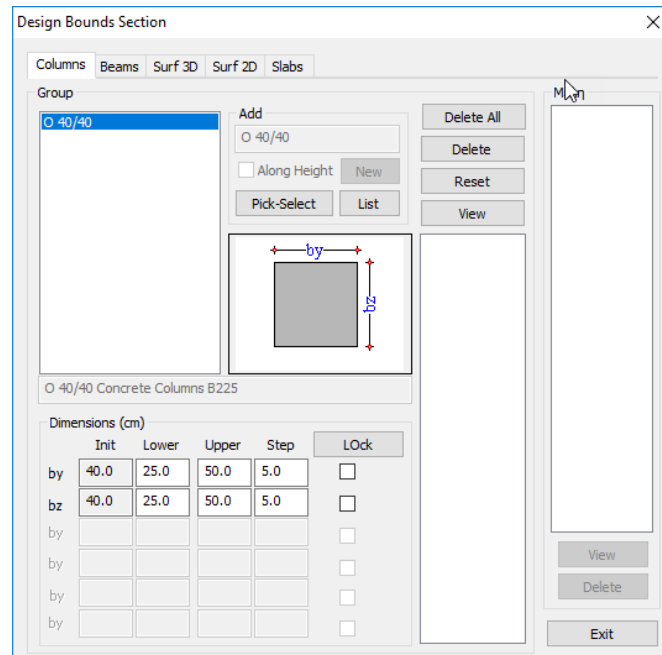
Click on the "Update" button and automatically the list of cross-sections is updated in case of changes applied since the first appearance of the list.

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

The other way of selecting cross-sections is by clicking on the "Pick-Select" button and then selecting a cross-section with the mouse. Next, this cross-section is also inserted in the "Groups" field.

ATTENTION!

The selection here only applies to cross-sections of columns, so in the 3D view of the three-dimensional model, the view of natural cross-sections should be activated.



Selecting a group from the list “Groups”, the dimensions of the corresponding cross-section are displayed in the “Dimensions” field according to the figure appears.

More specifically, the first column includes the values of the initial dimensions, the second the minimum dimensions, the third the maximum, and the fourth the change step.

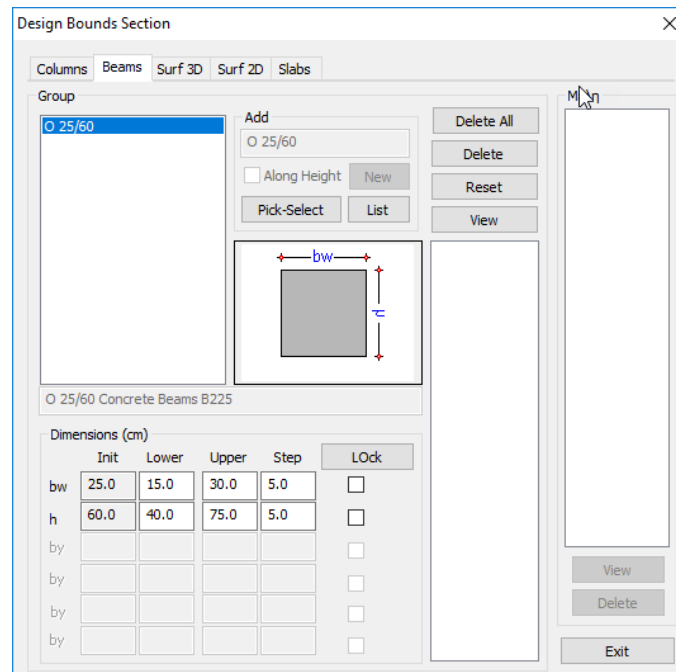
The "Lock" column locks this dimension to keep it unchanged at its original value. By clicking on the "Lock" button all dimensions are checked, i.e. the entire cross-section geometry is locked.

The default values of the section bounds in columns “Minimum”, “Maximum” and “Step” are those defined in the “General Design Parameters” that initially obey all the elements.

Using the "Delete All" button, all the cross-sections in the list are deleted, while using the "Delete" button only the selected cross-section is deleted. With the "Initialize" button, the initial default defined bounds are restored to the dimensions of the cross-section.

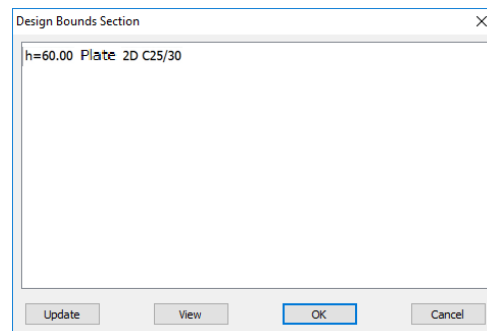
Finally, when selecting the "Show" button, the elements that have the cross-section will be displayed in red color in the 3D view of the structure.

2) The section “Beams”



Is the same procedure as for the columns.

The next two sections are related to a cross-section of 2D and 3D finite surface elements “**Surf 3D**” and “**Surf 2D**”. The definition of the optimization parameters is the same in both cases. So, choosing the “Surface 3D” section can add surface cross-sections either from the “List”

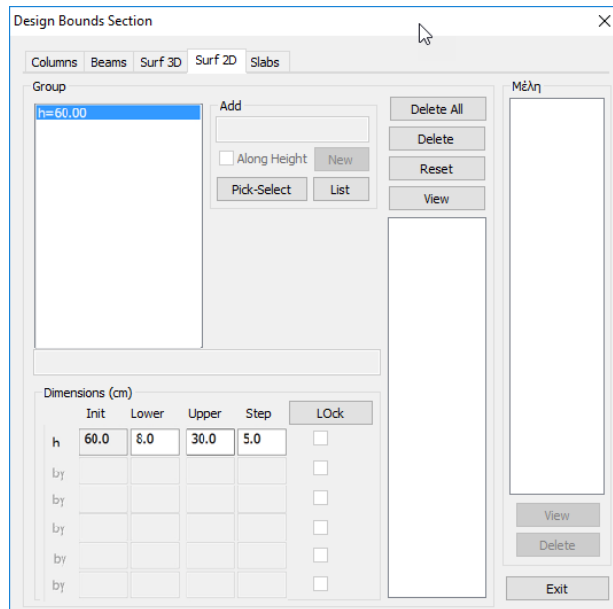


or graphically in the user interface. The selection process from the “List” works like in the previous section “Columns”.

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

- Thickness
- Material
- Layer
- Direction (horizontal/vertical)

By inserting the cross-section

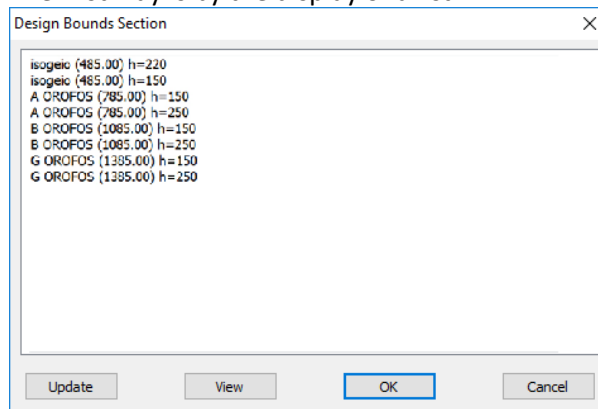


the initial thickness, the lower and the upper bound and the change step appear in the “Dimensions” section. Finally, there is the option to lock the thickness dimension.

3) The last section is about solid “**Slabs**”.

The selection here can also be applied in two ways:

The first way is by the display of a list

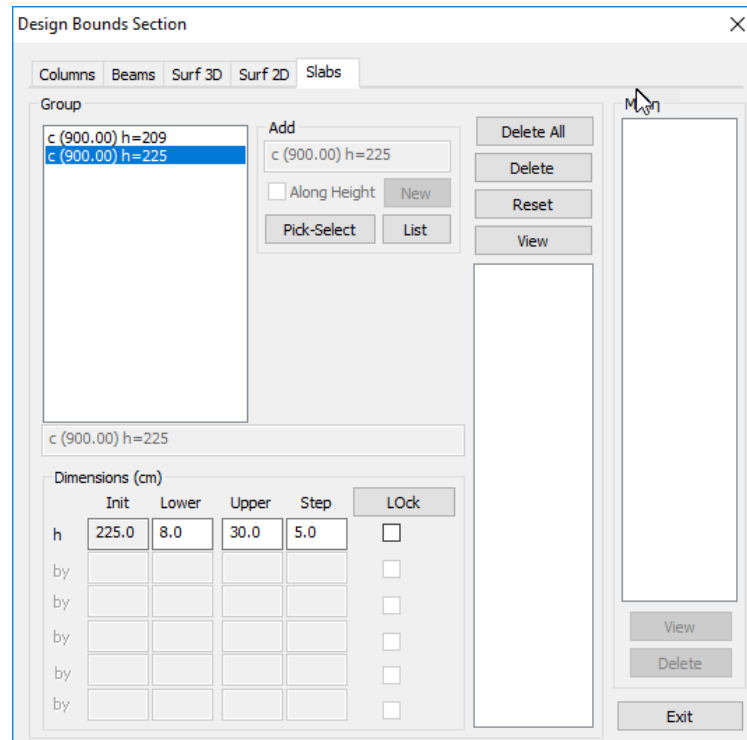


Where it includes all the slabs of the structure. The grouping of the cross-sections is based on two criteria:

- The floor
- The different thickness dimension

The list shows the name of the floor, then the height altitude and the thickness “h” of the slab.

By selecting one or more slabs, they are added to the group.



The selection of the slab or slabs can also be done with the "Pick-Select" button. To select, you need to turn the view of the structure in 2D mode.

Then, by selecting from the "Groups" field of the slab that you wish to edit, you will see the initial thickness in the "Dimensions" field, also the minimum and maximum bound of the thickness and the option to lock the thickness dimension of the particular slab.

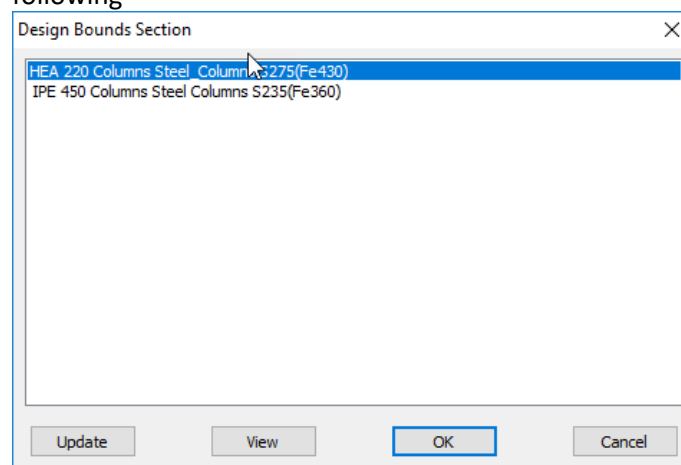
Steel Sections

The grouping method for the structural steel frame sections is the same applied to the reinforced concrete in the frame section of reinforced concrete.



EXAMPLE

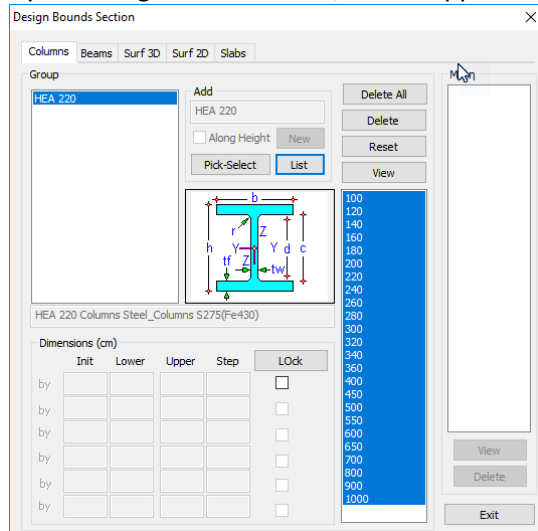
For example, for steel columns, the corresponding list displayed using the "List" command is the following



and includes all types of cross-sections included in the structure. The grouping is based on the following criteria:

- Type of cross-section
- Layer
- Class of material

By selecting a cross-section, it now appears in the “Groups” field.

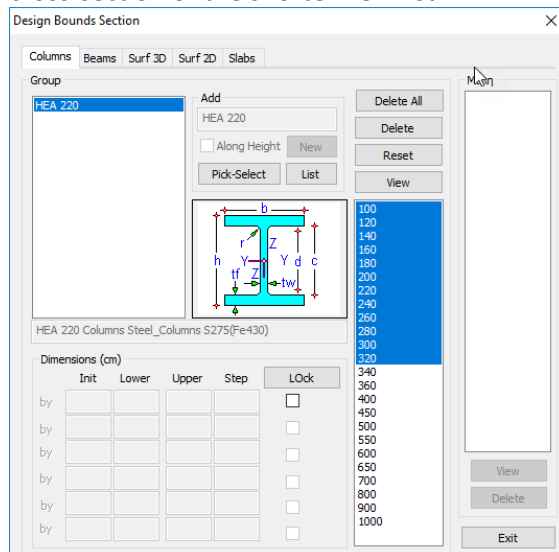


the shape of the cross-section with the corresponding dimensions appears as well as the list of the cross-sections of the particular shape. The blue color in the panel of the cross-sections means that the cross-section can vary across the list of sections of the particular type of cross-section.



EXAMPLE

For example, the IPE450 cross-section is selected, as shown in the above figure. The entire width of the dimensions of the IPE cross-section in the library has become blue, which means that the cross section can vary across the range of sections shown in the list with blue color. Here you can either define a new shorter range by clicking while pressing the Shift key, the first and the last cross-section of the shorter new list.



Or by selecting, while pressing the Ctrl key, specific cross-sections, that will compose the new shorter range.

Finally, the following commands apply to all sections and apply to all cross-sections:

- "Delete All" deletes all groups already added.
- The "Delete" command deletes the selected group.
- The "Initialize" command restores the bounds to their original values as defined in the "General Parameters" section.
- Finally, the "Show" command displays the selected items in red color.

Finally, it is worth noting that using the "Pick-Select" command, if you choose a different type of element than the one in the section you are in, the software drives you automatically to the corresponding type of element.

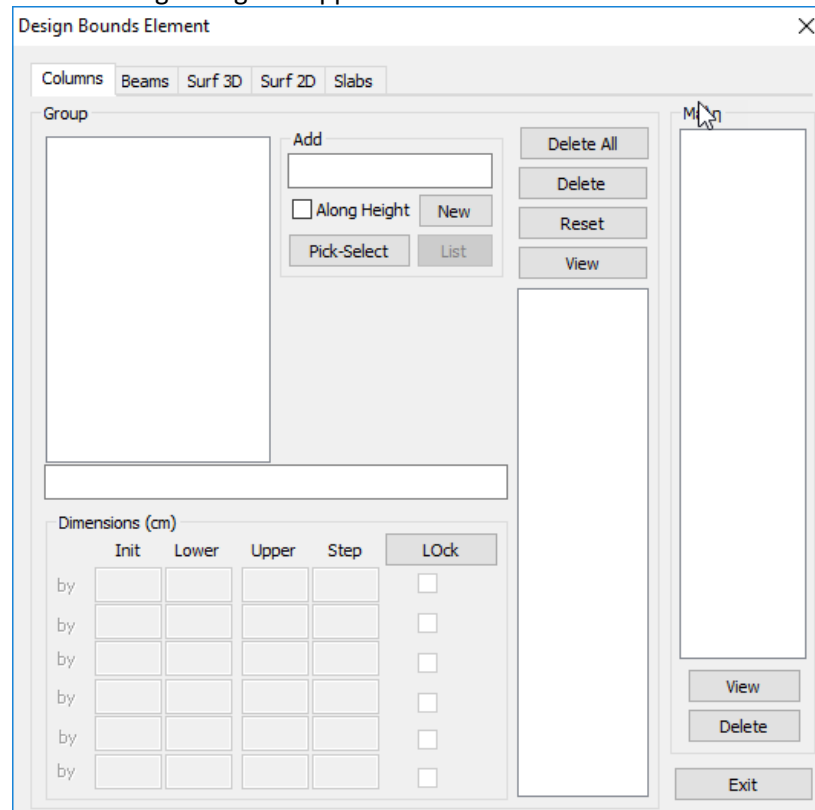
1.3.2 Elements

The next command to set a group or groups of elements to set the design bounds per structural element.

By selecting the command "**Elements**"



the following dialog box appears

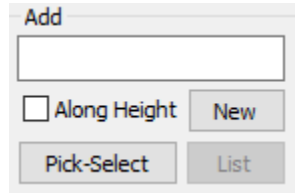


where you are given the ability to create groups of elements for which you want to impose specific boundσ during their structural design or even to lock their dimensions. The the dialogue box is the same as the one for grouping per cross-sections, that is, divided into modules (Tabs) according to the type of the element.

The function is the same in all modules:

You start by defining a group or groups of elements that must have the same cross-section.

- 1) In the "**Columns**" section, you start by defining the name of the group you are about to create. The group may contain one or more individual elements that should have the same initial dimensions and will obey specific lower and upper limits



After writing the name in the field "Import", press the "New" button to create the group and select "Pick-Select" to graphically select the members you want to join this group. The cross-section of the first column that you will show will also determine the type of the cross-section of the group you will create.



EXAMPLE

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

NOTE

- ⚠ If you do not complete a name for your group and say "New", the program initially puts the English question marks "??" and then names the group based on the cross-section of the first item you choose. Of course, you can modify the group name.

By selecting these items, they are listed 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 on the structural model with the "Show" button.

- ⚠ The definition of a group of elements can be done either by selecting all elements for the first time or by adding elements to an existing group. Select the existing group from the list and select the elements you want to add in and then click the "Pick-Select" command.

When the feature ☐ Along Height is checked, then a series of columns along height can be selected automatically, by selecting only a single column of the series.

In the field "Dimensions" parameters are the same as previously described for the cross-section dialog box.

Dimensions (cm)					
	Init	Lower	Upper	Step	LOCK
by	40.0	25.0	50.0	5.0	<input type="checkbox"/>
bz	40.0	25.0	50.0	5.0	<input type="checkbox"/>
by					<input type="checkbox"/>
by					<input type="checkbox"/>
by					<input type="checkbox"/>
by					<input type="checkbox"/>

Finally, the following commands apply to all sections and apply to all cross-sections:

- "Delete All" deletes all groups already added.
- The "Delete" command deletes the selected group.
- The "Initialize" command restores the bounds to their original values as defined in the "General Parameters" section.
- Finally, the "Show" command displays the selected group of elements in red color.

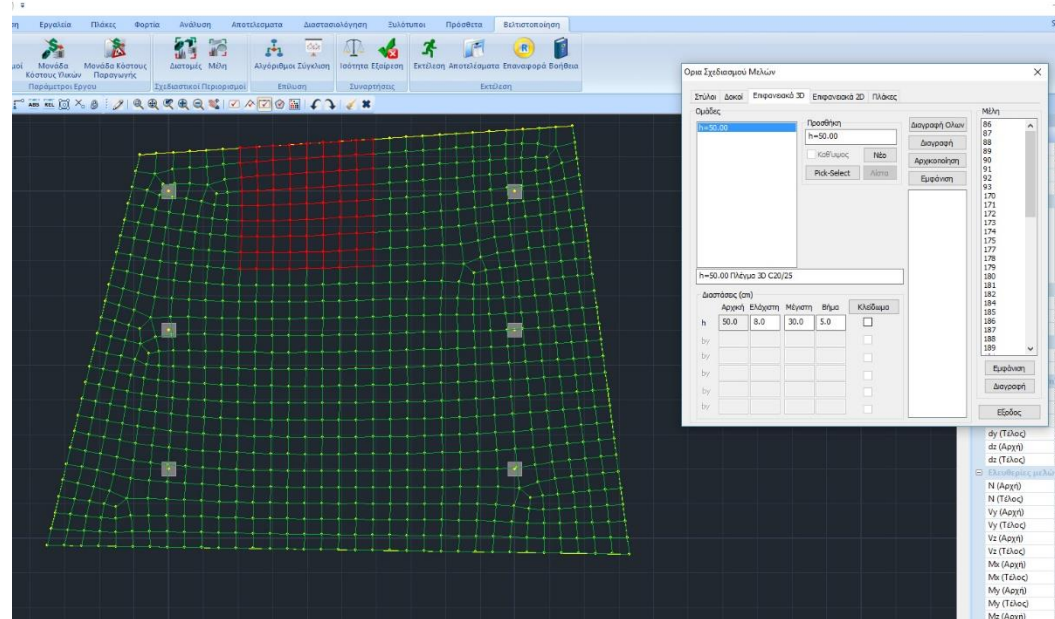
2) For the **Beams**, the same ones previously mentioned for the columns apply.

3) For the next two sections **2D** and **3D Surface** finite elements, the following apply:

The team or groups you create can consist of one or more surface elements. It is recalled that in the surface elements the only design parameter that can be determined is the thickness.

So you define a new group and with the graphical selection tools (individually, with a window, etc.) you select the surface that will be the group you will create.

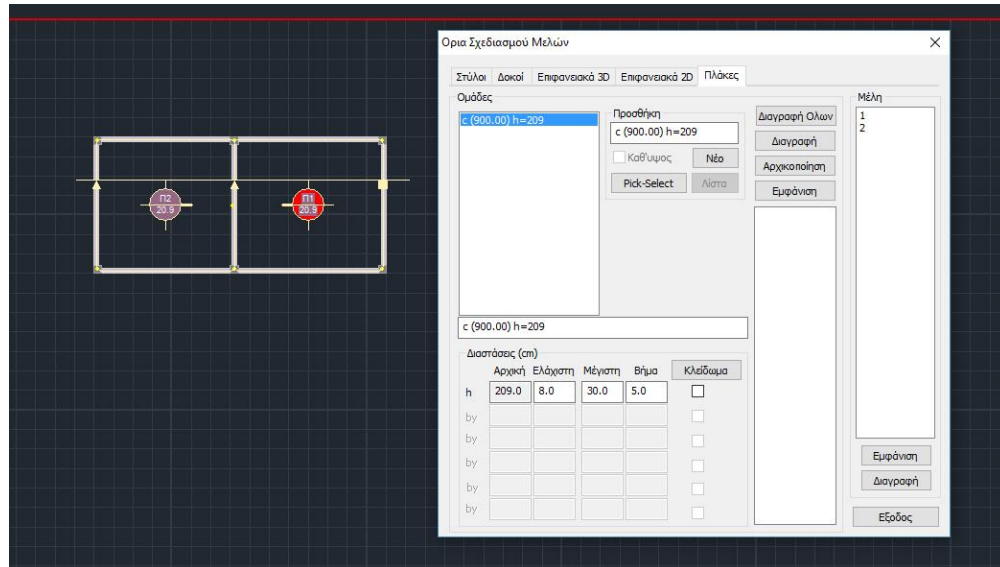
So you define a new group and the graphical selection tools (individually, with a window).



Surface elements can belong to different groups or subgroups but must have the same initial thickness, the same class of material and belong to the same layer.

By selecting them, they appear in the member's list.

In the slabs section, you create a group where you put slabs of the same thickness that belong to the same floor.



EXAMPLE

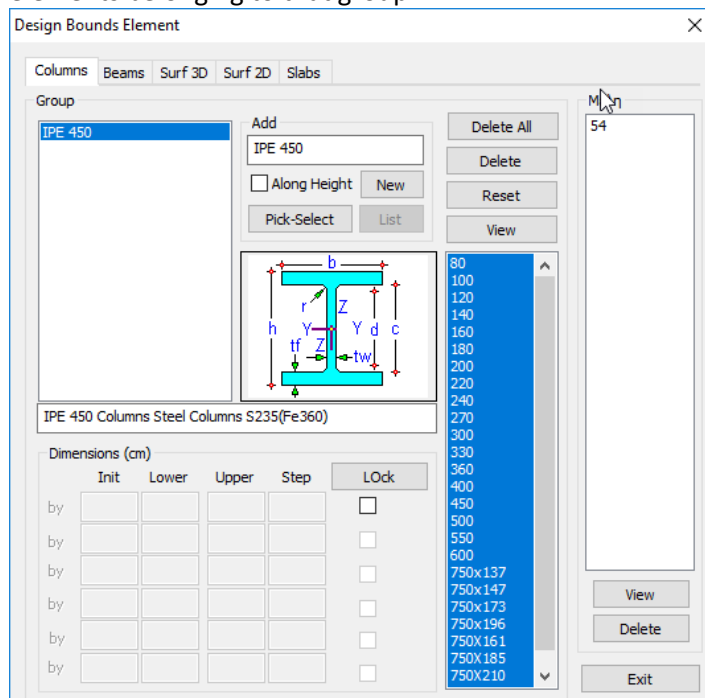
For example, the above picture shows the definition of a group of slabs with 209 mm thickness belonging to the level “c” at an altitude of 900.00 cm and includes two slabs (1, 2) as shown in the list of elements.

Finally, to indicate that for the display of the slabs and their graphical selection, you have to go to the corresponding section of the slabs.

Steel Sections

For the structural steel sections, the definition of the groups is also done for beams and columns, in the same way as for the reinforced concrete cross-sections.

The choice of the first element determines the cross-section, the material and the layer of the elements belonging to that group.

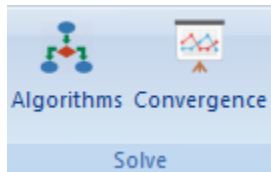


**EXAMPLE**

For example, in the above picture, a group that contains the columns 37 and 38 with cross-section IPE 450, belong to the layer Steel Columns and their material is S235 (Fe360). All of this information is displayed in the bar above the dimension field.

IPE 450 Υποστυλώματα Μεταλ. Υποστυλώματα S235(Fe360)

So all of the above tools and commands give you the option and flexibility to set design constraints on all members and in whatever way you want.

1.4 Solution

This section includes two commands:

“Algorithm”, where you select the algorithm to be used as the solution method for the optimization problem.

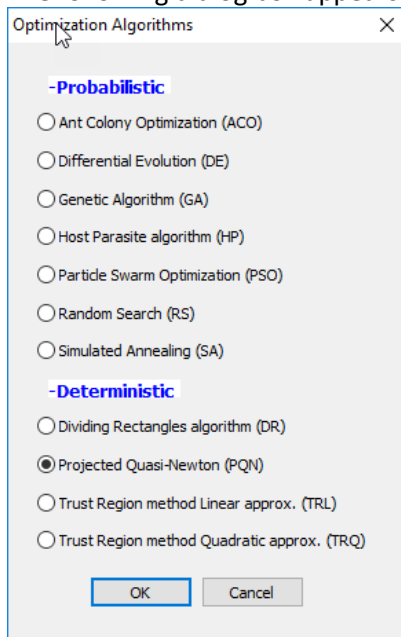
“Convergence”, where you specify the convergence criteria to be used as well as the analysis and design scenarios.

1.4.1 Algorithms

By selecting the command “Algorithms”



The following dialog box appears

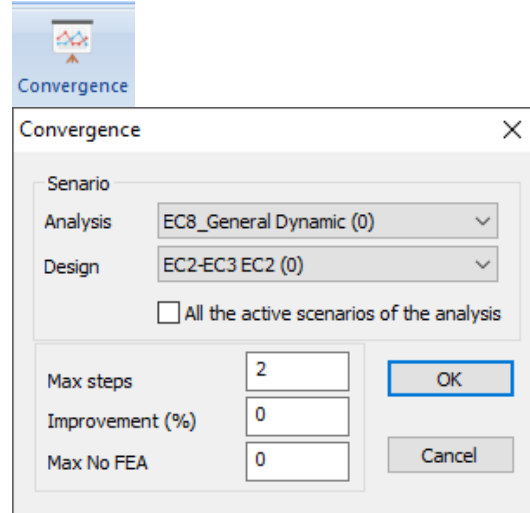


where you can choose the optimization algorithm from two categories

- Probabilistic
- Deterministic

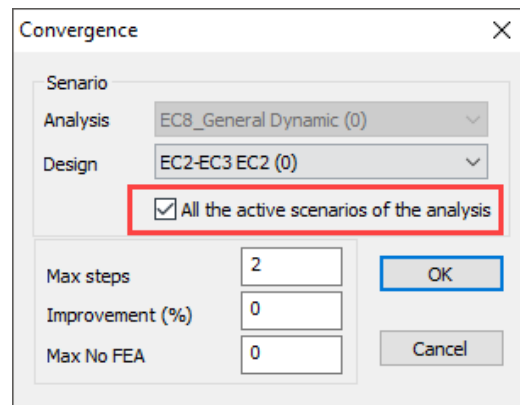
1.4.2 Convergeance

By selecting this command, the following dialog box appears



Where in the "Scenario" field you can specify the Analysis and Design scenarios to be used for optimization.

Activating the command «All the active scenarios of analysis», OCP runs all the ACTIVE analysis scenarios sequentially. Active scenario means what is not canceled (without an asterisk).



In the next section there are two convergence criteria:

Max Steps: Here you can state the maximum number of steps (repeating cycles) where there will be no improvement.

Max. Step Number: Set the maximum number of iterations

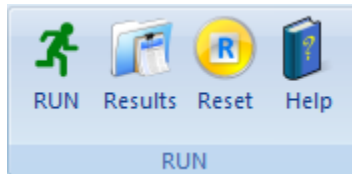
Improvement (%): Define the minimum rate of improvement to be considered as a lower limit to achieve convergence.

1.5 Functionalities



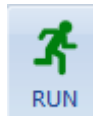
The two commands in this section will be activated in a future version of the software.

1.6 Run

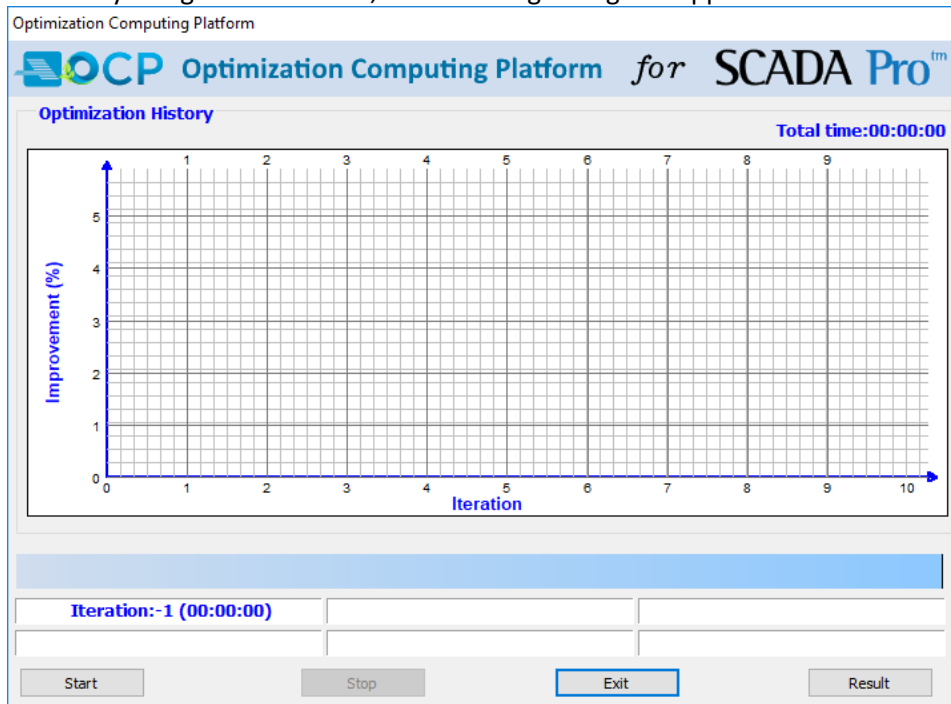


Η τελευταία ενότητα περιλαμβάνει εντολές που αφορούν στην εκτέλεση της διαδικασίας βελτιστοποίησης καθώς και στα αποτελέσματά της.

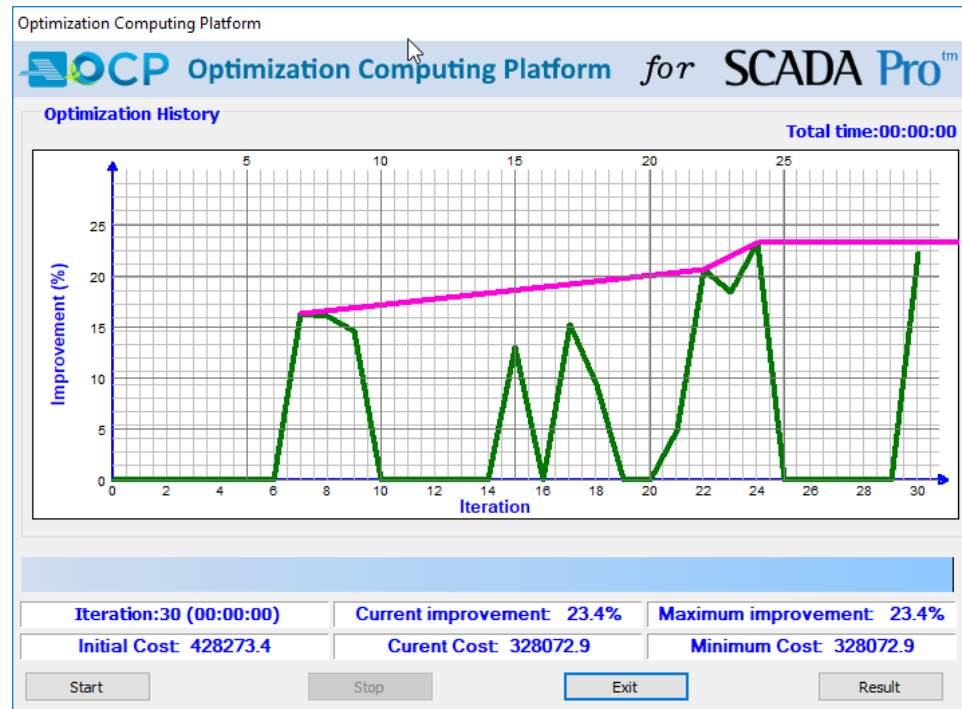
1.6.1 Run



By using this command, the following dialog box appears



Pressing the Start button starts the iteration process of optimization and the following graph is generated and the fields with the corresponding information are filled in.



By pressing the “Start” button starts the optimization iterative process.

The top of the dialog box shows the optimization history graph where the number of iterations is the horizontal axis and the percentage of the improvement of the structure is the vertical axis. Below the graph is the progress bar

Run Analysis

where messages describe the successive stages of the process.

Below the progress bar there are six fields that:

Iteration:2 (00:00:37)

The number indicates the number of iterations so far and in brackets the time that the current iteration runs.

NOTE:

The iteration with number 0 is the structure where the upper bounds of the design variables are applied, i.e. the structure with the largest cross-sections. The iteration with number 1 is the structure where the lower bounds of the design variables are applied, i.e. the structure with the smallest cross-sections. The iteration with number 2 is the initial structure designed by the designer and considered as the reference structural model.

Current improvement: 0.0%

The percentage indicated in the “Current improvement” field relates to the improvement of the structure of the current iteration relative to the initial structure (reference design).

Maximum improvement: 0.5%

The percentage indicated in the “Maximum improvement” field relates to the improvement of the best structure achieved so far in one of the previous steps relative to the initial structure (reference design).

Initial Cost: 423364.8

The amount indicated here refers to the total construction cost of the structure of step 2 (reference design).

Current Cost: 195828.2

The amount indicated here refers to the total construction cost of the structure in the current iteration.

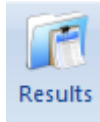
Minimum Cost: 195828.2

The amount indicated here refers to the total construction cost of the best optimized structure achieved so far.

The optimization process ends when convergence is reached or ends if you press the key

Stop

1.6.2 Results



By using this command, the following dialog box appears

Results

Step Number	Failure Degree	Total Cost
1	Max	541796.69
2	Min	229542.69
3	Init	428273.36
4	1.69	332170.88
5	1.52	382265.20
6	1.57	342169.11
7	0.00	358089.08
8	0.00	359265.91
9	0.00	365662.81
10	2.00	363194.19
11	1.33	353226.17
12	3.40	336521.59
13	1.77	324378.33
14	1.41	327128.60
15	0.00	372206.74
16	1.57	365670.47
17	0.00	262667.02

Report

☒ Cover Page

☒ Technical Report

☒ Overview

☒ Cross Section Data

☒ Analytical Results

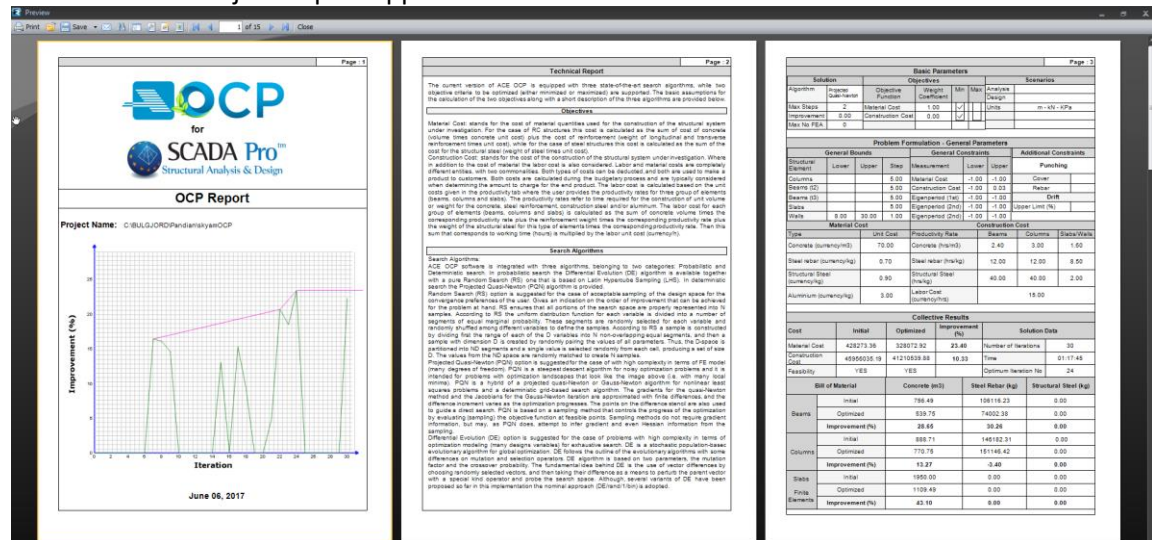
Show Report

Exit

Where the left side shows three columns with the optimization steps. The first column includes the step sequence number, the next step, the penalty grade for each step. The step with the optimum solution has a penalty grade of 0 and is indicated in blue color.

Results			Report	
Step Number	Failure Degree	Total Cost	<input checked="" type="checkbox"/> Cover Page	<input checked="" type="checkbox"/> Technical Report
15	0.00	372206.74	<input checked="" type="checkbox"/> Overview	<input checked="" type="checkbox"/> Cross Section Data
16	1.57	365670.47	<input checked="" type="checkbox"/> Analytical Results	
17	0.00	362667.83	Show Report	
18	1.08	349312.93		
19	1.39	325340.50		
20	2.00	340316.10		
21	1.14	336841.38		
22	0.00	339677.14		
23	0.00	349112.90		
24-24	0.00	328072.92		
25	1.36	354233.43		
26	1.32	343081.46		
27	2.00	322684.78		
28	1.39	342327.68		
29	1.51	336934.56		
30	0.00	332678.78		
31	0.00	328072.92		
			Exit	

On the right side, you select which parts you want to include in the Project Report. By selecting the button the Project Report appears.



The screenshot displays the SCADA Pro OCP Report interface, which is divided into three main sections:

- Left Panel:** A line graph titled "Improvement (%)" showing the percentage of improvement over 30 iterations. The graph shows a general upward trend, reaching approximately 25% improvement by iteration 30. The x-axis is labeled "Iteration" and the y-axis is labeled "Improvement (%)".
- Middle Panel:** A "Technical Report" section containing detailed text about the optimization process, including the use of the OCP algorithm and the results of the optimization.
- Right Panel:** A "Basic Parameters" table summarizing the optimization results. The table includes columns for "General Results", "General Constraints", and "Additional Constraints".

The "Basic Parameters" table is as follows:

General Results		General Constraints		Additional Constraints	
Iteration	Improvement (%)	Iteration	Improvement (%)	Iteration	Improvement (%)
Initial	0.00	Iteration	0.00	Iteration	0.00
Optimal	25.00	Iteration	0.00	Iteration	0.00
Feasibility	YES	Iteration	0.00	Iteration	0.00

The Printed Report is a complete and detailed report that includes the theoretical background, the problem parameters, the convergence criteria, analytical presentation of the final cost of the structure, the percentage of improvement achieved as well as the variations in the cross-sections of all structural elements.

1.6.3 Reset



Using this command, the optimization procedure that you have run before and all the optimization parameters return to their original default values.