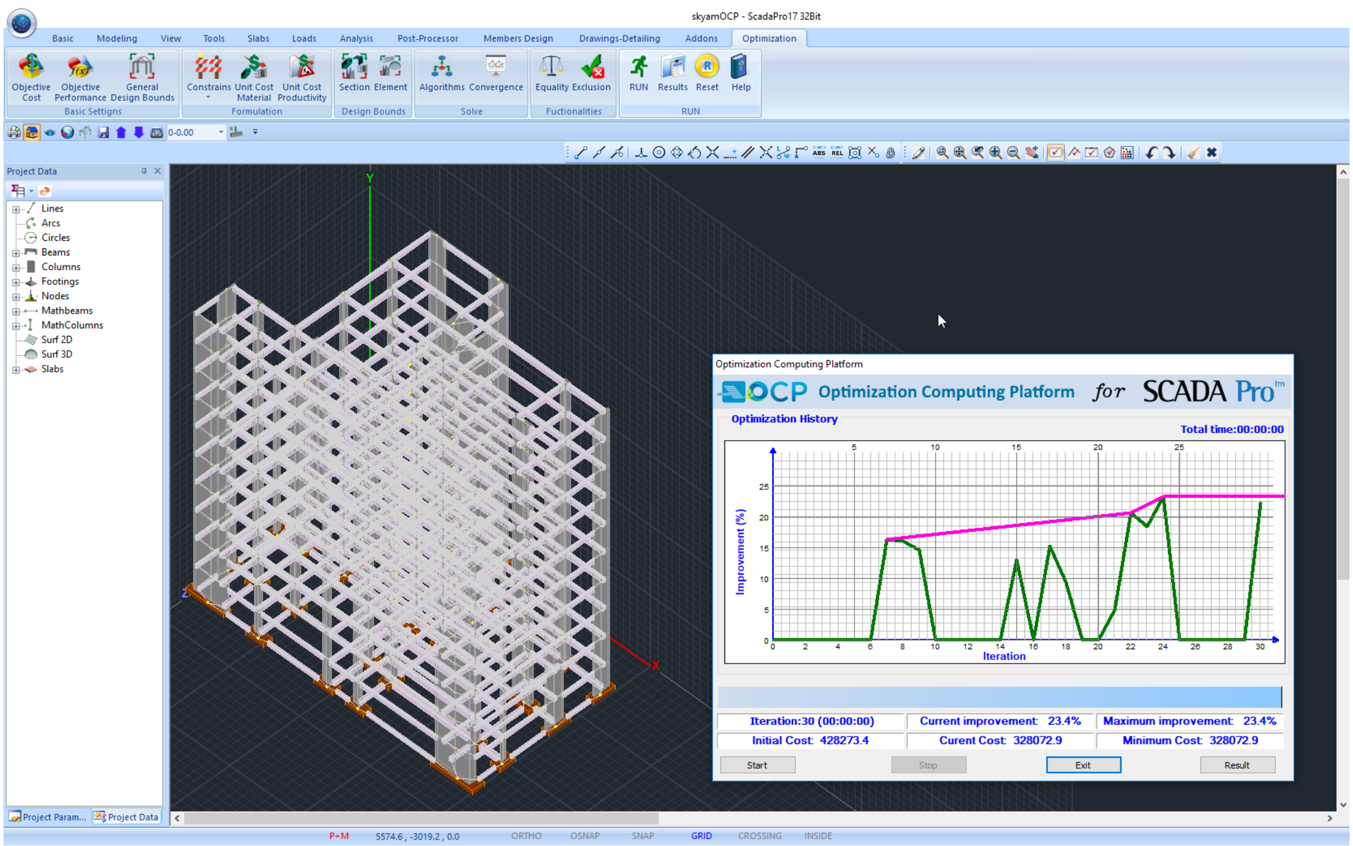
SCADA Pro™ 18

Structural Analysis & Design

Example 8
Optimization



ACE HELLAS

INTEGRATED SOLUTIONS

CONTENTS

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.

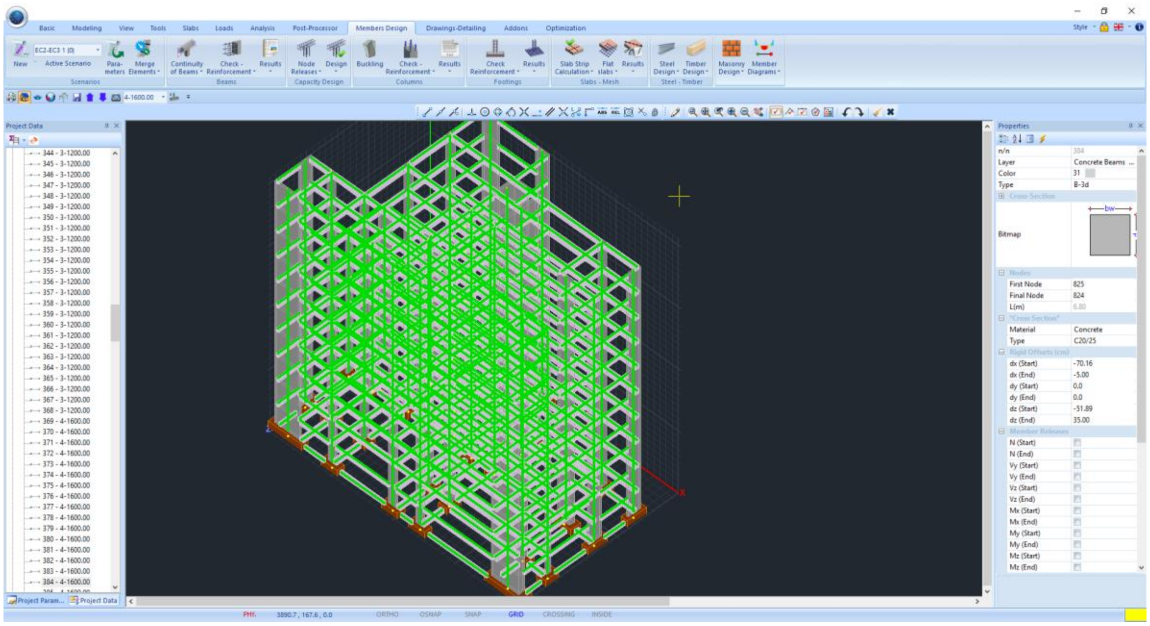
SCADA Pro OCP module provide:

- **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. Before the Optimization

The process of optimization of a structure presupposes the modeling and the design.

This example concerns a ten storeys reinforced concrete structure (C20/25 and S400s), with footings and connection beams.
For the Modeling we used a dwg file and the necessary commands of Scada for editing.
The slabs were imported automatically. Loads and slab reactions came next.
Then we run a eurocode dynamic analysis and we created the combinations.
The last step was members design using a eurocode scenarios.

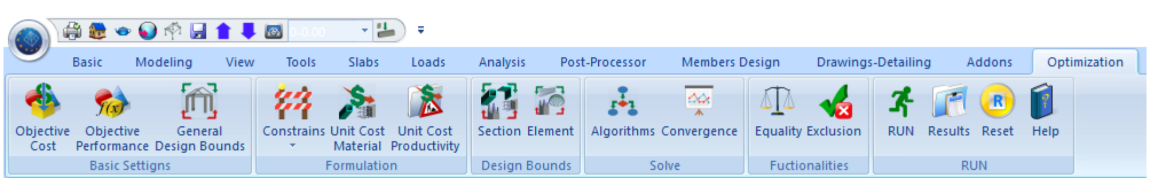


The aim in each project is the sufficiency of all the elements under the checks provided from the Regulation. This example demonstrates sufficiency both in beams and columns, as well as in the slabs.
A project like this could be considered completed and ready to proceed in stamps and designs.

The new innovative SCADA Pro OCP module, with the main goal to reduce construction and material cost, offers us the opportunity to re-examine the already sufficiently project, using smaller cross sections, always consistent with the required performance, reliability, quality and safety within an innovative technological framework.

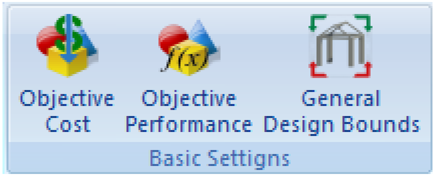
2. Optimization

Complete the project, and before printing, open the unit 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.

2.1 Basic Settings

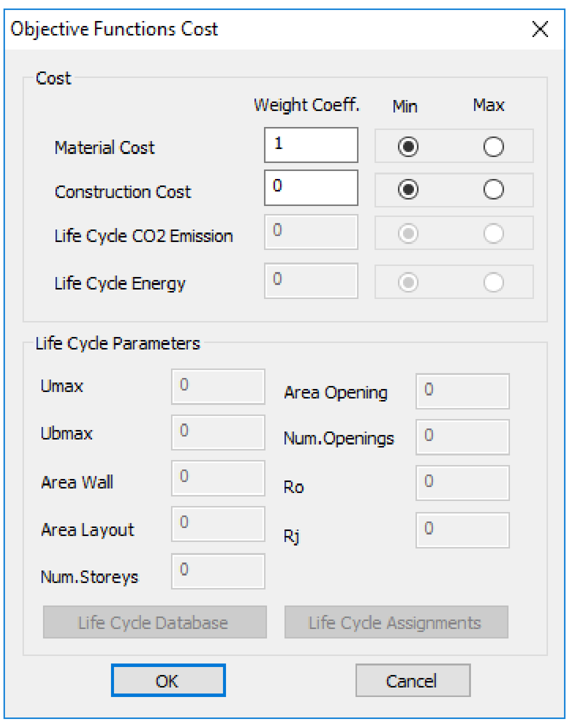


The first command subsection includes the following three commands:

- Cost Objective Function
- Performance Objective Function
- Design Bounds

2.1.1 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 determines the percentage of participation of each criterion in optimization. The sum of weight coefficients of the criteria should be equal to the unit. Next to the Weight Coeff. Field, there is a "Min" and "Max" indication for each criterion to indicate if the goal of optimization is to minimize or maximize each criterion of our choice.

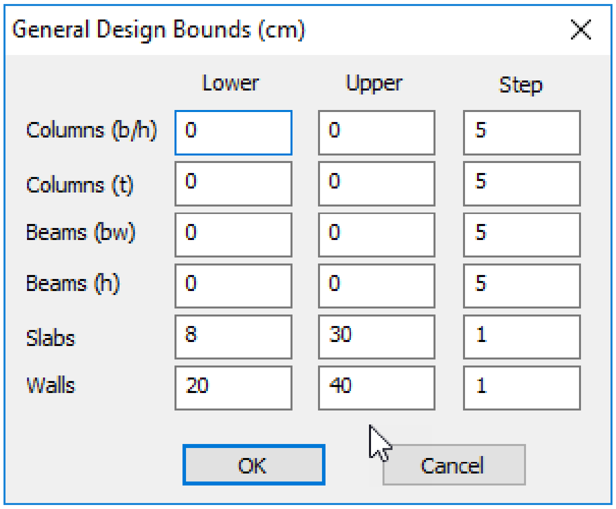
- For this example as objective functions was considered only the **Cost of Materials**.

2.1.2 Performance Objective Function

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

2.1.3 Design Bounds

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



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 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. This includes solid slabs as well as those simulated with finite surface elements.

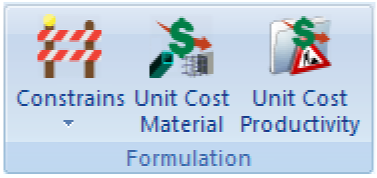
Finally, in the "Walls" section, the bounds of vertical elements simulated with finite surface elements are defined.

NOTE:

The value 0 in the fields of the maximum and the minimum bound means that the program receive 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.

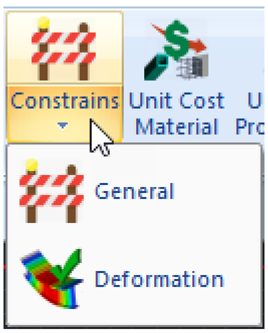
For this example, keep the value 0 so that the program specifies, by automatic calculation the dimensional change limits by changing the initial dimensions by 30%.

2.2 Project Parameters



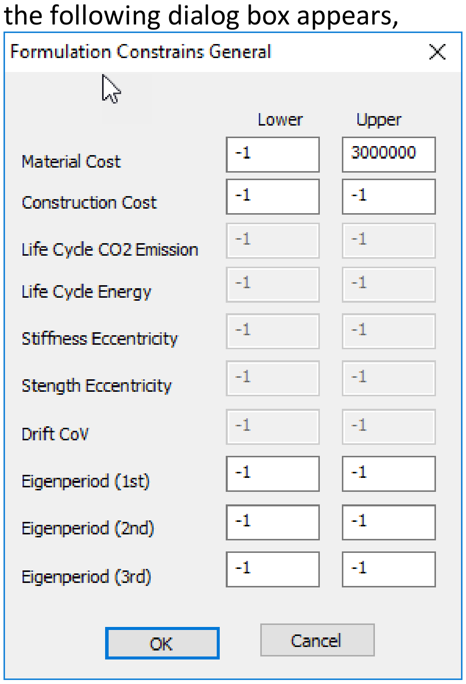
The next subsection is about the project parameters and includes constraints imposed by the designer as well as the units of the material and productivity cost.

2.2.1 Constraints



Using the "General Constraints"

the following dialog box appears,



In this window you can specify the sizes to be used as additional limitations of the problem setting a minimum and a maximum value of the material costs, the construction cost of your structure, as well as a minimum and maximum bound for the 1st (fundamental), 2nd and 3rd Egenperiods 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).

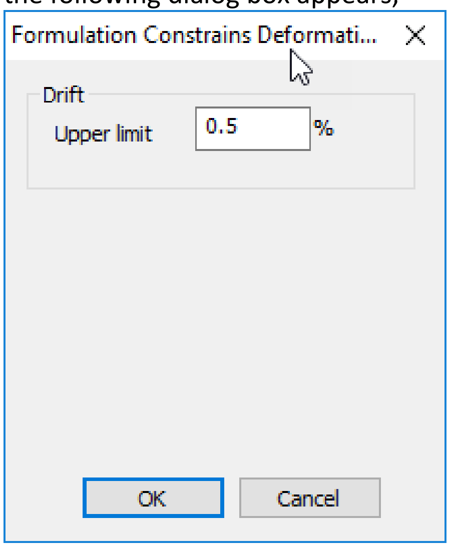
NOTE:

- ⚠ 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 inactive limit.

For this example, we set the maximum cost of 300,000 (in the currency of the place where it will take place) and we do not specify any other limitation, so we keep the -1 values in the remaining fields. This means that the resulting construction will have the lowest possible cost of materials, which will be less than 300,000 (local currency) and will satisfy the checks of the selected regulation.

Using the following tab "**Deformations**"

the following dialog box appears,

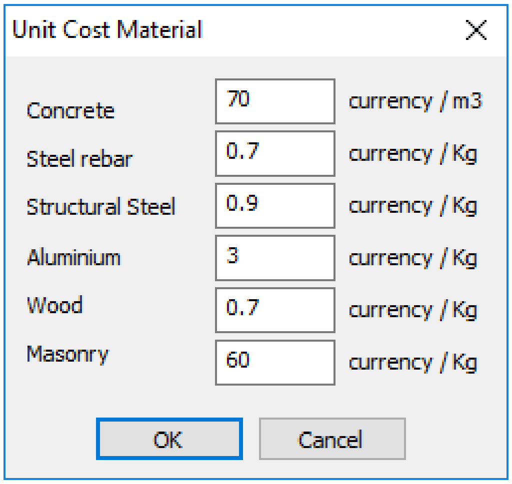


You set an upper bound value of the drift of the floors of the structure. The best solution that will emerge will also satisfy this limitation.

2.2.2 Material Cost Unit

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

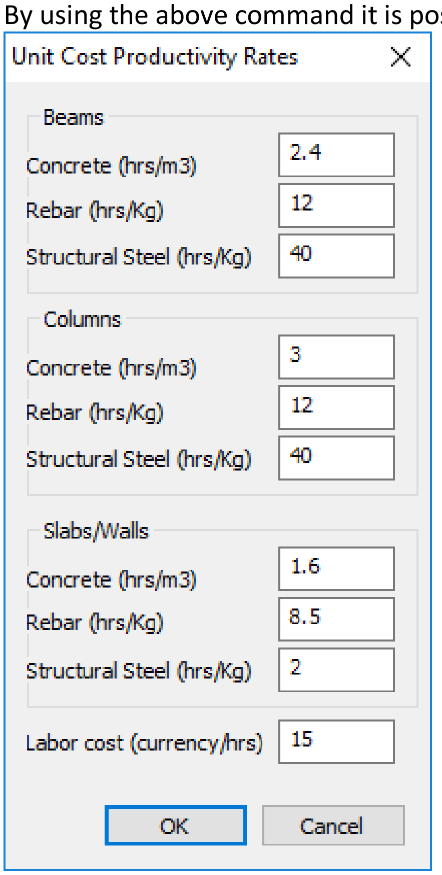
The supported materials are depicted in the dialog box below



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

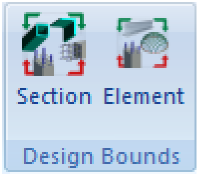
2.2.3 Productivity Cost Unit

By using the above command it is possible to specify



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.

2.3 Design Bounds



The Design Bounds are the limits of the design variables, which you can specify by section type or element type. The design variables are the sizes of the construction that change during optimization, that is, the dimensions of the structural elements, and the bounds are the range of each dimension.

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

-finish 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.

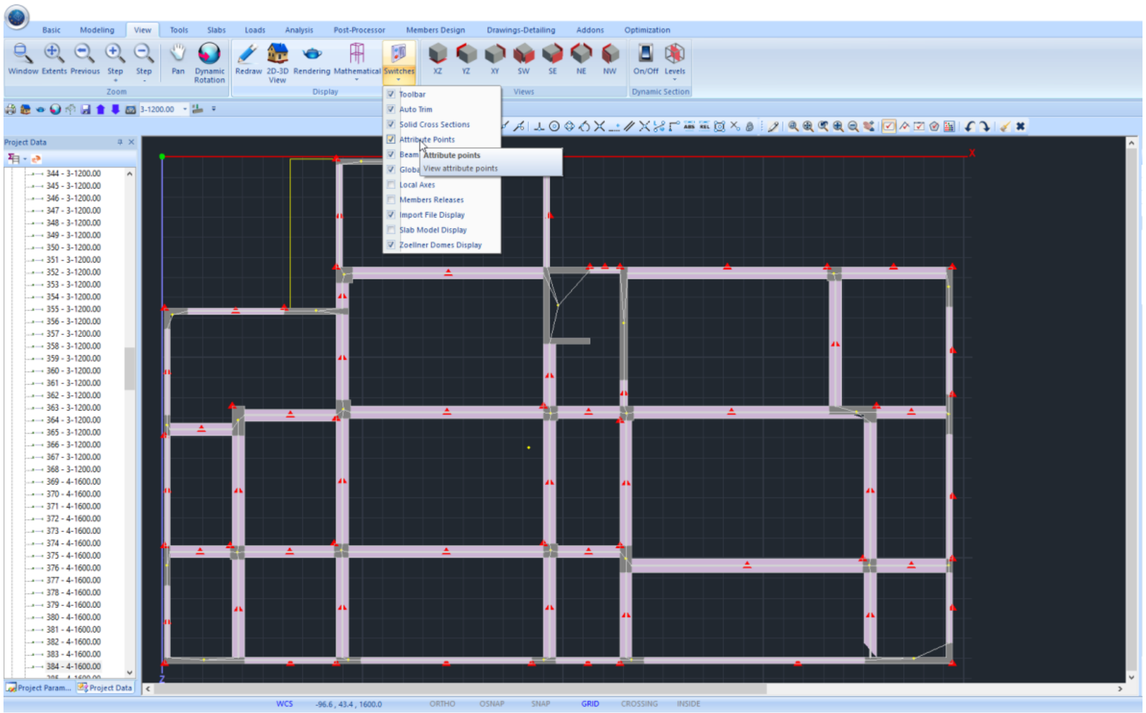
The next two commands are the Design Constraints subsection

IMPORTANT NOTE:

It is important to notify that all the variations of the cross-sections dimensions during Optimization are made always maintaining constant the Attribute Points.

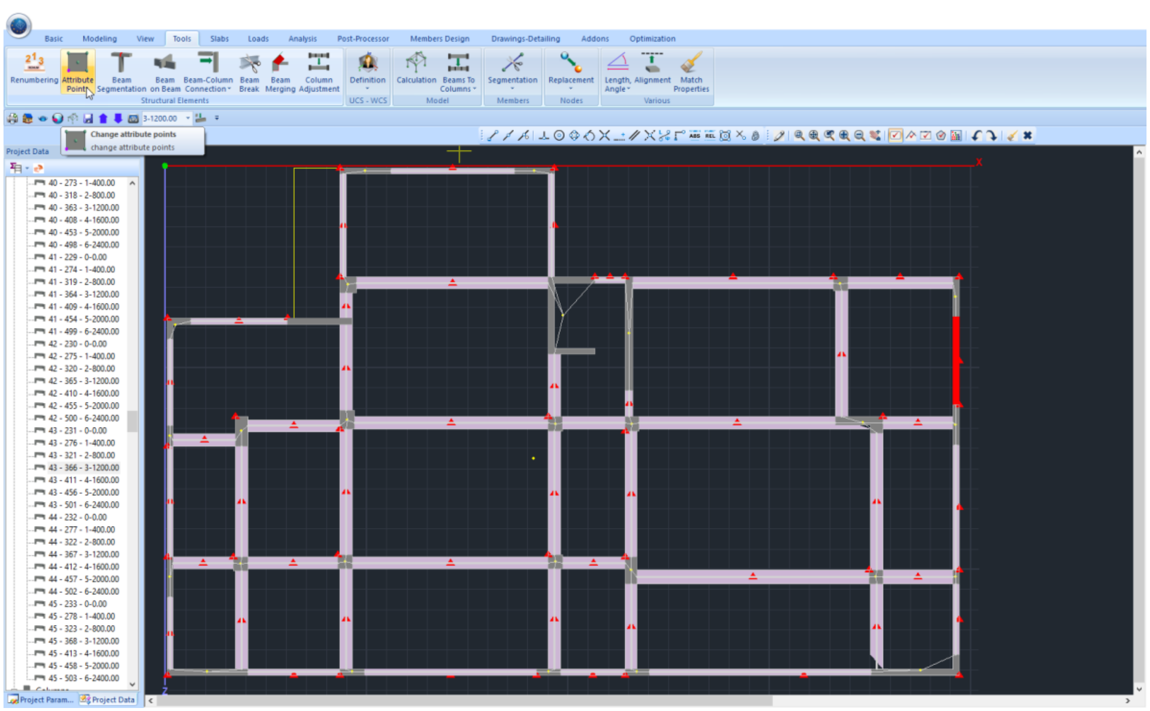
This means that before starting the optimization process, is necessary to adjust the Attribute Points of the sections, in a way that does not deform the floor plans, the outer boundaries and the alinement of columns and beams.

Inside the "View" unit, in "Switches", activate the Attribute Points and open all the floor plans to check if there is necessary to change them.



Using “Attribute Points” in “Tools” unit, you can modify the position of the Attribute Point in beams and columns.

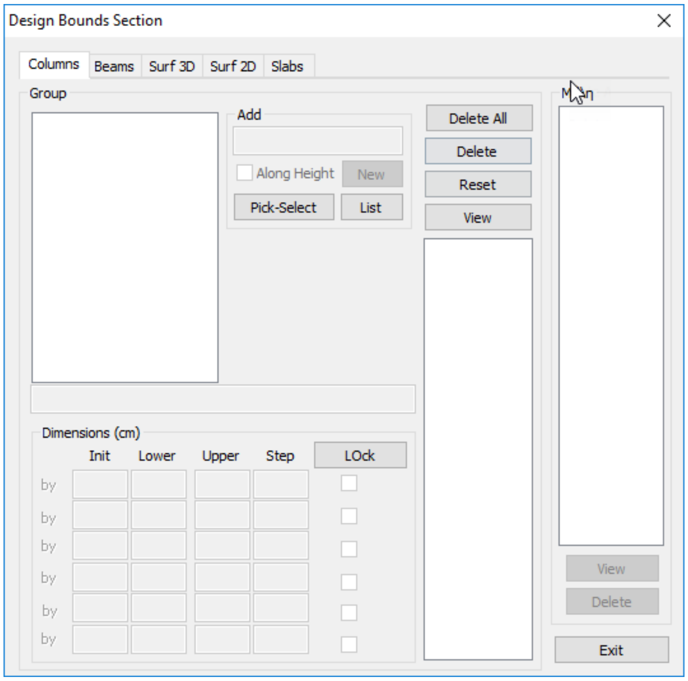
In this example, the “Attribute Points” of the outer columns and beams, were changed so that they belong to the outer perimeter.



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

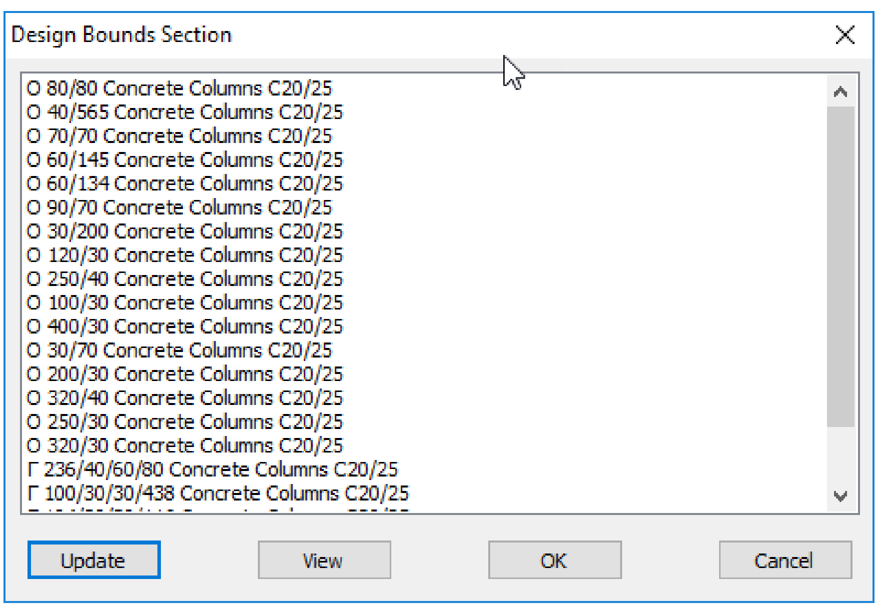
2.3.1 Cross-sections

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 is applied per type of structural element.

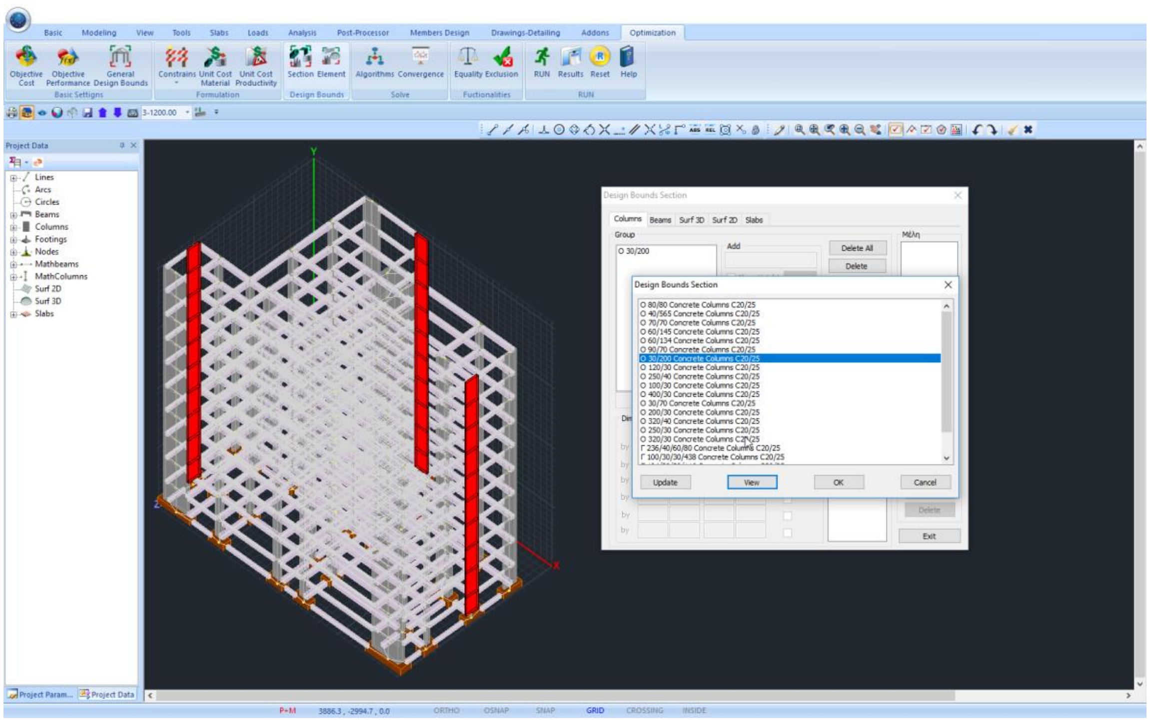
1) In the first tab "**Columns**", the way of inserting the cross-sections can be done in two ways: With selection from List and or with graphically.
The graphical selection is done with the "Pick-Select" button.
Clicking the "List" button, the following dialog box appears



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

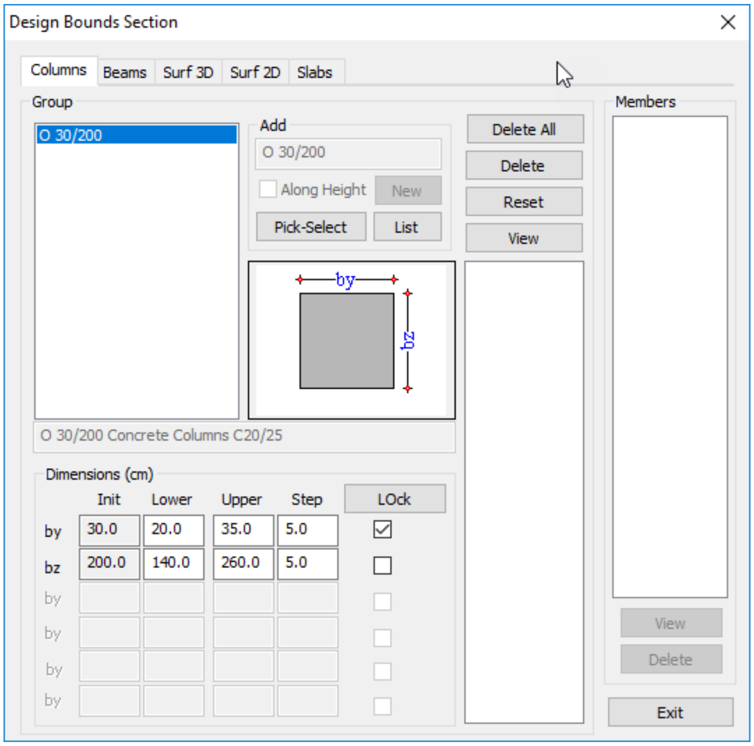
- Type of the cross-section
- Layer
- Material (Type & Class)

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.



- For this example, we select the 30X200 walls.

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.

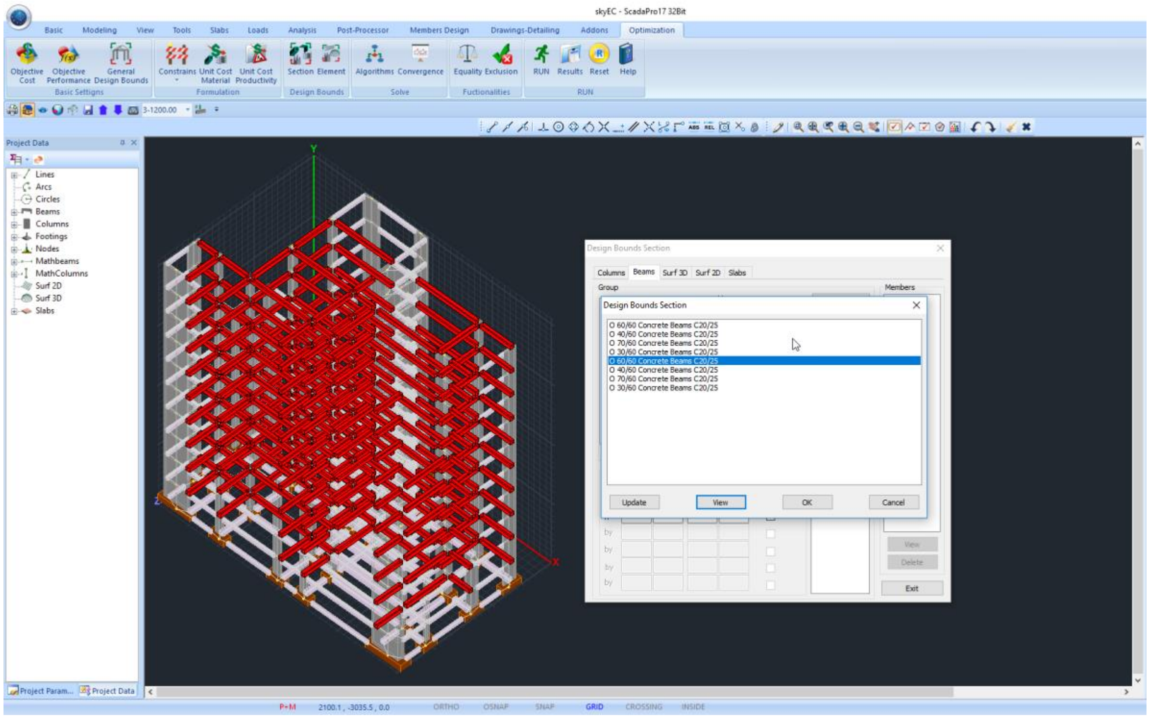
The "Lock" column locks this dimension to keep it unchanged at its original value. By clicking on the "Lock" button all dimensions are checked, i.e. the entire cross-section geometry is locked. The default values of the section bounds in columns “Minimum”, “Maximum” and “Step” are those defined in the “General Design Parameters” that initially obey all the elements.

- For this example, we lock the 30cm dimension.

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.

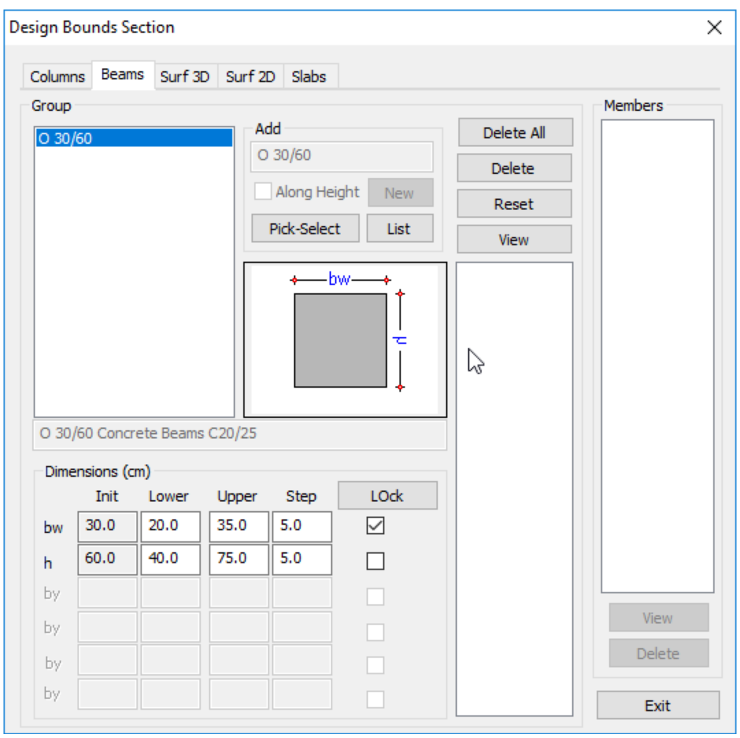
Finally, when selecting the "Show" button, the elements that have the cross-section will be displayed in red color in the 3D view of the structure.

2) The section “**Beams**”



Is exactly the same procedure as for the columns.

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



3) The next two sections are related to cross-section of of 2D and 3D finite surface elements **"Surf 3D"** and **"Surf 2D"**. The definition of the optimization parameters is the same in both cases.

This example does not contain surface elements**,** **τ**hat's why is omitted. The procedure is the same in both cases and is explained in detail in the corresponding user manual.

4) The last section is about solid "**Slabs**".

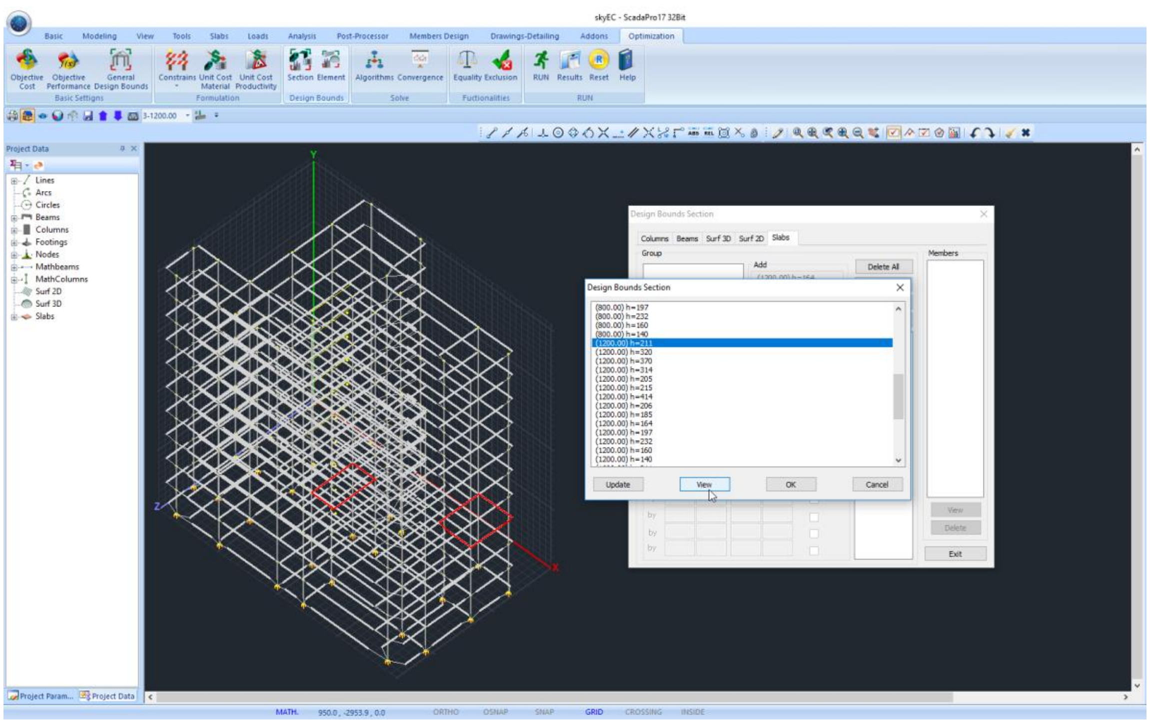
The selection here can also be applied in two ways:

The first way is by the display of a list

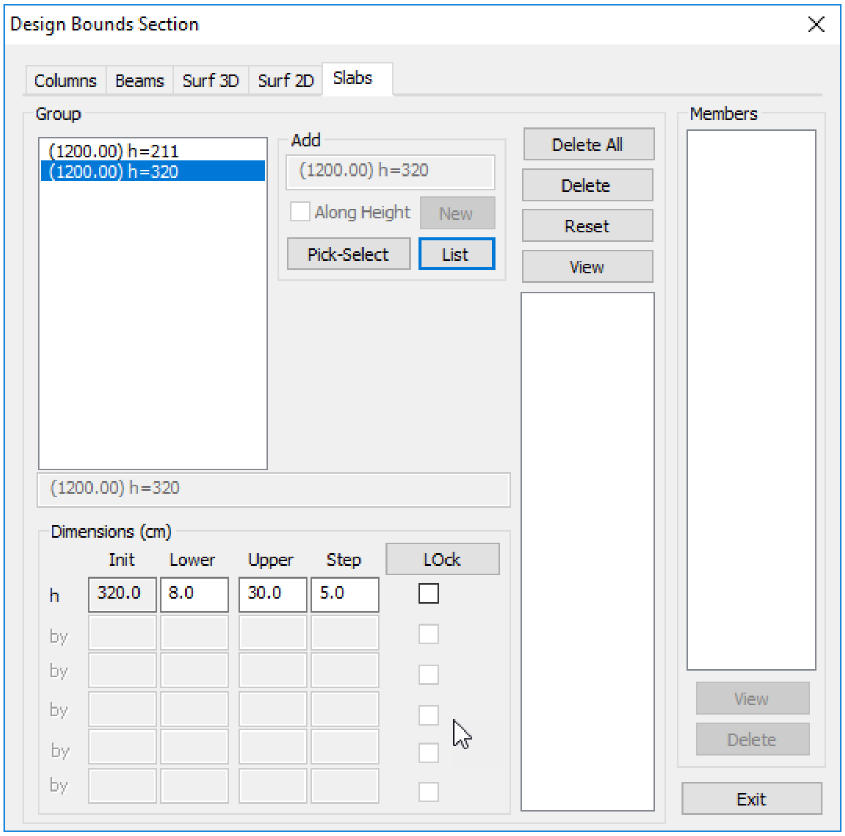
Where it includes all the slabs of the structure. The grouping of the cross-sections is based on two criteria:

- 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.

- For this example, we do not specify any additional limitations for the slabs.

Steel Sections

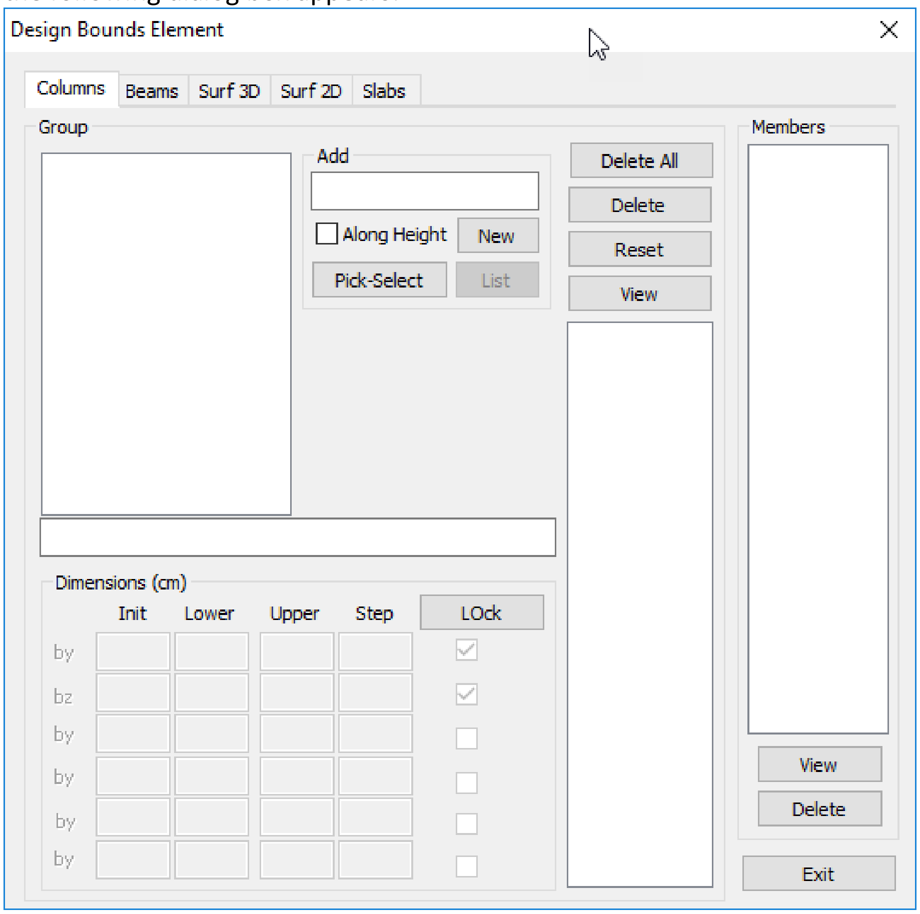
The grouping method for the structural steel frame sections is the same applied to the reinforced concrete the frame section of reinforced concrete.

- This example does not contain steel sections, τhat's why is omitted. The procedure is the same in both cases and is explained in detail in the corresponding user manual.

2.3.2 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:

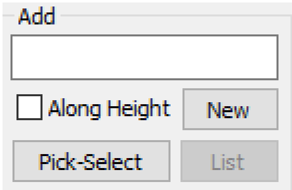


Where you are given the ability to create groups of elements for which you want to impose specific boundσ during their structural design or even to lock their dimensions. The the dialogue box is the same like the one for grouping per cross-sections, that is, divided into modules (Tabs) according to the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



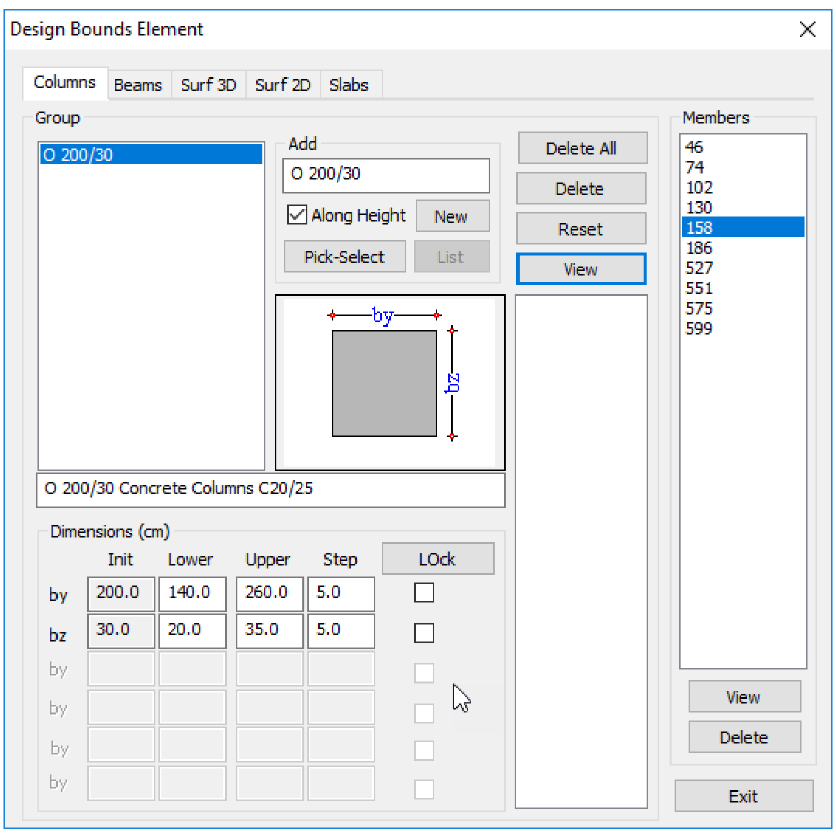
After writing the name in the field “Import”, press the “New” button to create the group and select "Pick-Select" to graphically select the members you want to join this group. The cross-section of the first column that you will show will also determine the type of the cross-section of the group you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

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.

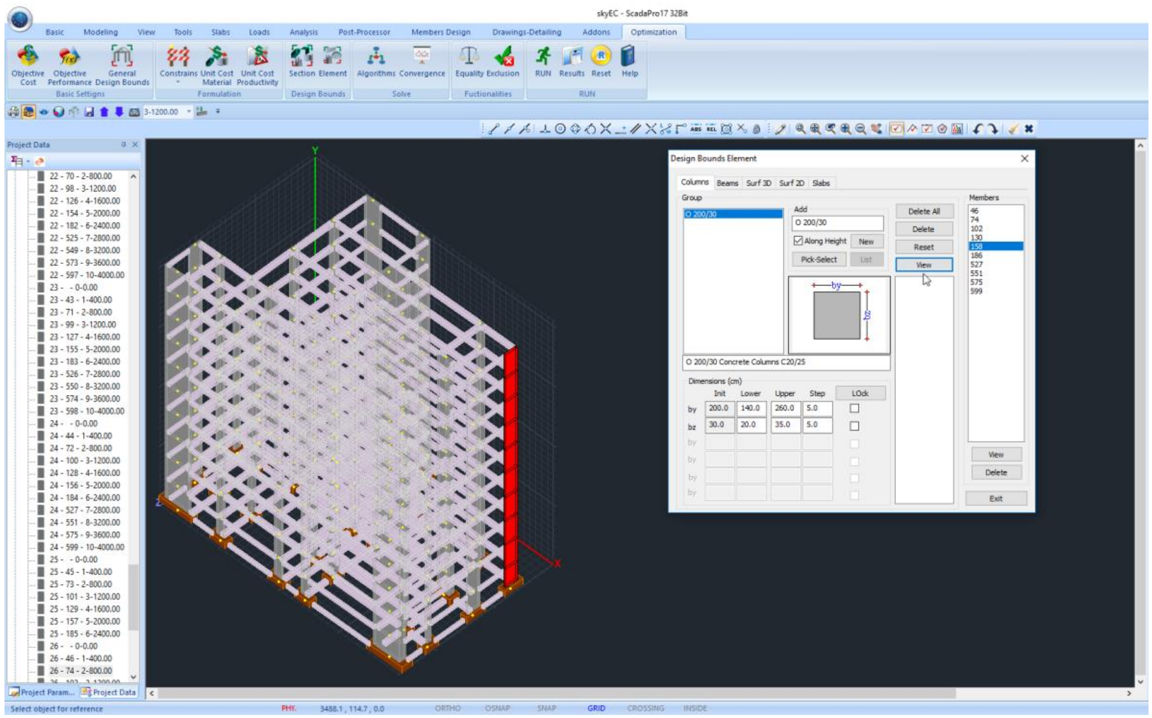
NOTE

⚠ If you do not complete a name for your group and just 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



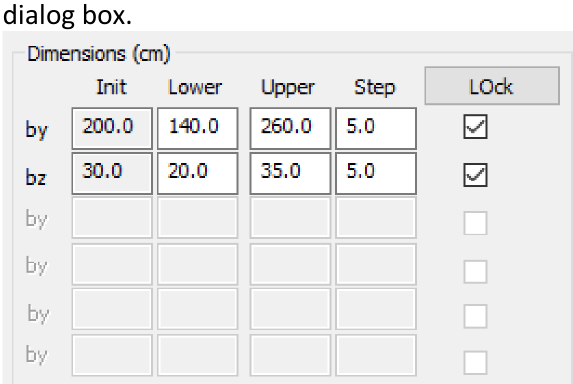
Here you can delete one or more items by selecting them (one at a time) and pressing the "Delete" button. You can also display them on the structural model with the "Show" button.



⚠ 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	200.0	140.0	260.0	5.0	☑
bz	30.0	20.0	35.0	5.0	☑
by					☐
by					☐
by					☐
by					☐

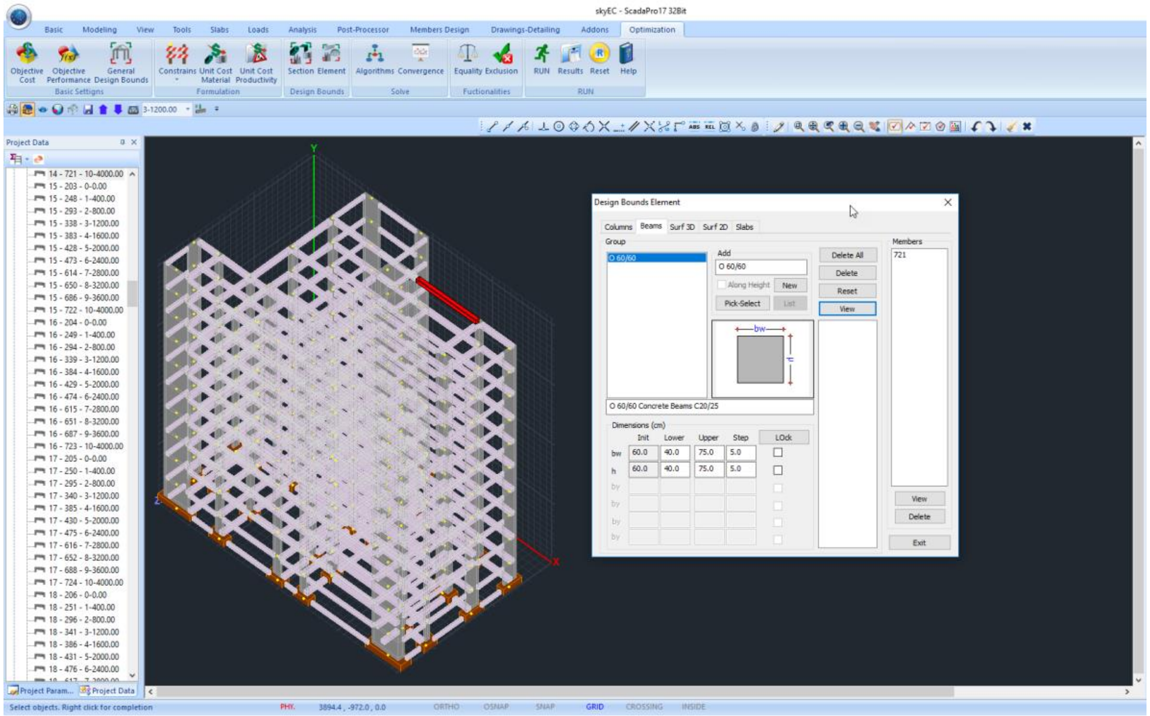
For this example, we choose to lock both dimensions of the whole group of 200X30 columns.

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.

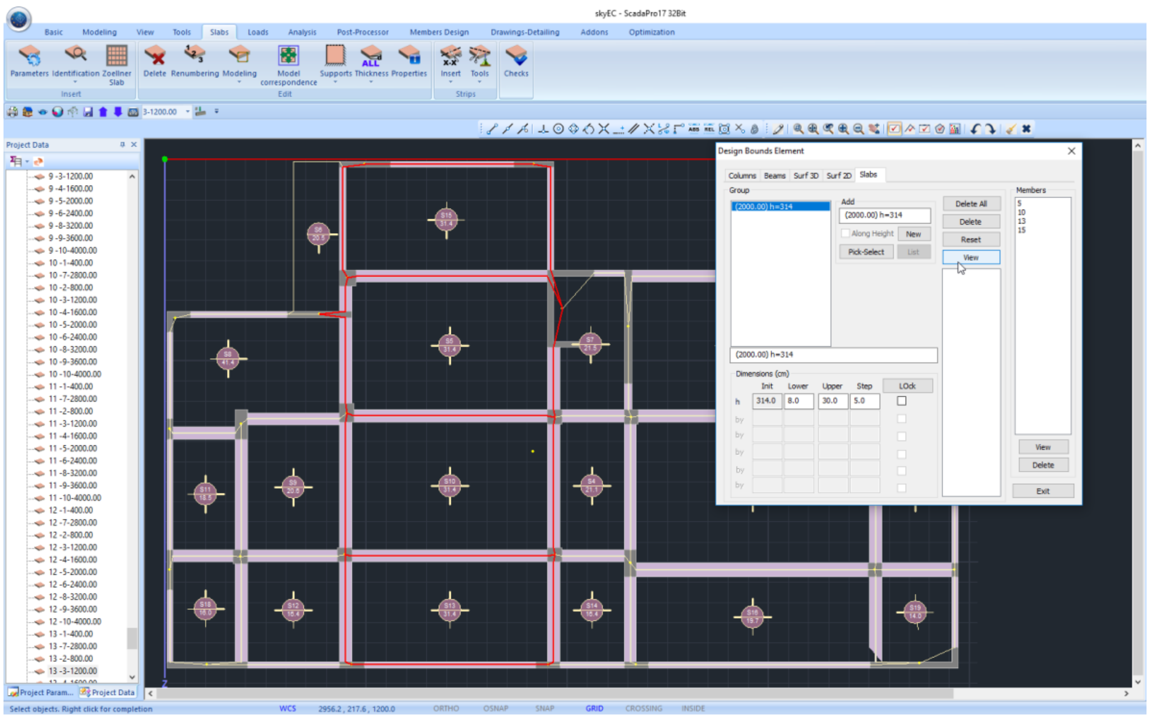
2) For the **Beams**, exactly the same ones previously mentioned for the columns apply.



For this example, elements design bounds are not defined in the beams.

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

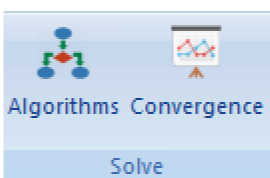
4) In the **slabs** section, you create a group where you put slabs of the same thickness that belong to the same floor.



- In this example, the above picture shows the definition of a group of slabs with 314 mm thickness belonging to the level 5 at an altitude of 2000.00 cm and includes four slabs 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.

2.4 Solution

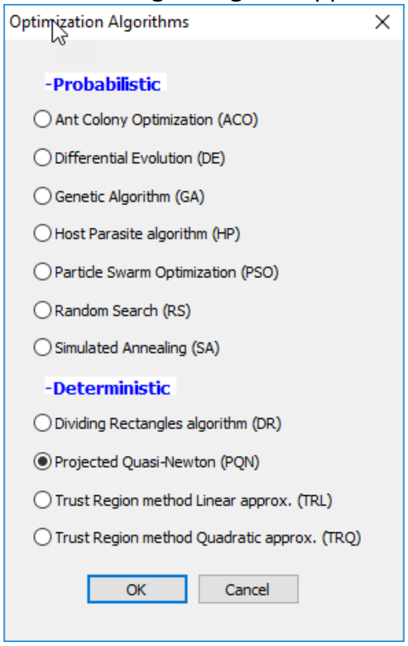


This section includes two commands:
“Algorithm”, where you select the algorithm to be used as the solution method for the optimization problem.
“Convergence”, where you specify the convergence criteria to be used as well as the analysis and design scenarios.

2.4.1 Algorithms

By selecting the command "Algorithms"

The following dialog box appears

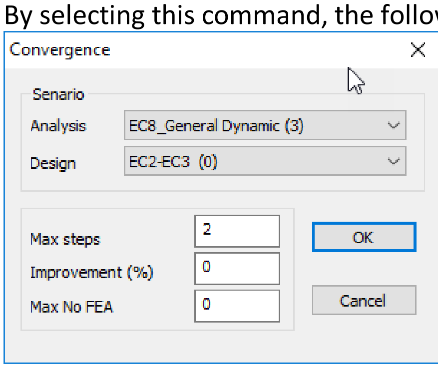


where you can choose the optimization algorithm from two categories

- Probabilistic
- Deterministic

2.4.2 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.

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.

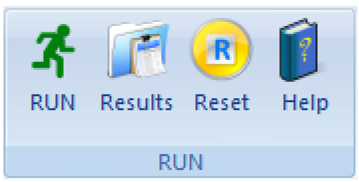
Max. Step Number: Set the maximum number of iterations

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

2.5 Functionalities

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

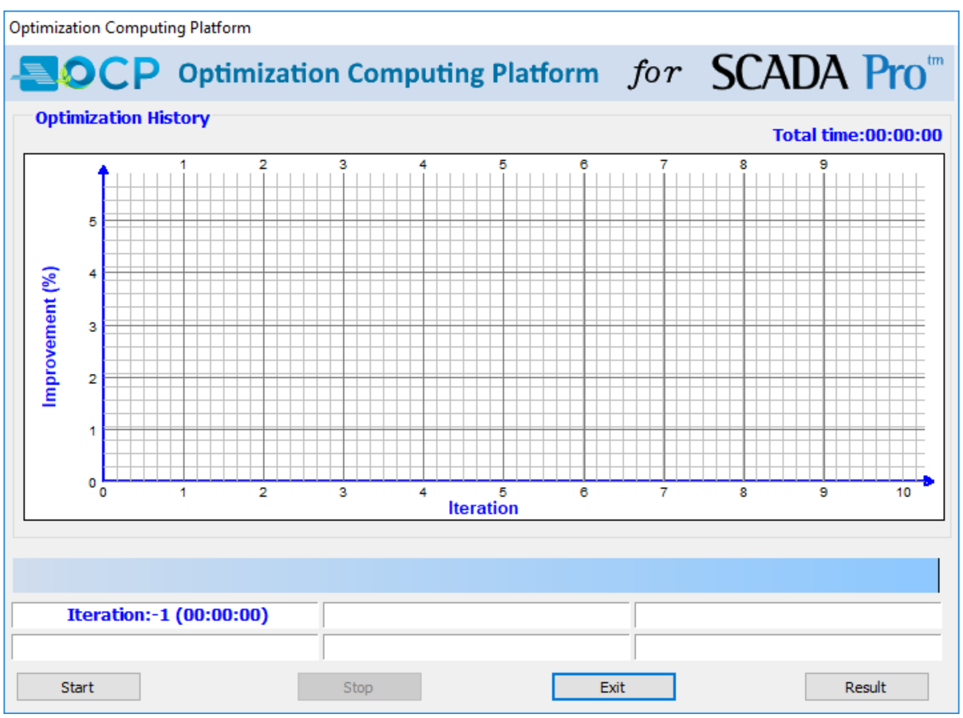
2.6 Run



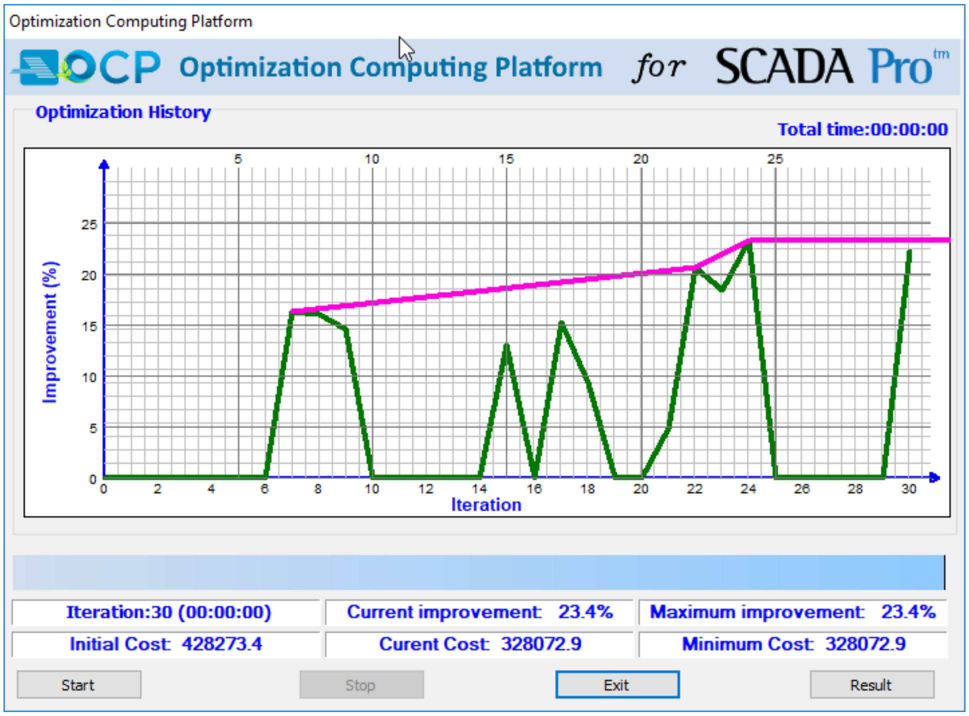
Η τελευταία ενότητα περιλαμβάνει εντολές που αφορούν στην εκτέλεση της διαδικασίας βελτιστοποίησης καθώς και στα αποτελέσματά της.

2.6.1 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 are:

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).

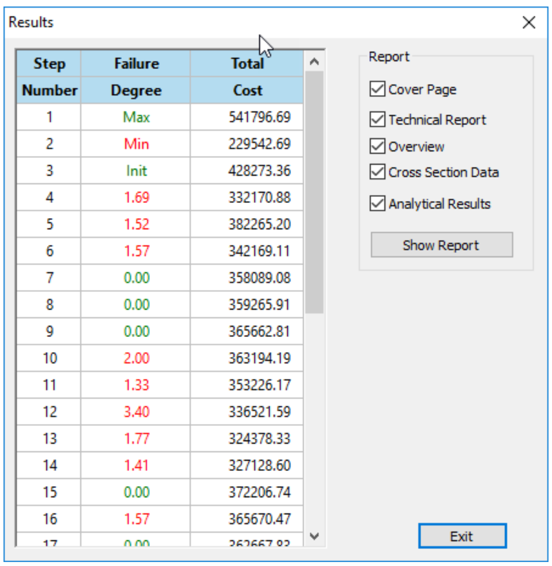
Curent Cost: 195828.2 The amount indicated here refers to the total construction cost of the structure in the current iteration.

Minimum Cost: 195828.2 The amount indicated here refers to the total construction cost of the best optimized structure achieved so far.

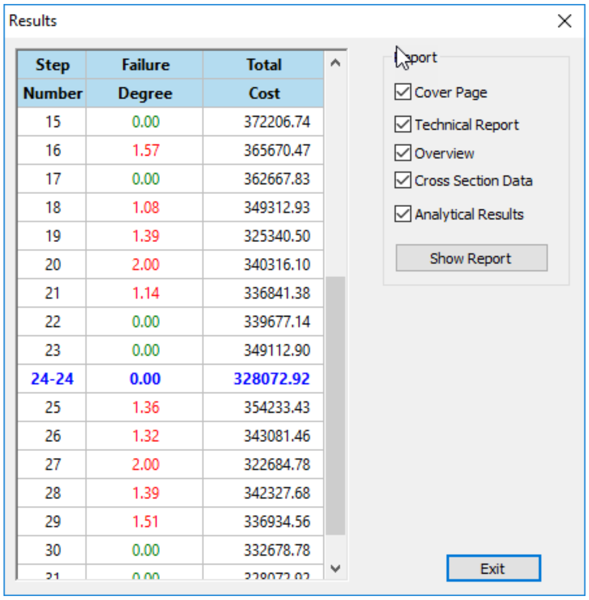
The optimization process ends when convergence is reached or ends if you press the key Stop .

2.6.2 Results

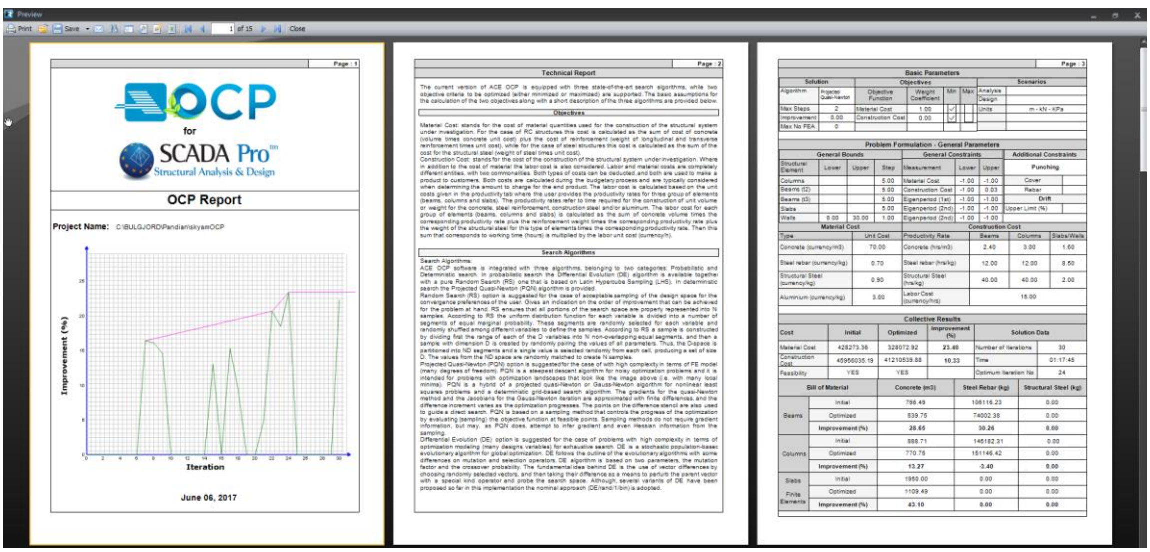
By using this command, the following dialog box appears



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 of each step. The step with the optimum solution obviously has a penalty grade of 0 and is indicated in blue color.



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 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 structure, the percentage of improvement achieved as well as the variations in the cross-sections of all structural elements.

2.6.3 Reset

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