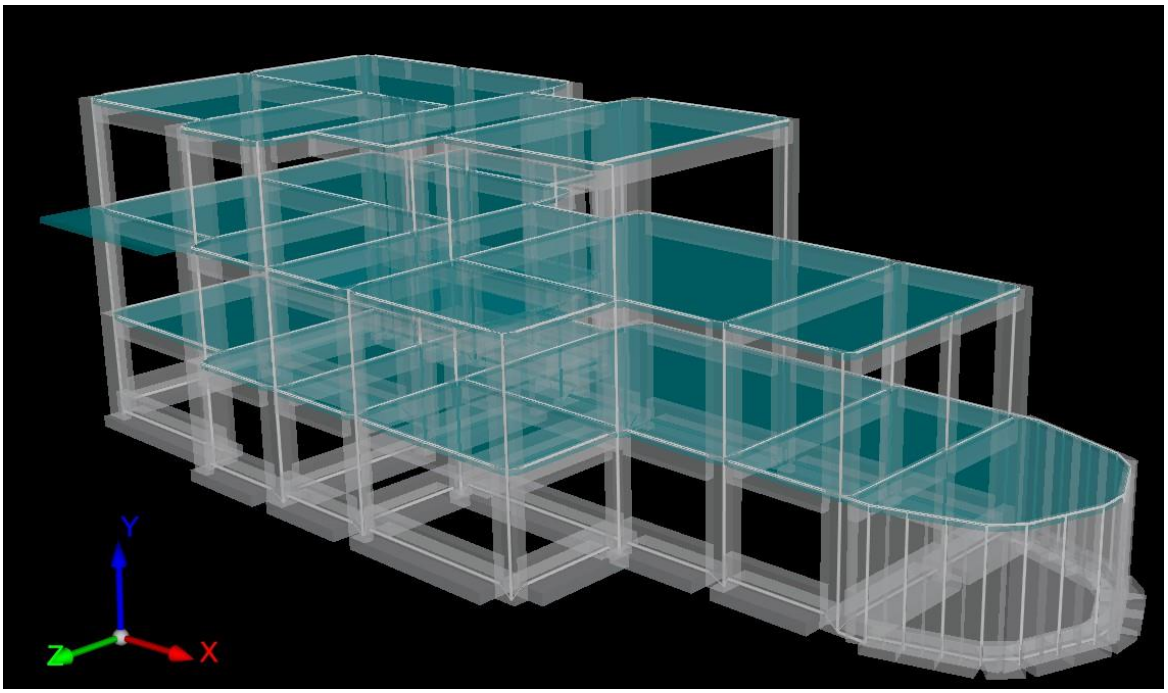
SCADA Pro 25™

Structural Analysis & Design

User Manual

4. EDIT

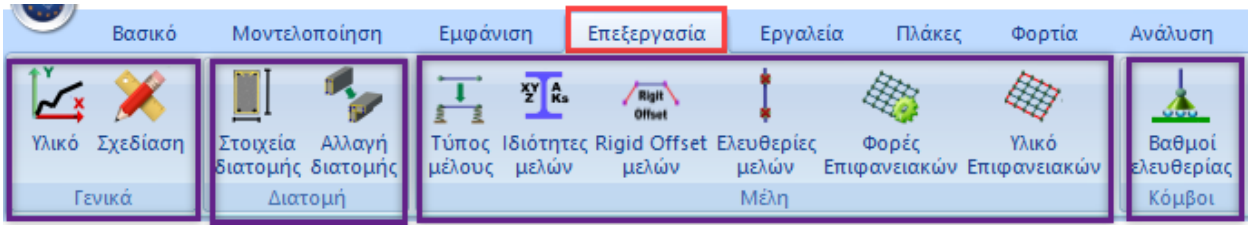


ACE HELLAS

INTEGRATED SOLUTIONS

CONTENTS

Chapter 4: Processing



The 4th Module is called "PROCESSING" and includes the following 4 groups of commands:

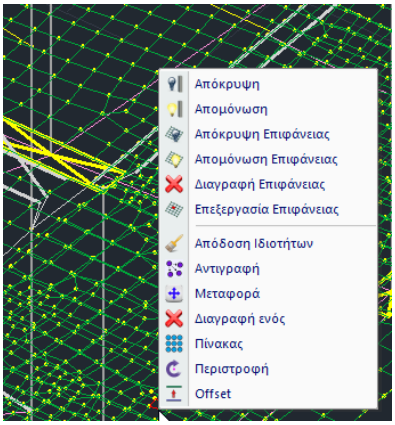
- √ General
- √ Cross section
- √ Members
- √ Nodes

OBSERVATION:

In the new version of SCADA Pro this new "TAB" of commands named "Edit" has been added and includes all the features of Multiple Options (*see User Manual 1. "Basic"*) and separate commands for greater convenience when editing an element or group of elements.

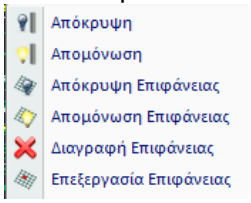
ATTENTION: *The commands with the word "**Cross-section**" refer only to the elements entered as physical cross-sections and allow changes to their characteristics. Conversely, the fields with the word "**Members**" refer to the elements defined as members with cross-sectional attributes as well as the mathematical members of the cross-sections, and allow changes to their attributes.*

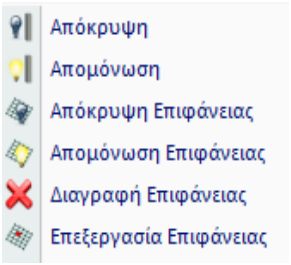
RIGHT CLICK:



And in the Edit section, right-clicking on the , specifically on an item, brings up a list of commands that allow you to directly edit it.

In particular, right-clicking on a surface element opens a list of surface operations:

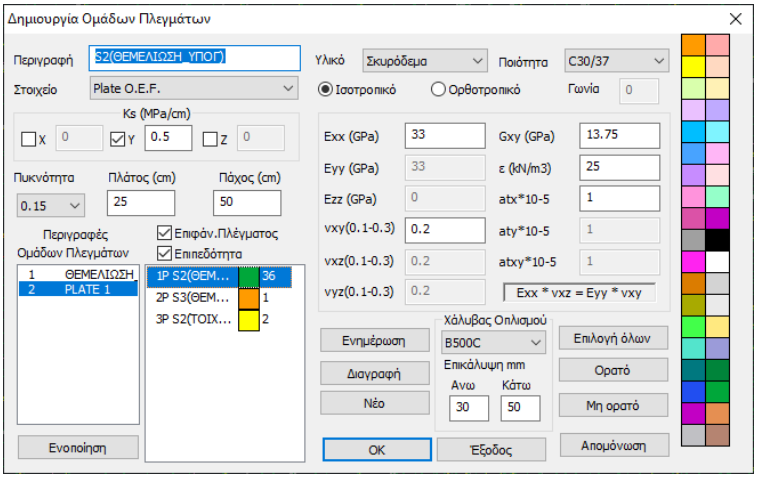




Hides the selected item Isolates the selected item

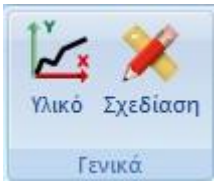
Hides all the surface to which the selected surface belongs Isolates all the surface to which the selected surface belongs Deletes all the surface to which the selected surface belongs

It takes us directly to the Create Grid Groups window with the specific sub-slice selected: **



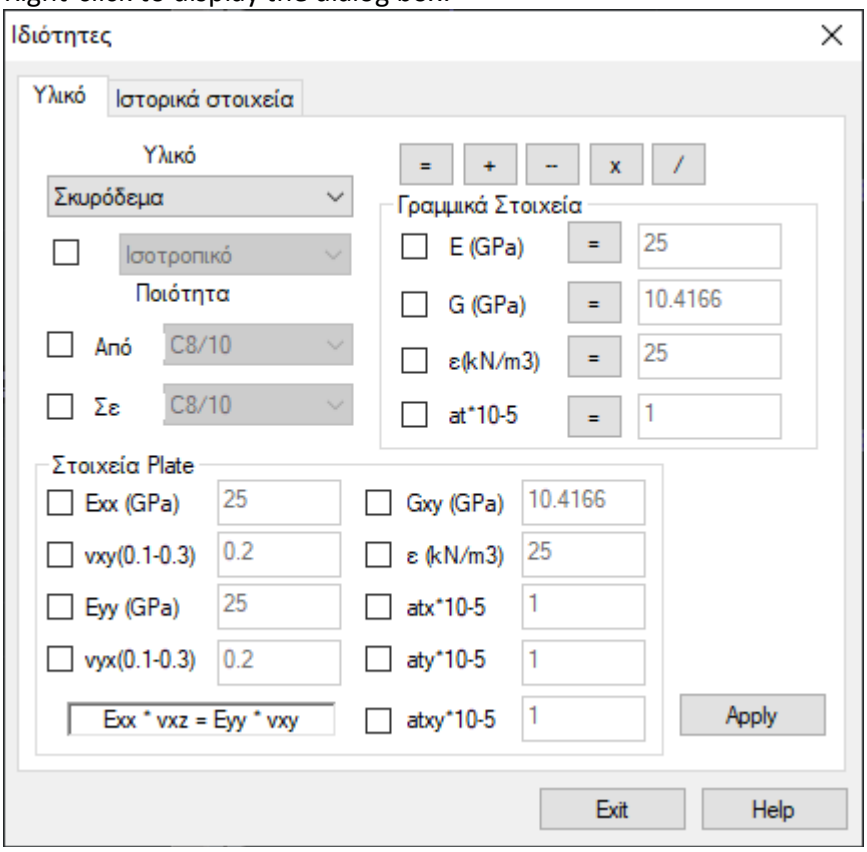
** A NEW easy way to locate a surface when there are many surfaces that fall on top of each other, making it difficult to locate them.

1. General



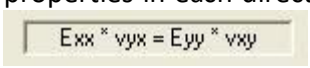
1.1 Material

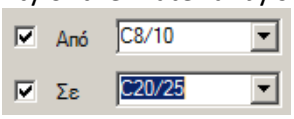
Select the command and the objects to modify by selecting one by one, or selecting with a polyline, or selecting with a window, or selecting with a polygon, or selecting a group  .
Right-click to display the dialog box:

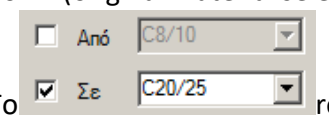


Where you can modify:

- the type of material, from the list of materials,
- the designation orthotropic or isotropic, (recall that a material is orthotropic when it has different properties in each direction. By selecting "Orthotropic", the parameters must satisfy the relation),

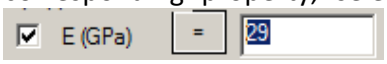


- the quality of the material by selecting "From" (original material selected) "To" (modified material) 

, or only "To 

regardless of the original.

- the physical properties of linear and surface members. To modify a value, activate the corresponding property, select the operation 

, enter the new value. 

NOTE: In subsequent modification of the material, all its properties should be checked by default.

EXAMPLE

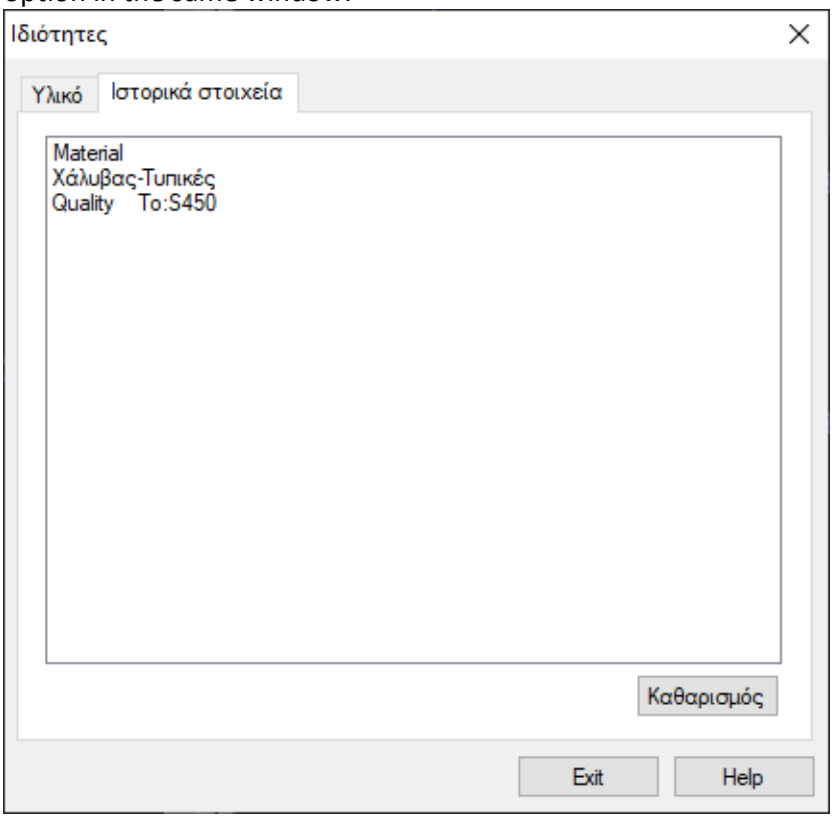
If, for example, you want to multiply the value of the elastic constant E, by a factor of 2, select 

and enter the value 2.

Select 

to save the modifications.

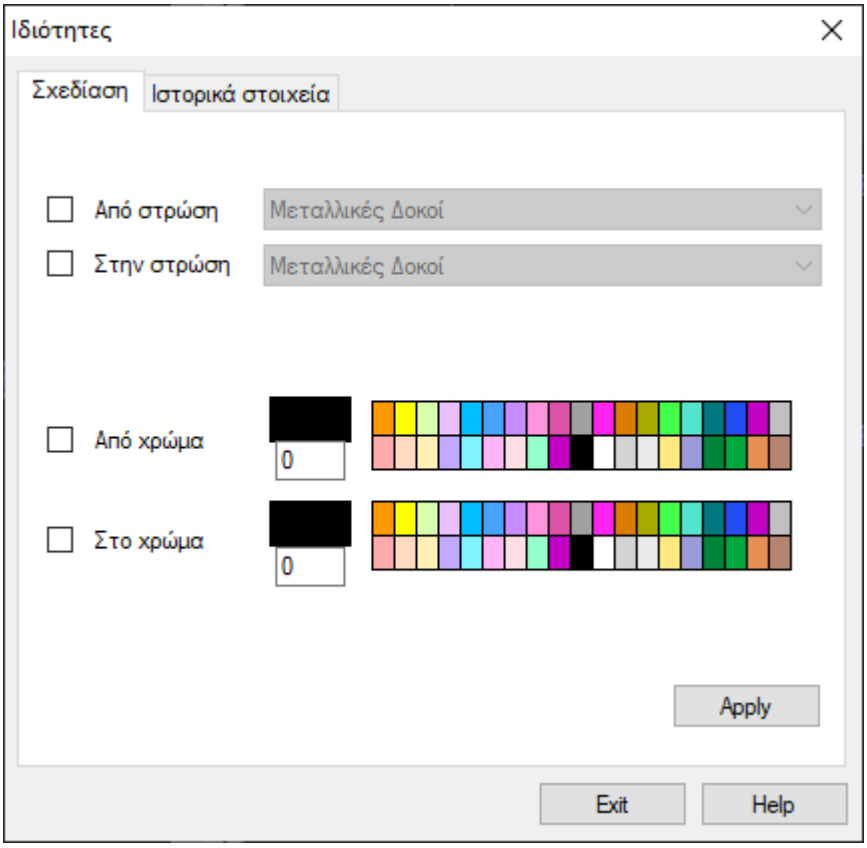
Every change you make to the Material updates the list of Historical Elements that is available as an option in the same window.



Selecting the Clear command clears the list of modifications.

1.2 Design

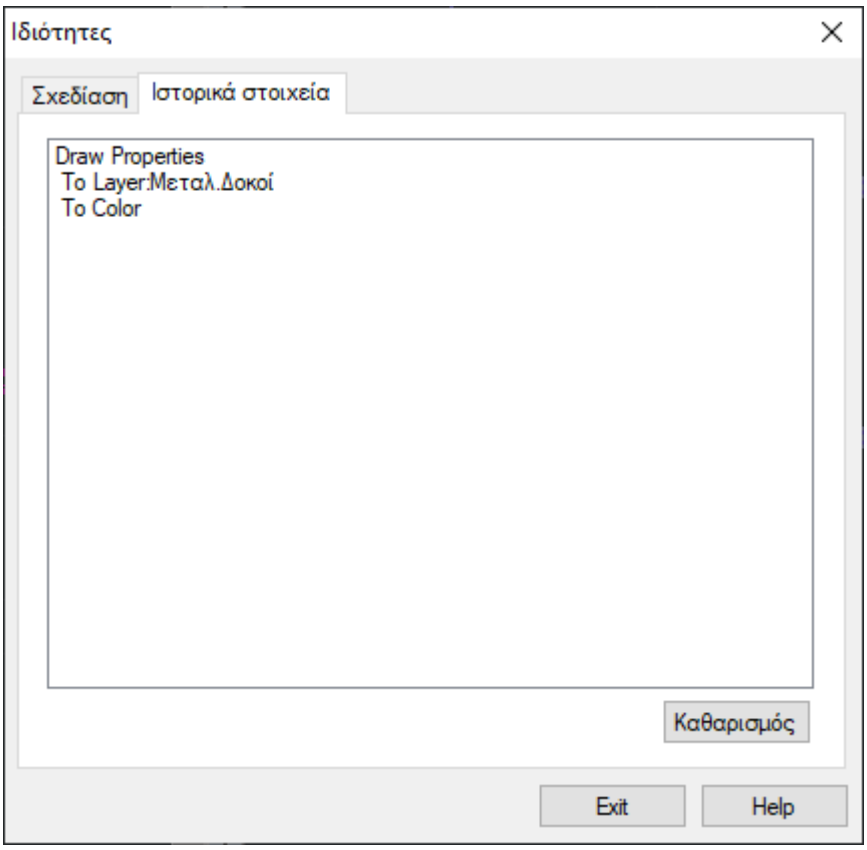
Where you can modify the overall layer & color of the selected elements.



Enabling "From layer" acts as a filter. It is optional and requires the selection of the layer to which the items to be layered belong.
Activating "Layered" changes the layer of the selected or "filtered" items.

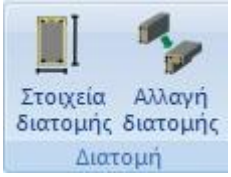
In exactly the same way you can modify the colour of the selected or "filtered" items.

Every change you make to the layer or color updates the list of Historical Elements that is available as an option in the same window.



Selecting the Clear command clears the list of modifications.

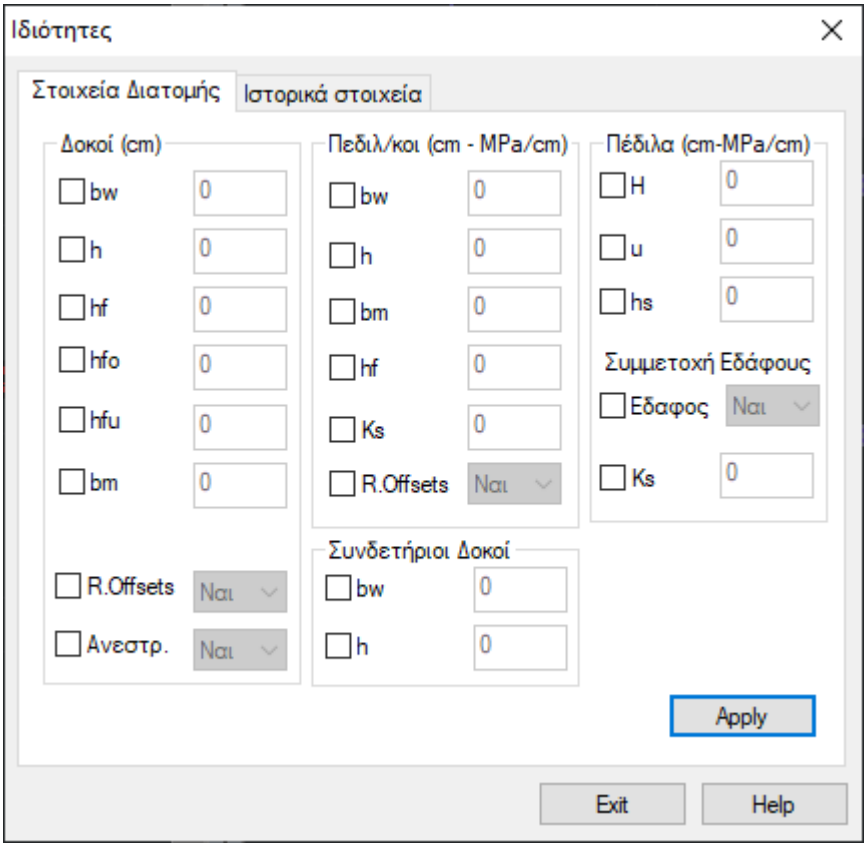
2. Cross section



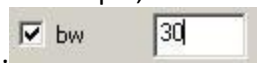
2.1 Cross-sectional data

Where you can modify the overall geometric characteristics (of the elements imported as physical cross-sections), namely:

- all the selected beams
- of all the selected pedestals
- of all the selected pedis
- of all selected connecting beams,



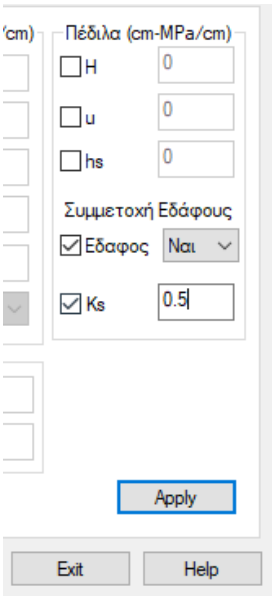
EXAMPLE 1

To change, for example, the dimensions of all selected beams, activate bw and enter the desired value, 30cm . 

You can also enable or disable rigid offsets, provided of course that the mathematical model of the beams exists.

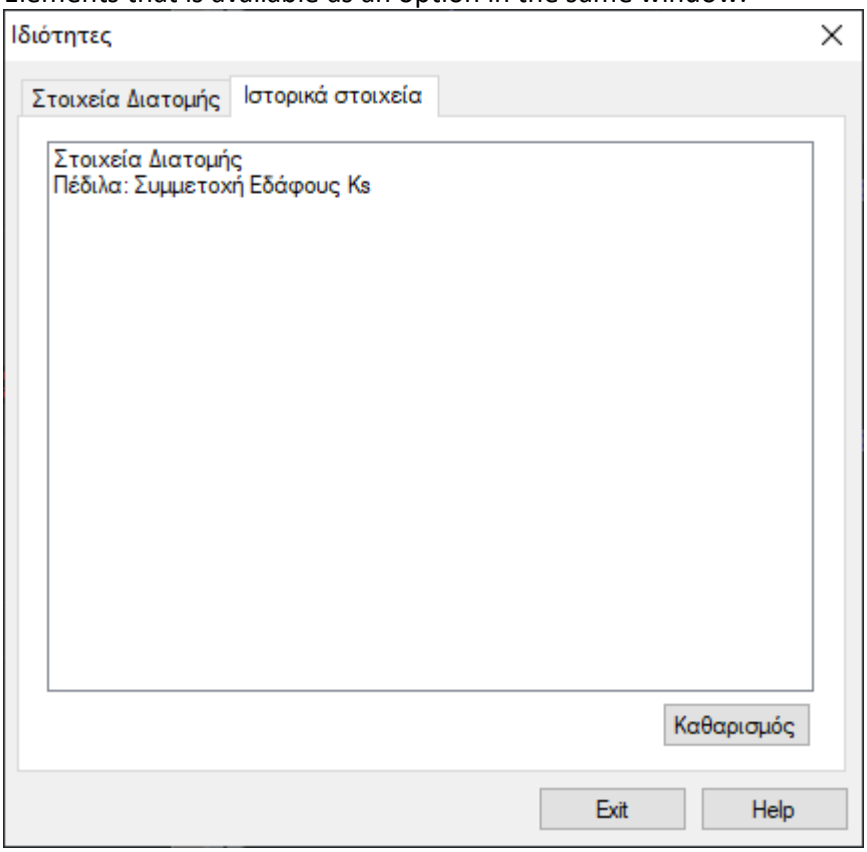
EXAMPLE 2

You can window select the entire 0 level that includes skids and set the Ground and spring's Ks involvement.



Apply to implement the changes. Exit to close the window.

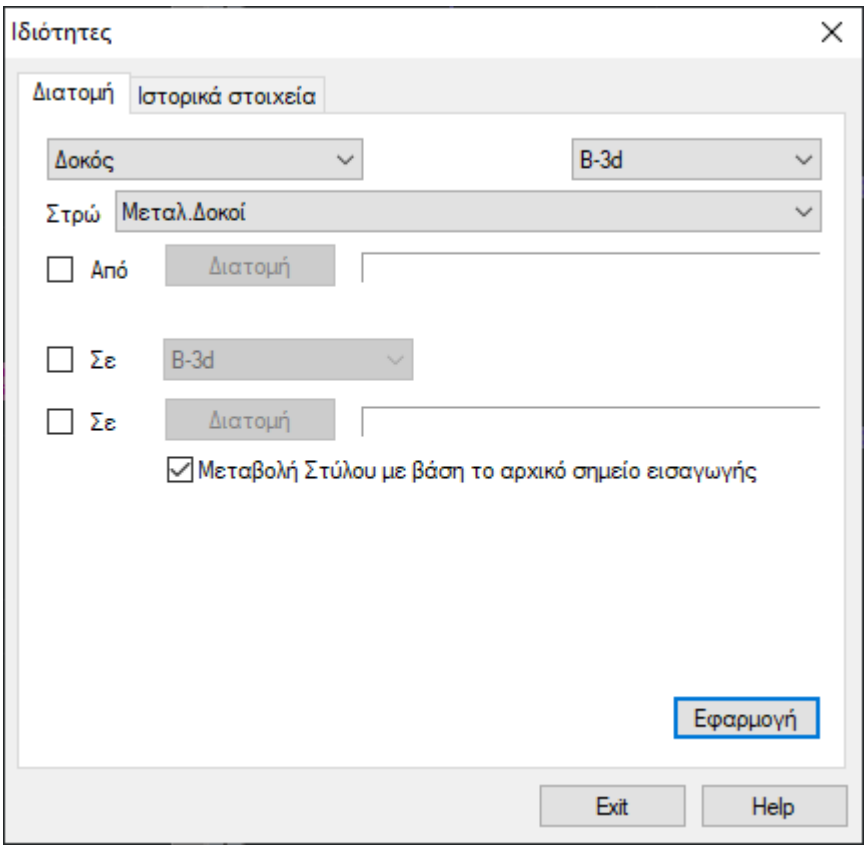
Every change you make to the geometric characteristics of the elements updates the list Historical Elements that is available as an option in the same window.

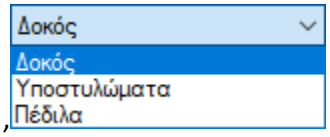


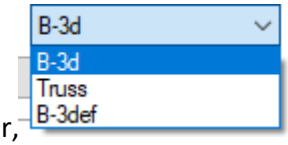
Selecting the Clear command clears the list of modifications.

2.2 Change of cross-section

Where you can modify a cross-section when it has been entered as a physical cross-section and change:

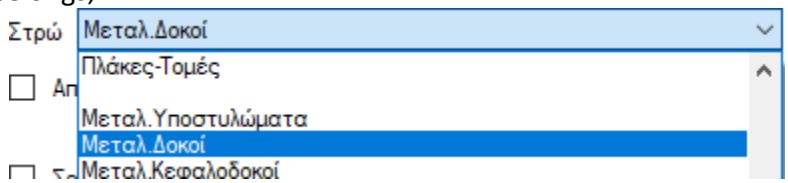


The building category, 

The type of linear member, 

,

The layer to which the element belongs,



and finally,

The filters 

as in the example:

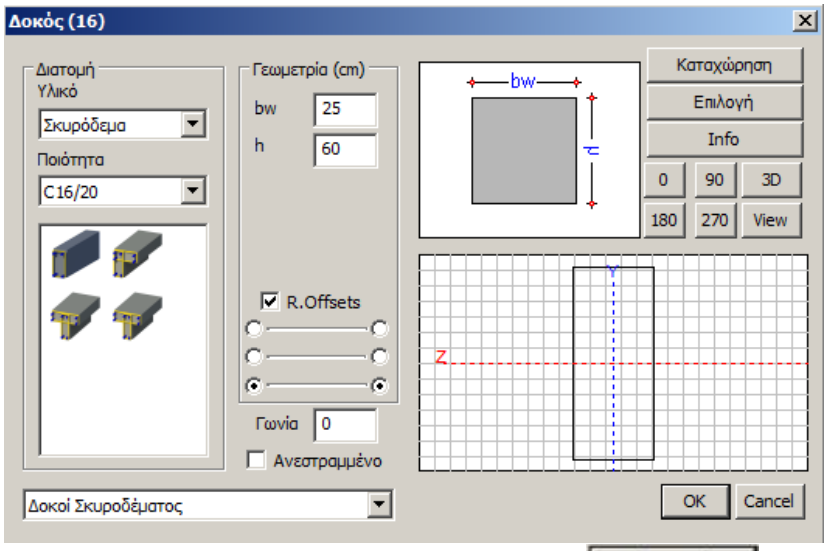
EXAMPLE

Modify, at one level, the dimensions of the cross-sections of the beams of a plane from 25/60 to 35/70.
For this modification the existence or not of the mathematical model is irrelevant.

Select the command "Change Cross-section" and by activating the selection option with window  select the whole floor plan. Right click and the dialog box appears. The beams selected with the window may not all be 25/60, or you may have included columns and other elements, so a filter is required to filter the selection leaving only the 25/60 beams.

Select "Beam" and "B-3d", (if the mathematical model has not yet been created, then the choice of the type of linear member is irrelevant).
Select the layer of the beams, that it is "Concrete Beams" and finally, activate the checkbox "From" and the command . 

The beams dialogue box appears on the screen, where you enter the dimensions of the beams to be modified (25/60).



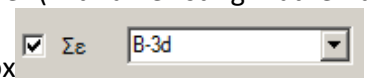
Activate the "In" checkbox and the command 

and enter the new dimensions (35/70).

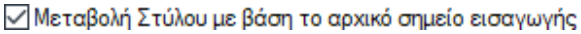
In case you want to modify the dimensions of all the beams independently, the procedure is the same, but without activating the "From" 

.

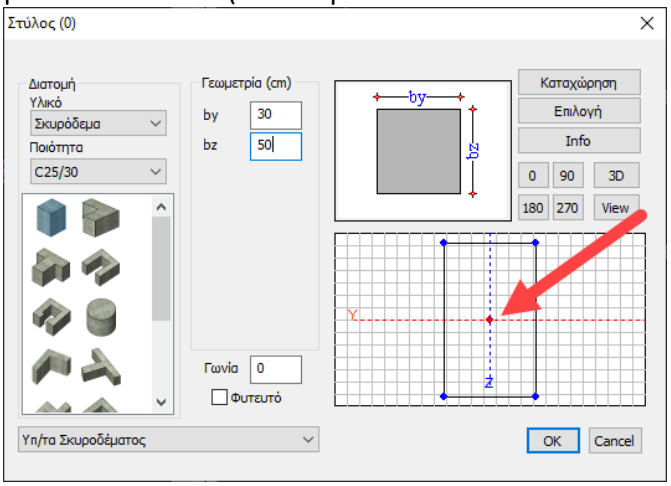
It is also possible to modify only the type of linear member (with an existing mathematical model).

Select the objects and the command, activate the checkbox 

and from the list select the new type.

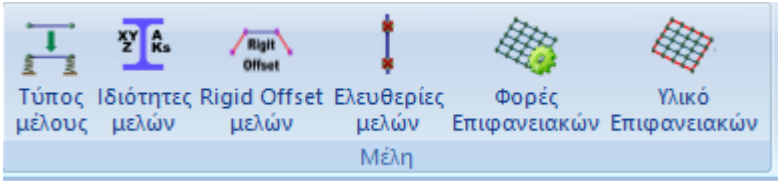
The choice 

, concerns the modification of the cross-sections of the Pillars and its activation means that any modification will take place with the initial point of insertion (i.e. the point shown in red in the Pillars window) as a fixed point.



Every change you make to the cross-sections of the elements updates the list of Historical Elements that is available as an option in the same window.
Selecting the Clear command clears the list of modifications.

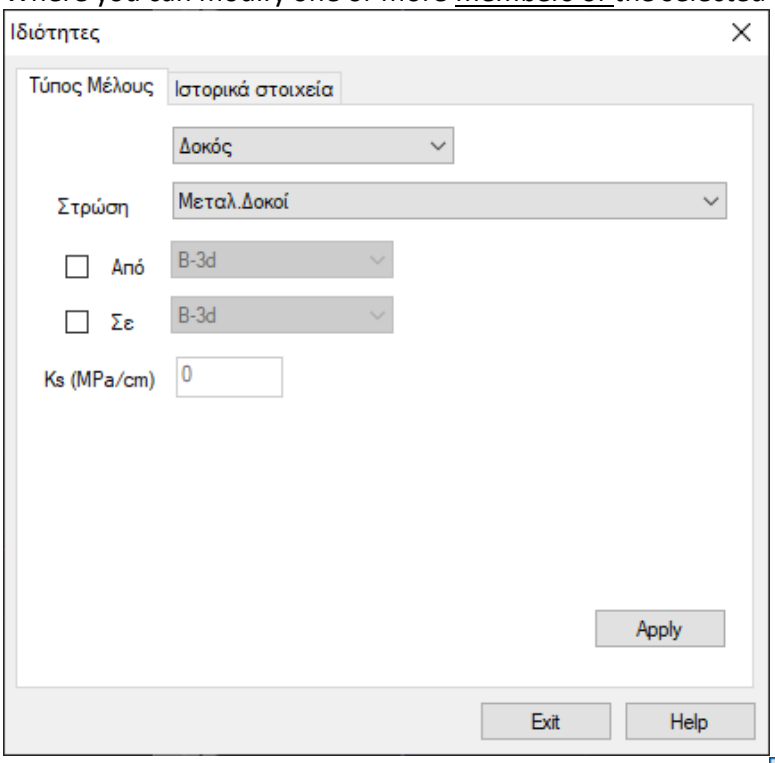
3. Members

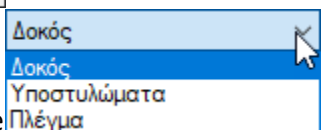


The "Members" command group contains commands to modify the members and surface times.

3.1 Type of Member

Where you can modify one or more members of the selected items:





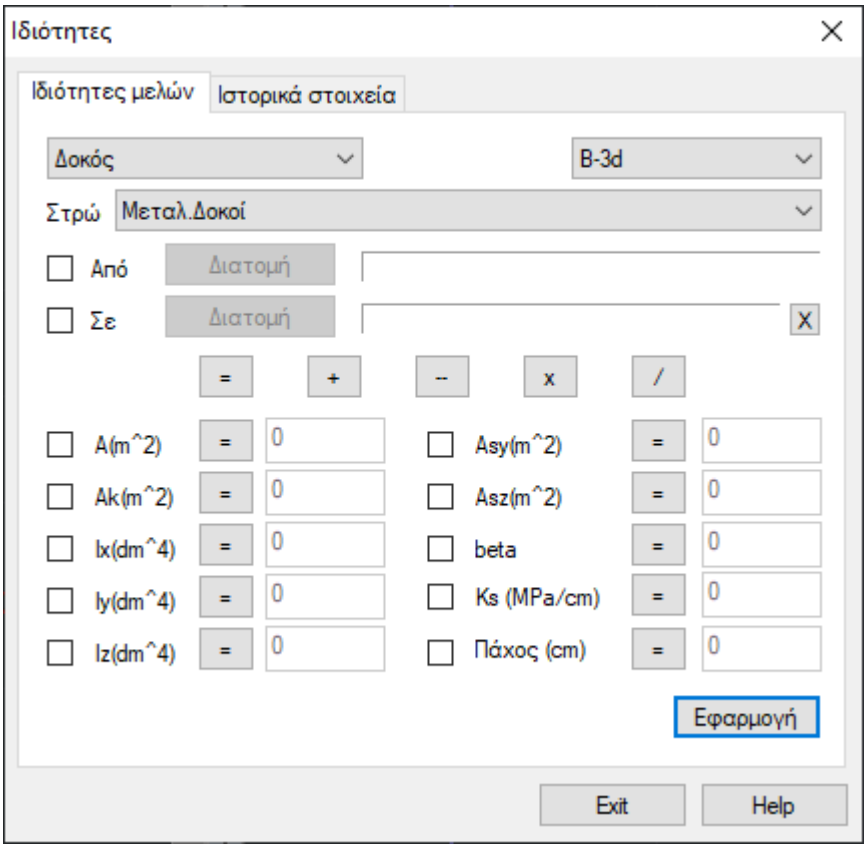
Select the category of building elements and change their type . Use "Layer" and/or "From" to filter the selected elements, or select "To" directly to change the type by selecting from the list.

For the foundation data "on Elastic Foundation (ef)" activate the Ks field where you enter the value in Mpa/cm.

Any change you make to the membership type updates the list of Historical Data that is available as an option in the same window.
Selecting the Clear command clears the list of modifications.

3.2 Members' properties

Where you can modify a cross-section when it has been entered as a mathematical member with a cross-sectional attribute (such as in standard constructions) and change:

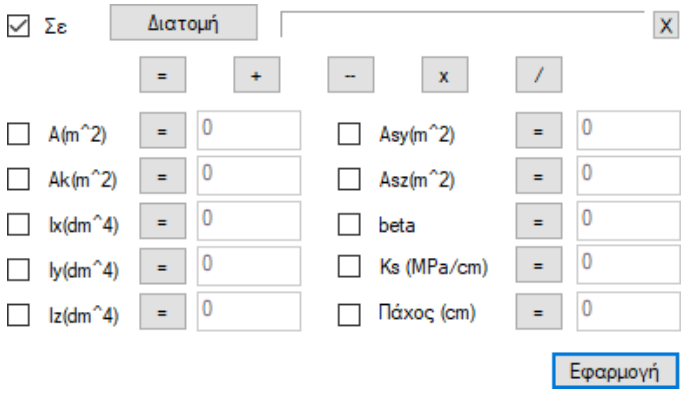


the physical properties of all selected elements of the same type or only those of a specific cross-section by activating the 

filter.

Select a new cross-section 

and the program automatically fills in the values, which you can modify or fill in yourself,



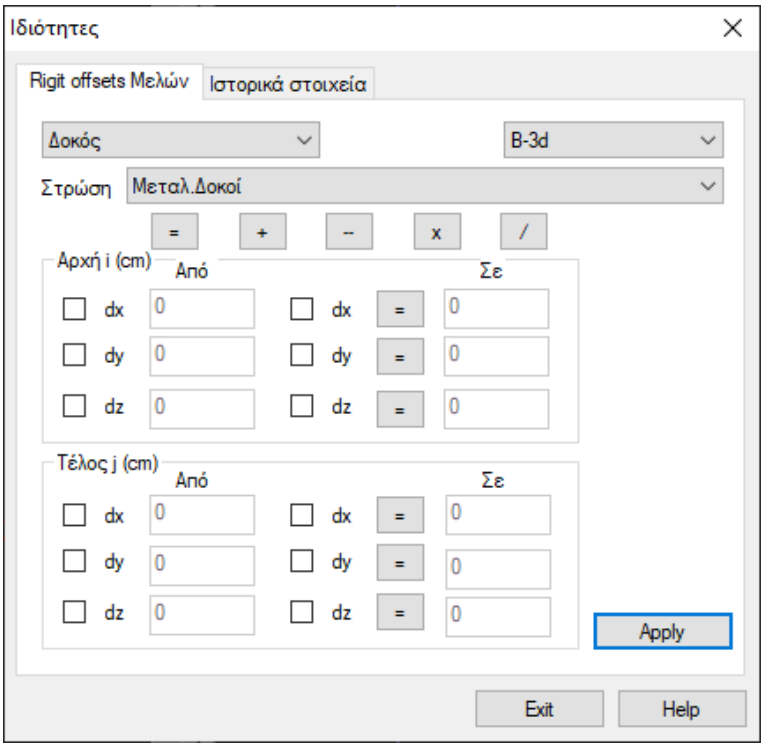
by activating the corresponding checkboxes, selecting an operation (=) and typing the new value. If, for example, you want to multiply the area by the factor 2, select the operation x and type 2.

Select Εφαρμογή to save the changes.

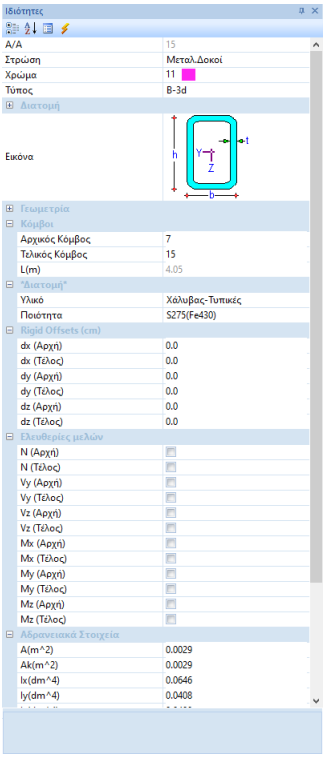
Every change you make to the membership properties updates the list of Historical Data that is available as an option in the same window.
Selecting the Clear command clears the list of modifications

3.3 Rigid Members' offsets

Where you can globally modify the Rigid offsets of the mathematical model members of the selected elements.



NOTE: Recall that this command allows global modifications, while for individual modifications it is better to select the member and edit it directly from the "Properties" field.



Activating "From" acts as a filter. It is optional and requires entering the values of dx, dy and/or dz of the rigid offset to be modified.

To set a new value, activate the corresponding checkbox 

, select the action 

and enter the new value.

EXAMPLE:

Goal: To zero all rigid offsets by x (dx) of the beginning of the beams of a floor plan.

Select the command " Rigid offsets members ", activate the command  (option with window) and select the whole floor plan. Right click to display the dialog box. The selected group includes all the elements of the floor plan.

Select the beams from the list 

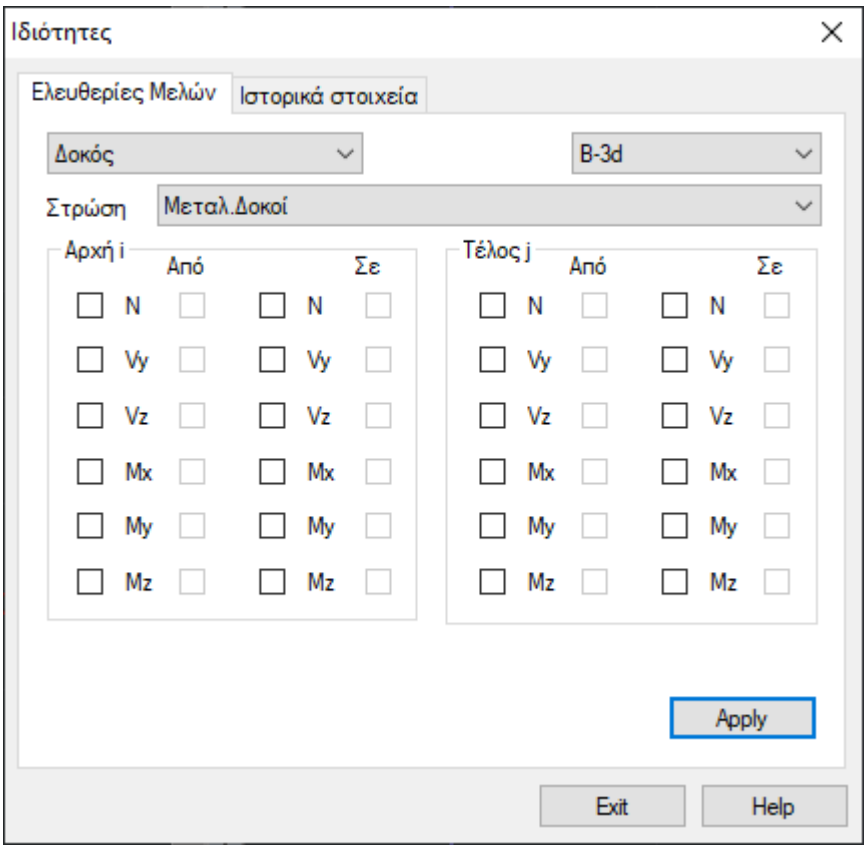
.
In the "In" field of "start" activate "dx" type the value 0 and select the =


"Apply" and "Exit".

Any change you make to the member's rigid offsets updates the list of Historical Data that exists as an option in the same window.
Selecting the Clear command clears the list of modifications.

3.4 Members' freedoms

Where you can globally modify the start and end freedoms of all members of the same type of the selected elements.



Select the intensive size(s) of the start to activate the "From" (which as always works as a filter and is optional) and the end to activate the "To" so that you can select:

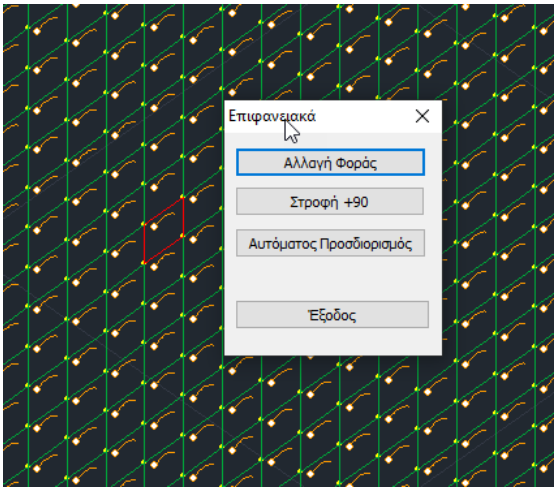
☐ = the intensive is transferred

☑ = the intensive care unit is not transferable

Every change you make to the member's freedoms updates the list of Historical Data that is available as an option in the same window.
Selecting the Clear command clears the list of modifications

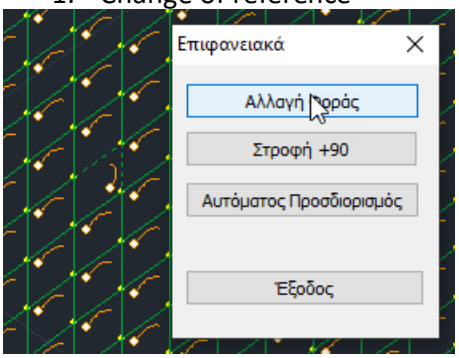
3.5 Loads of Surface

Where you can selectively or globally modify the times of the local axes of the selected surface elements.



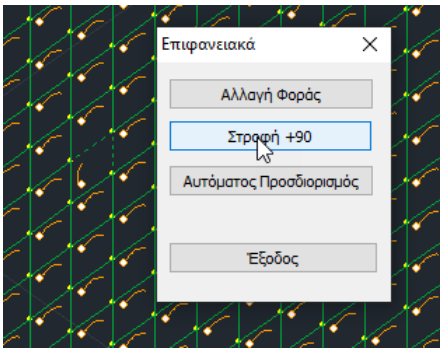
Select the command (local axes are automatically displayed) and one or more surface axes. Right click and the window opens allowing you to modify the times in 3 ways:

1. Change of reference



you select the command and it reverses the direction of the local axes from x to y and vice versa) and rotates the axes from clockwise to counterclockwise.

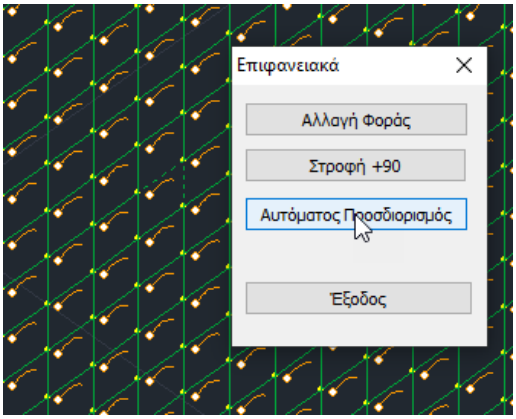
2. Turn +90



you select the command and change the axes 90 degrees with each

one click.

3. Automatic Identification

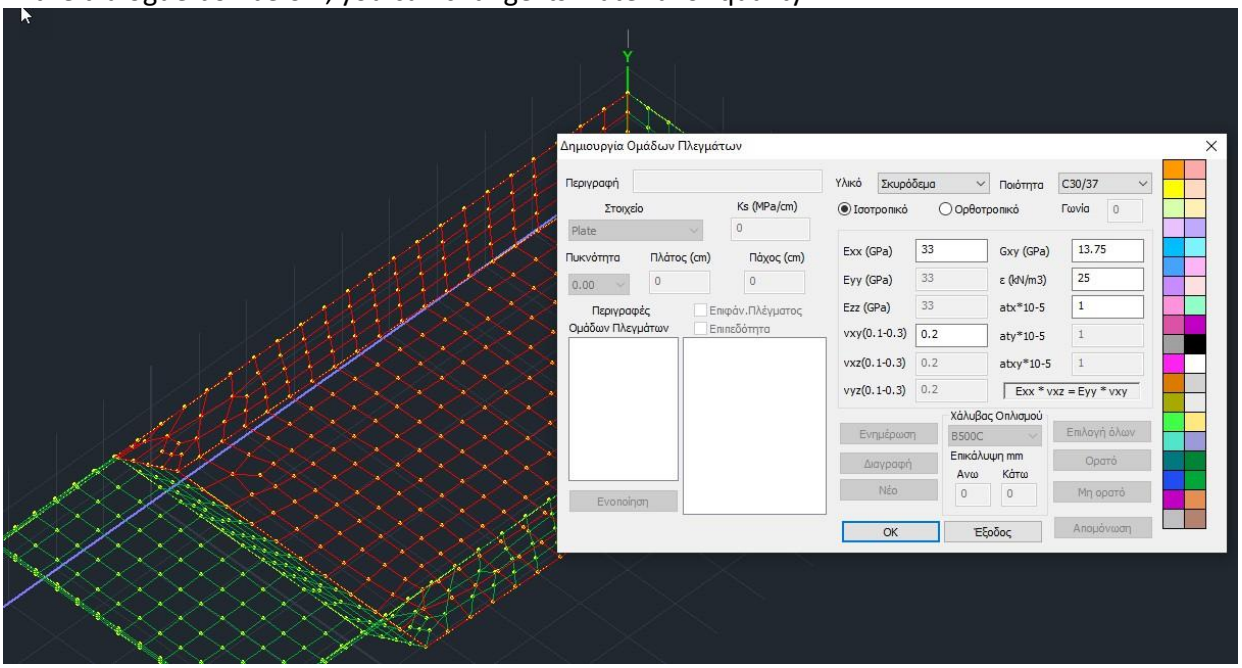


select the command and the program automatically determines the times of the elements, based on the geometry and topology of the surface.

3.6 Material Surface Material

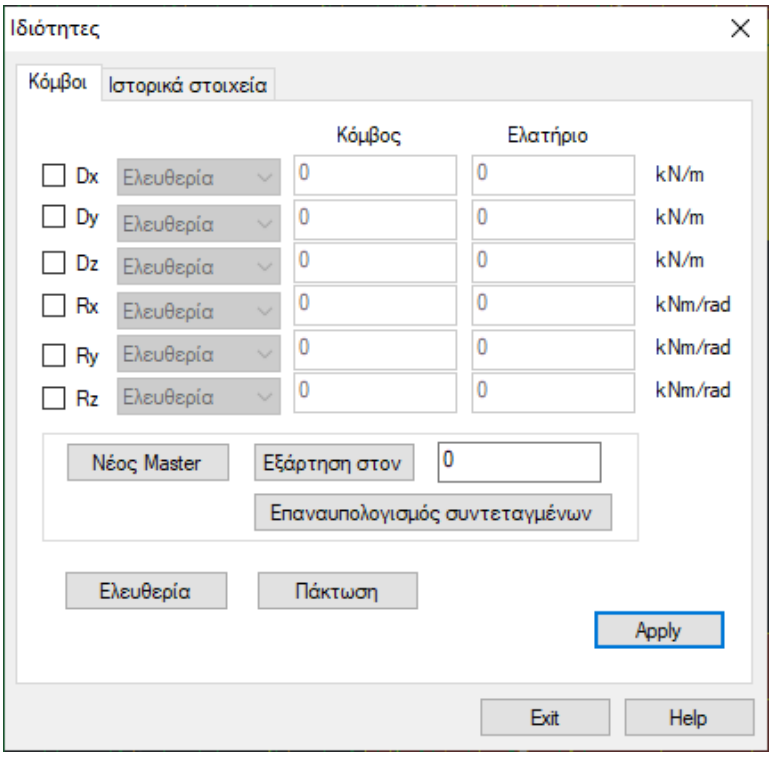
The Surface Material command is a new command for editing the material and quality of surface meshes.
By selecting it and then using the new way of selecting one or more surface sub-groups from one or more elements

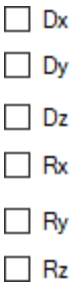
in the dialogue box below, you can change its material or quality.



4. Nodes

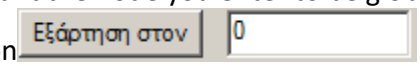
Command where you can selectively or globally modify the degrees of freedom of the selected nodes:





Activate the relative shift/rotation and select between:
"Freedom," "Concentration," "Dependence," "Spring."

- "**Freedom**": allows the node to move and rotate freely in the corresponding direction
- "**Packing**": binds the movements and rotations of the hub
- "**Dependency**": means that the specific movement or turn of the node depends on the corresponding node, whose number you specify in the "Node" column which is automatically activated when you select "Dependency". Here you have the possibility to make movements and turns dependent on more than one node.
- "**Spring**": automatically activates the "Spring" field where you set the spring constants for the desired displacements and rotations.

If you want the node you enter to be globally dependent on another node press the "Dependency onbutton 

and enter the node number.
The "New Master" option implies dependency of the selected nodes on the Master.

With the "Recalculate Coordinates" option, the program automatically calculates the new coordinates of the Master Node.

The "Freedom" and "Pack" options respectively free and pack all degrees of freedom.
Select "Apply" to enter the changes and "Exit".

Every change you make to the nodes updates the list of Historical Data that is available as an option in the same window.
Selecting the Clear command clears the list of modifications.