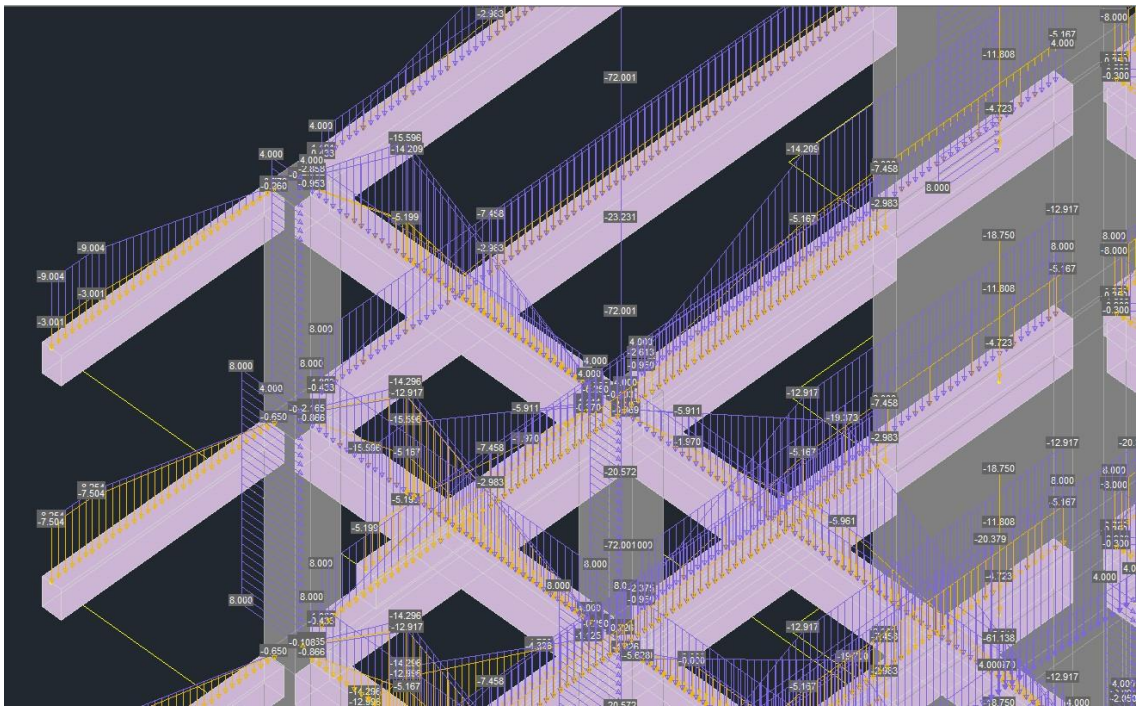
SCADA Pro 25tm

Structural Analysis & Design

User Manual

7.LOADS



ACE HELLAS

INTEGRATED SOLUTIONS

1. CONTENTS

1. CONTENTS 2

1. DEFINITION 4

1.1 CHARGE 4
1.2 CARGO TEAMS 6

2. LOAD OF PLANES 8

2.1 INTRODUCTION 8
2.1.1 Overall 8
2.1.2 SELECTIVELY 12
2.2 EDIT 13
2.3 LEAKAGE LINES 14
2.4 REACTION PERFORMANCE 15
2.4.1 TOTAL 15
2.4.2 SELECTIVELY 15
2.4.3 UNIFORMITY 15

3. MEMBERSHIP LOAD 16

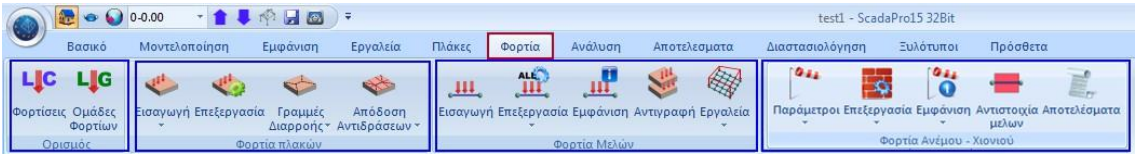
3.1 INTRODUCTION 16
3.1.1 MEMBER LOADS 17
3.1.1.1 Temperature Change Member 19
3.1.2 NODE LOADS 20
3.1.3 LOADS IN PLATE ELEMENTS 22
3.1.3.1 Temperature Change Plate 22
3.1.4 Loads in Solid elements 24
3.2 EDIT 28
3.2.1 Overall 28
3.2.2. OPTIONAL 29
3.3 SHOW 33
3.4 COPY 39
3.5 TOOLS 41
3.5.1 LOAD DISTRIBUTION ON SURFACE 41
3.5.1.1 FLAT SURFACES 43
3.5.1.2 SUCCESSIVE SURFACES 45
3.5.1.3 CURVED SURFACES 48
3.5.2 IMPORT OF LOADS FROM ARCHIVES 49
3.5.3 POOL LOADS 54
3.5.4 SCALE LOADS 56
3.5.5 LANDFILL OF THE LOAD FROM LANDFILLS 57

4. WIND AND SNOW LOADS 60

4.1 PARAMETERS 61
4.1.1 REGULATIONS 61
4.1.2 WIND PARAMETERS 61
4.1.3 SNOW PARAMETERS 63
4.2 EDIT 64
4.2.1 WALL TREATMENT 64
4.2.1.1 SEPARATE TO USE THE "STANDARD CONSTRUCTIONS" 64
4.2.1.2 USING THE "STANDARD CONSTRUCTIONS" 66

4.2.2 ROOF TREATMENT 66
4.2.2.1 SEPARATE TO USE THE "STANDARD CONSTRUCTIONS" 67
4.2.2.2 USING THE "STANDARD CONSTRUCTIONS" 68
4.3 SHOW 68
4.3.1 WIND SHOW 68
4.3.2 SNOW SHOW 69
4.4 MATCHING MEMBERS 69
4.4.1 MANUAL PROCESS - SEPARATE TO USE "STANDARD CONSTRUCTION" 71
4.4.2 SEMIAUTOMATIC PROCESS - SEPARATE TO USE THE "STANDARD CONSTRUCTIONS" 72
4.4.3 AUTOMATIC PROCESS - USING "STANDARD CONSTRUCTIONS" 74
4.5 RESULTS 75

Chapter 7: Loads



The 7th Module is called "FORTS" and includes the following 4 groups of commands:

- ✓ Definition
- ✓ Plate loads
- ✓ Member Loads
- ✓ Wind and Snow loads

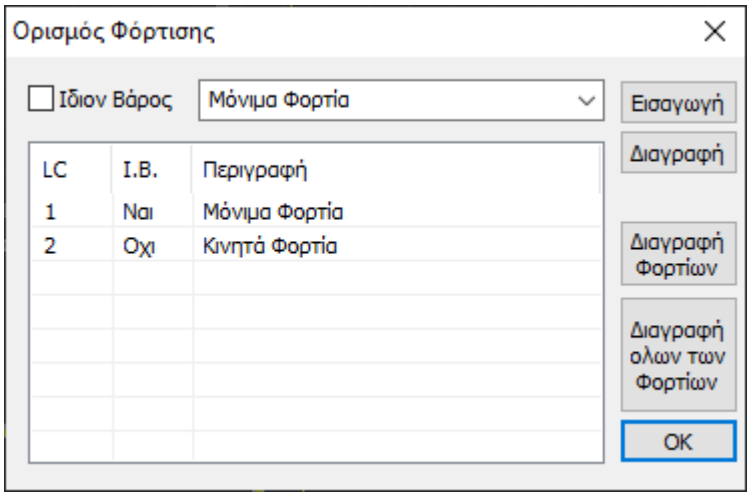
1. Definition

The commands of the "Definition" group allow the definition of the Loadings and the corresponding Groups, where all the loads of the carrier will be included.

⚠ *A basic requirement for the introduction of loads into a carrier is that the corresponding loadings have been defined. Each load will belong to one of them.*

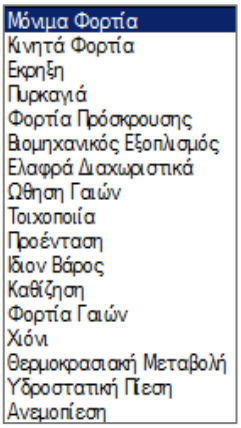
1.1 Downloads

The definition of loadings is done through the "Loadings" command. In the dialog box that opens:



There are, by default, some two loadings:

1. Permanent Loads (L.C.=1)
2. Mobile Loads (L.C.=2)



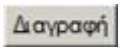
The I.B. column indicates the participation of the individual weight in the specific charge.
In addition to the permanent and mobile defaults, you can import other loadings by selecting from the list and then "Import".
In addition to the library loadings , you can type and enter your own loading.

- By pressing the "Insertbutton 

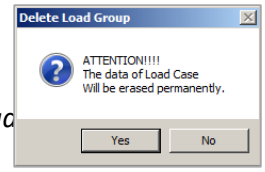
the load is given a serial number in the LC (Load Case) column and in the I.B. (Same Weight) column a "Yes" or a "'No" appears depending on whether or not the same weight is included in the load.
"OK" to save.

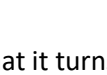
- When you want to include the same weight of the structure in a load (usually in Permanent), check the "Same weight" option. 

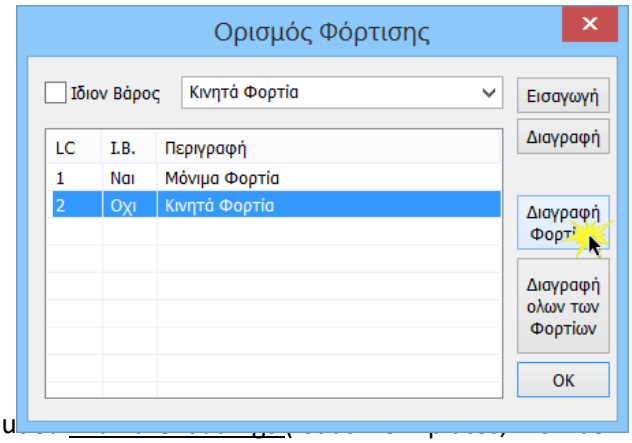
.

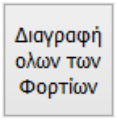
- To delete a charge you have already entered, first select it and then 

. The program will ask you to confirm your selection. If you select "Yes", it will delete it.
⚠ *A prerequisite for deleting a load is that it does not include any load*



- To delete the loads included in a load (plate loads, member loads, etc. that you have already entered), select the specific load from the list of loads (so that it turns blue) and then . 



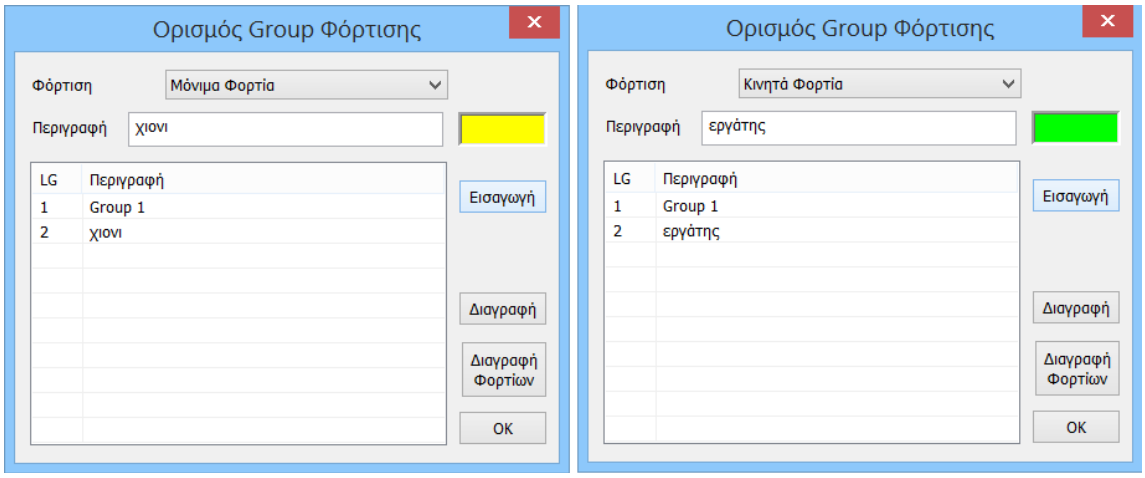
- To delete all the loads of the study inclu loads etc. that you have already entered), press the button 

1.2 Cargo Groups

Creating Single Load **Load Groups** is an optional procedure. For each load there is one "Group1" group predefined by default.

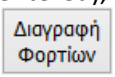
EXAMPLE:

To define a new group, for example, if you want to additionally define a Snow load belongs to the permanent loads and a Worker load that belongs to the mobile loads. First select the permanent loads then in the Description field type "Snow", select a colour for this load and press the "Insert" button. specific snow load is group (LG) 2. In the same way you define a second load group (Load Group) which is the worker's load which will belong to the mobile loads. Here again as default there is Group1 where the "classic" mobile loads will belong.

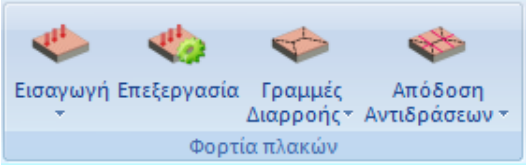


- With the "Delete" button you can delete a group of loads you have created.
  ⚠ *A prerequisite for deleting a group is that it does not contain any loads.*

Having multiple groups for the same charge allows you to individually (for each group) locate the charges of a charge using the "Show Charges" command.

- To delete the loads included in a group (loads you have already entered), select the specific group from the list of groups (so that it turns blue) and then . 

2. Plate loads



The "**Plate Loads**" command group contains the commands to import, edit and finally assign plate loads to the perimeter members and nodes.

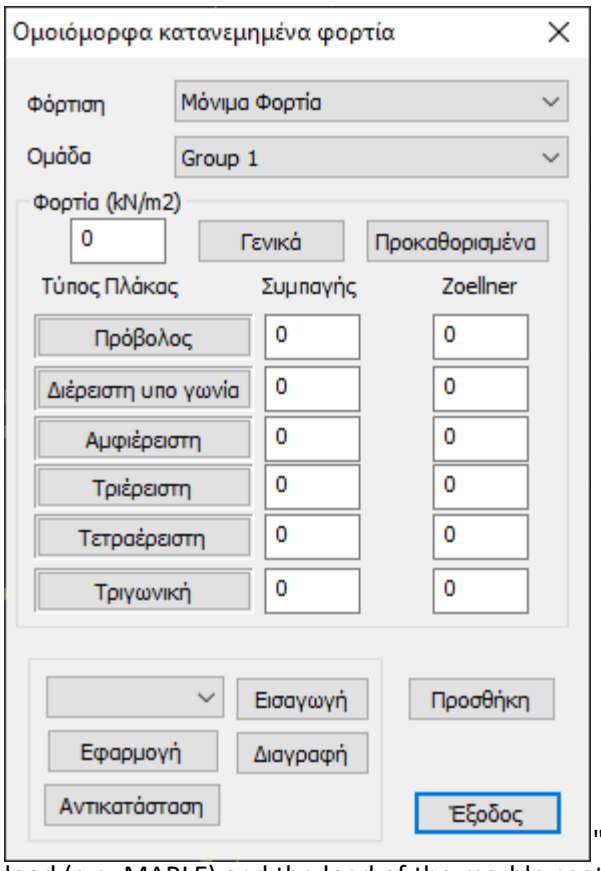
2.1 Introduction

Insertion of plate loads can be done globally per level or selectively per plate.

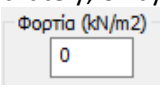
OBSERVATION:
After entering the plate loads, you can display their values via the Member Loads - Display command (§3.2) explained below.

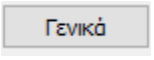
2.1.1 Overall

Total: to enter loads on all plates of the active level. In the dialog box:

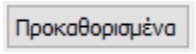


At the top you select the Load and the Group to which the loads you will set belong. Then enter the values of the loads for each type and type of plate.

- You can enter the load values, either by typing the values for each type of plate separately, or by typing a value (kN/m2) in the 

field and then pressing the 

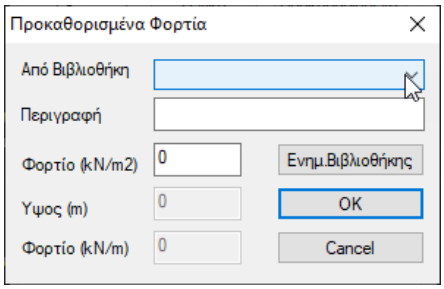
button, this value is passed to all fields.

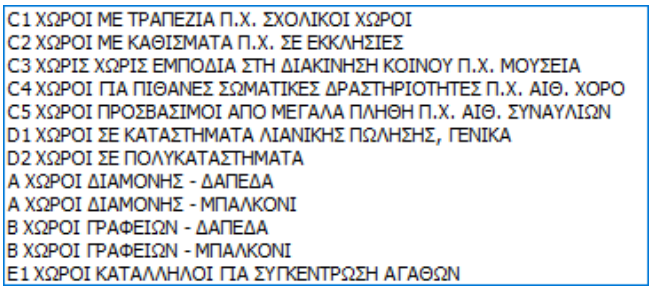
- You also with the option 

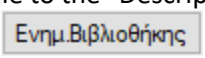
To import predefined load values from an existing library of values.

Selecting this option displays the following dialog box:

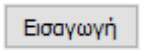
"From Library" list, you select the predefined load (e.g. MARLE) and the load of the marble coating is displayed. Having selected the *Mobile Loads* loading, you select one of the categories defined by the Eurocode depending on the use (A, B, C1, C2, etc.) to automatically enter the corresponding loads (without coefficients).





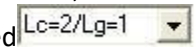
Similarly, you can define your own loads by giving a name to the "Description" field, entering a value in the "Load (kN/m2)" field and pressing the 

button, the load is permanently stored in the library.

After entering the values of the loads, press the "Insert" button 

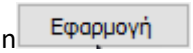
. Automatically, in the field to the left of the "Insert" button, the name of the load and group for the loads you have just entered is displayed 

. (Lc=1: Load 1/Lg=1: Group 1).

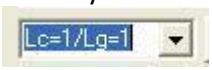
Then, you define the mobile loads and by pressing the "Insert" button, the name of the load and the group for the loads you have just entered is displayed 

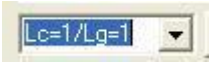
(Lc=2: Load 2/Lg=1: Group 1).

OBSERVATION:

Only after selecting the "Applybutton 

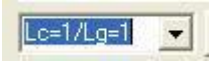
the loads will be applied to the active level plates. "**Insert**" simply inserts them into the list of those to be applied by pressing the "**Apply**" button.

- When you enter the loads for the first time, and select Apply**,** the loadings in the 

list will be applied to all plates.
- However, if you have already entered some loads, by selecting the "Apply" button, any loads you have entered will be replaced by the loads of the loadings available in the 

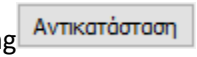
window.

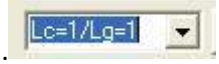
EXAMPLE 1:

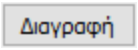
Suppose you have already entered the permanent and mobile loads for all the plates. If you enter new values for the permanent loads and do not enter for the mobile loads, by selecting the "Apply" button the program will apply the new permanent load values for all permanent loads and will reset the mobile values because there will be no mobile loading in the 

list. So selecting the "Apply" button will only apply the loads that exist in the list.

If in the previous case you wanted to replace the values of charge 1 (Permanent) with the new values and at the same time keep the mobile values, should select the "**Replace**" button.

So by pressing 

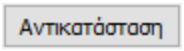
the existing values are replaced with the values you have just set and are present in the active window . 

With the "Deleteoption 

you delete the loads from a charge or from a charge group. The values of the loads are reset to zero and the load is deleted from the loads window.

EXAMPLE 2:

Suppose you have already set the permanent (charge 1, lc=1) and mobile (Lc=2) loads. If you set again only the permanent loads and press the "Apply" button, the program understands that there are no mobile loads (in the 

window there are none) and will replace the permanent ones and delete the mobile charge. However, if instead of the "Apply" button you select the 

button , then it will replace the specific load values for that group and charge.

There are two more options for which it is not necessary to use the "Insert" button. The first is to use the "**Add**" button.
By using this button you can add loads to the existing ones you have already defined.

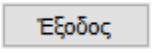
EXAMPLE 3:

Suppose you have set a permanent load of 1 KN/m2 as an overlap and you want to add, separately, another permanent load of 2 KN/m2. You define it in the corresponding fields of the plate types and press the "Add" button. The load is added as a separate entry in the plate.

You also have the option, depending on the type of plate, to replace individually, one price.

Type the value in the corresponding field 

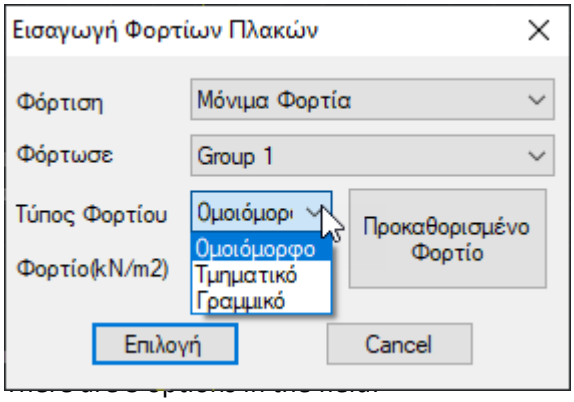
depending on the type of plate and press the corresponding button. This value is entered directly and replaces any other value for the specific type of charge and type of plate.

Finally, by selecting the "Exitbutton 

you close the dialog box without applying any of the changes you have made.

2.1.2 Selectively


Selectively and left click inside a plate. In the dialog box:

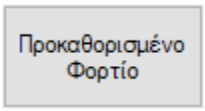


in the "Load" and "Group" fields you define to which load and group the load you are going to set will be assigned.
Then you select the type of load.

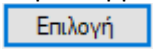
- **Uniform**
  A uniform load is imposed on the entire surface of the plate. After all the elements for the uniform load have been defined, the plates on which the uniform loads will be applied are selected by pressing with the mouse on a point inside them.

- **Serial**
  A sectional load is imposed on a specific area of the plate. After entering the load data, select the plate to which the load is to be applied, then enter the direction of application of the sectional load and finally define the area of application of the load with a rectangle
  OBSERVATION:
  ⚠ When you introduce a sectional load on a plate for the one load distribution is normally assigned to the members and for the plate is obtained uniformly.

- **Linear**
  Linear load applied to a plate. The process of imposition is similar to that sectional. The position of the load on the plate is defined by giving the two edges of the load (start point and end point).
  OBSERVATION:
  ⚠ Linear and sectional loads, as far as reactions in the members are concerned, will be replaced by an equivalent uniform load that loads the whole plate.

Finally, as far as the "Default Loadoption 

is concerned, everything mentioned in the "Default" option of the Total Import applies.

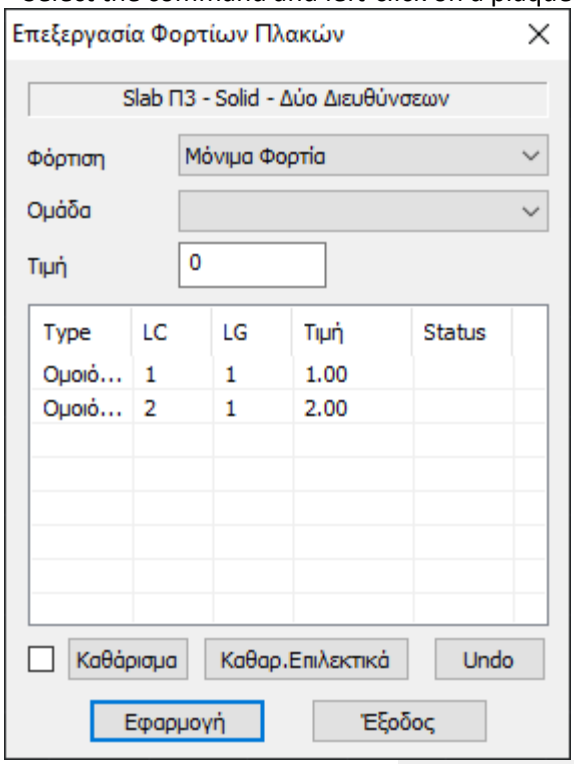
Clicking on the "Selectbutton 

closes the dialog box and you select the plate where the loads will be applied.

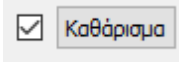
2.2 Edit

After importing the plate loads via the **Edit** command you can make modifications.

Select the command and left-click on a plaque. In the dialog box :



select charge and group. From the list loads, select a load and change the load to which it belongs, its group and/or its value.

Activate the command 

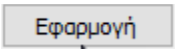
to delete <u>all</u> loads on that plate.

OBSERVATIONS:

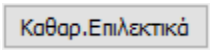
⚠ The shipments are not deleted immediately, but the word "Delete" is written in the "Status" column, which means that they are to be deleted.

Type	LC	LG	Τιμή	Status
Ομοιό...	1	1	1.00	Delete
Ομοιό...	2	1	2.00	Delete

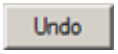
.

⚠ The final deletion is done by selecting the "Applybutton 

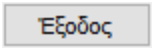
⚠ Selecting this button is also necessary to apply the changes you have made to the plate loads.

Selecting the "Clean Selectivelybutton 

deletes only the load you have selected in the table of plate loads. The logic and way of using the command is the same as that of the "Clear" command.

The choice of the "Undo" button 

undoes the cleanup, either total or selective.
It was mentioned earlier that using the "Clear" and "Clear Selectively" commands defines loads to be deleted by entering the "Delete" designation in the "Status" column. The 'Undo' command deletes the 'Delete' designation, in effect undoing the process of deleting the loads.

Finally, selecting the "Exitbutton 

closes the dialog box and you are returned to the interface.

OBSERVATION:

⚠ If you select the "Exit" button without first pressing the "Apply" button, all the changes you have made will not be taken into account.

2.3 Leakage lines

The calculation of the loading surfaces resulting from the geometric division of the floor plan surface, which are then used to calculate the design actions of the beams (surfaces whose loads will be applied to the beams),

The calculation is done automatically by the program according to the support conditions, either **Overall** per level, by simply selecting the command, or **Selectively**, by selecting the slabs one by one.

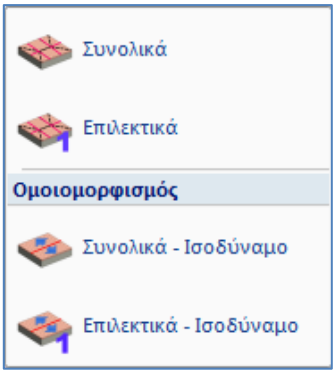
OBSERVATIONS:

⚠ The calculation of the Leakage Lines is for rectangular shaped plates or with rectangular modelling.

⚠ The calculation of the Leakage Lines depends on the support conditions.

⚠ The calculation of Leakage Lines is applied to conventional slabs and not to slabs with finite surface.

2.4 Reaction Performance



With this option the loads of the plates are assigned as reactions to the members defining the plates. More specifically, the loads are sent from the plates to the beams and nodes, based on the geometric division made previously (yield lines) (from the plates to the beams and nodes).

2.4.1 Overall

 to assign the loads of all the plates of the active floor.

2.4.2 Selectively

 to attach the loads of the selected plates by left-clicking its surface.

2.4.3 Homomorphism

means : to assign (globally  or selectively  , respectively) the loads of the plates to the members, but without the geometric division of the leakage lines into rectangles and triangles, but by reducing the whole surface, corresponding to the member, to an equivalent rectangle.

OBSERVATIONS:

- The attribution of plate loads as reactions to members, for plates that do not need modeling, is done with accurate distribution without them being homogeneous (uniform, triangular, trapezoidal, etc.). The program sends loads to beams and also directly to nodes corresponding to columns.

- In slabs that are defined by user-entered mathematical members (e.g. high stiffness beams between nodes of basement walls), the loads are assigned to the mathematical members, regardless of whether the mathematical or physical members (walls, columns, etc.) have been selected during modelling and matching of slab sides.

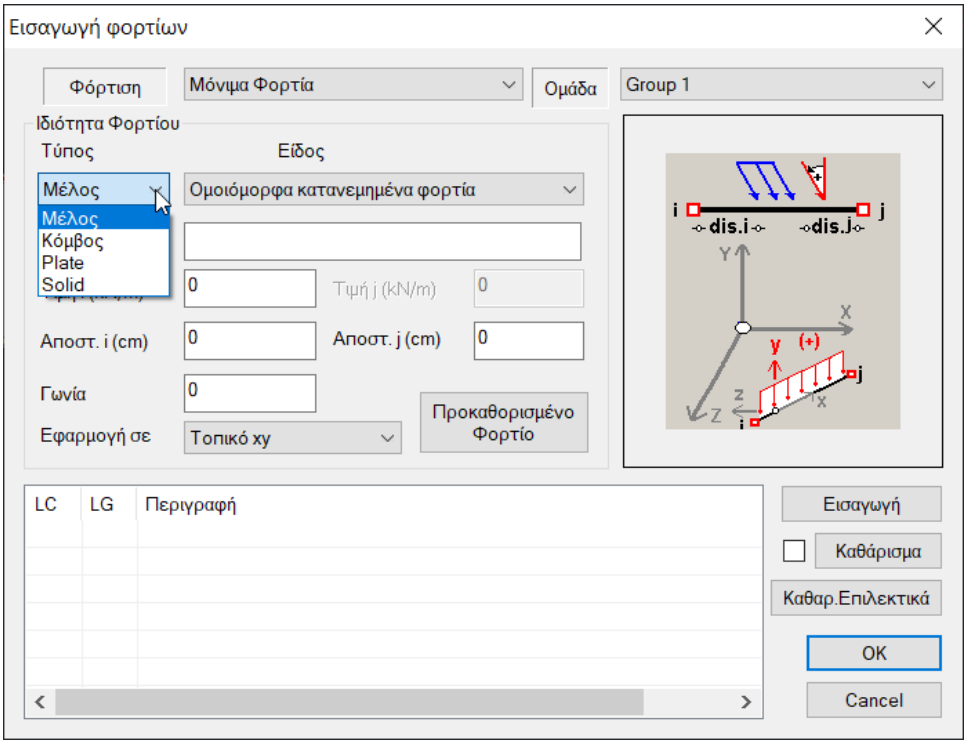
3. Member Loads

The "Member Loads" command group contains the commands to import, edit and finally display and copy member, node and surface loads.

3.1 Introduction

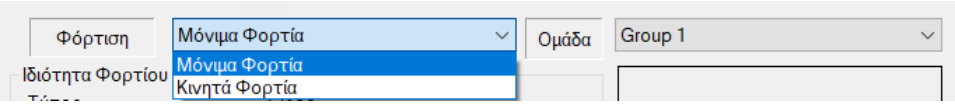
loads on members, nodes and finite surface elements.

Select the command and then show the elements of the member (members, nodes, finite surfaces) on which the loads will be applied. These elements can be selected in one of the known ways  . When the selection is complete, press the right mouse button and the following dialog box appears:

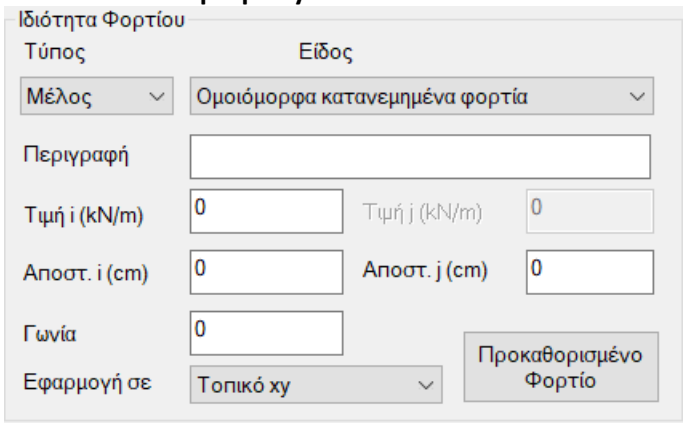


The lists contain all possible types of loads depending on the type of element.

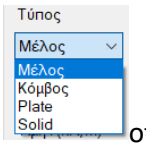
At the top you select the load and the group to which the load to be imported belongs.



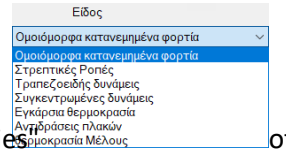
- In the **"Load property"** field:



You choose from the lists:



the "Press" of the item to load and



the "Species" of the cargo

According to the "Type" of the element and the "Type" of the load, the "Load Property" field is modified, some fields related to the geometry of the loads are activated and others are deactivated, as well as the drawing of the corresponding loads on the right is modified. Fill in the fields based on the drawing, entering a description, the values, the individual distances from the edges and anything else required depending on the "Type" and "Kind".

3.1.1 Member loads

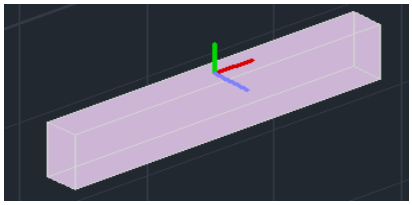
❖ **Marking of member loads :**

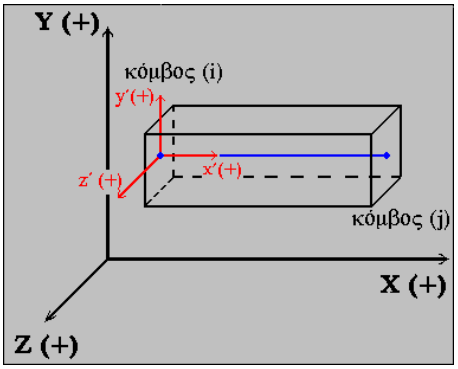
The loads shall be prefixed with respect to the local axis system of each member, which shall be located by right-handed triangular coordinate system rule.

In particular :

- **DOKOI** :

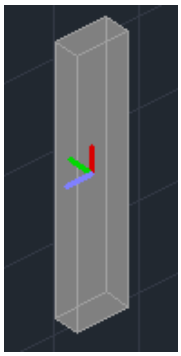
x-x is the axis of the beam with direction from the beginning to the end (red),
y-y is the axis (perpendicular to the local x-x) whose direction "pierces" the plate (green). Its direction is always in the positive direction of the y-y universal (from bottom to top).
z-z, the third axis, perpendicular to the plane defined by x-x and y-y (local) (blue).

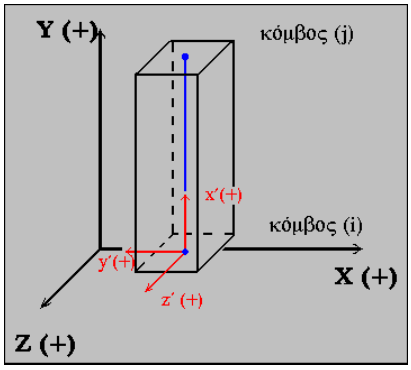




- **PENS** :

x-x is the axis of the pole with direction from its beginning to its end (from bottom to top) (red),
y-y is the perpendicular to the local x-x axis and facing the universal -x (green).
z-z the third axis perpendicular to the plane defined by x-x and y-y (local) (blue).





Another way of defining the local axes for both beams and columns is the right-hand rule with the thumb on the positive x-x axis, index on the positive y-y axis and the middle on the positive z-z axis.

Loads on members are considered positive (+) when they are opposite to the local axes.

3.1.1.1 Temperature Change Member

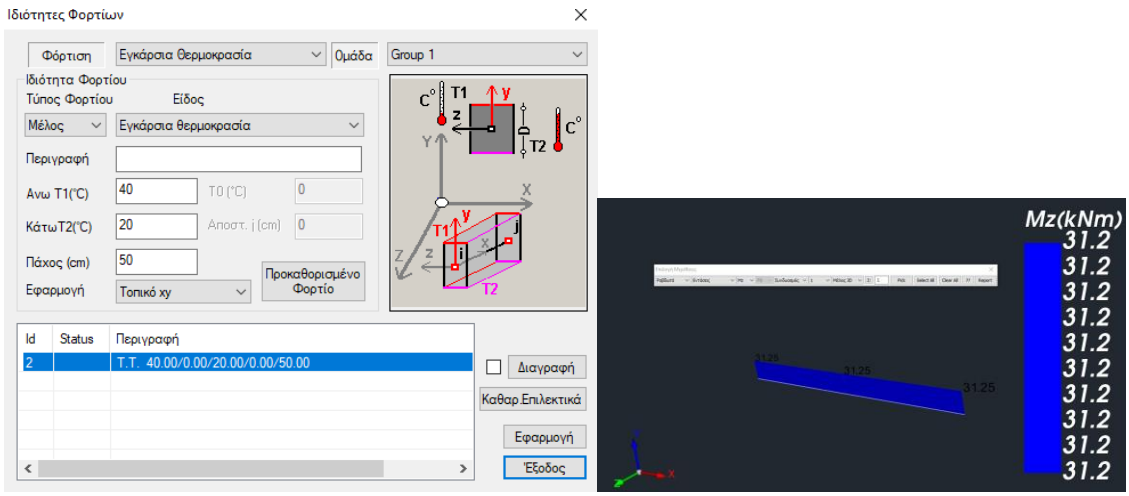
There are 2 types of temperature change in the member:

These two types of temperature input have no essential difference except that a temperature difference **ΔT** from a reference temperature, from which axial stress is applied, is defined in the temperature change. In both types there is a temperature definition of the upper fiber T1 and the lower fiber T2, the difference of which causes bending stress.

Finally there is also a temperature load input to the nodes:
When there is a diaphragm function the beams move uniformly and not deform horizontally.
So the temperature change is meaningless because they will not sense it.
So when there is a diaphragm function, the temperature change should be applied to the nodes which nodes have their own degrees of freedom.

A) "Transverse temperature" :

Which creates torque in the member due to the temperature difference between the upper and lower .
For example, if we want the difference to be 20◦C, we can set T1=40,T2=20 or T1=20,T2=0. As long as the difference is 20◦C.
As thickness we always define the dimension of the member in which the temperature difference is defined.



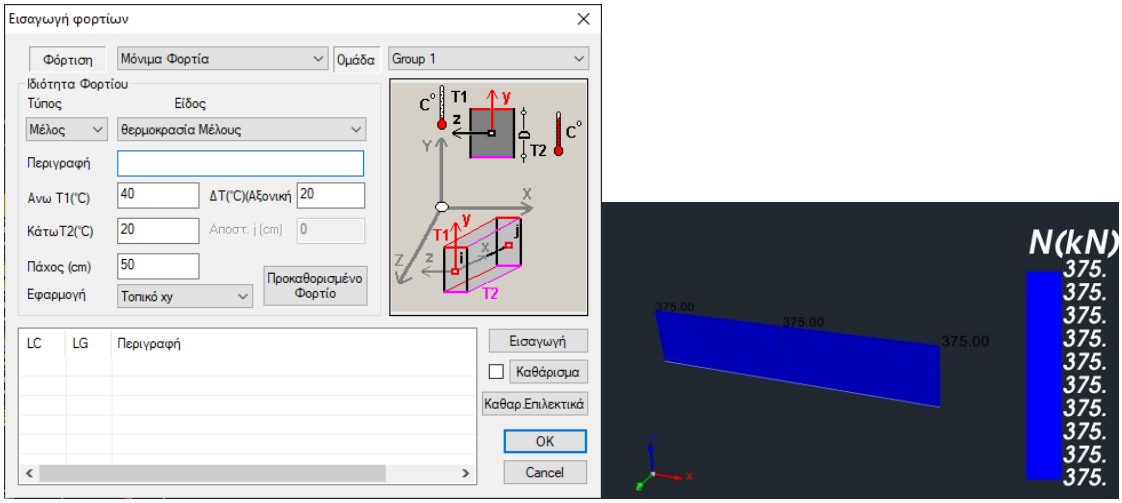
As thickness we always define the dimension of the member in which the temperature difference defined. The variation is by the thickness of the member.

Attention: Application to local xy produces Mz torque, while application to local xz produces My (applies to the right and left fibre)

B) "Member Temperature":

The left leg works in exactly the same way as the transverse temperature. To be ignored, it is sufficient that T1=T2.

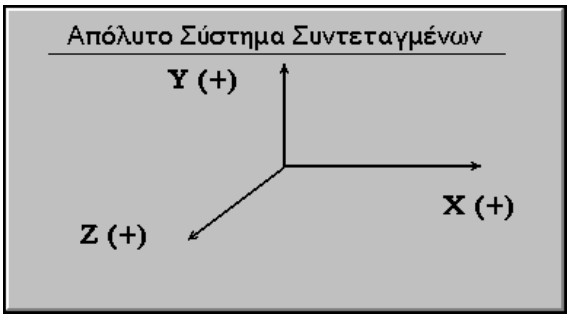
addition, a temperature difference ΔT can be defined, which is universal and stresses the member with axial tension



3.1.2 Node loads

- **Node load marking :**

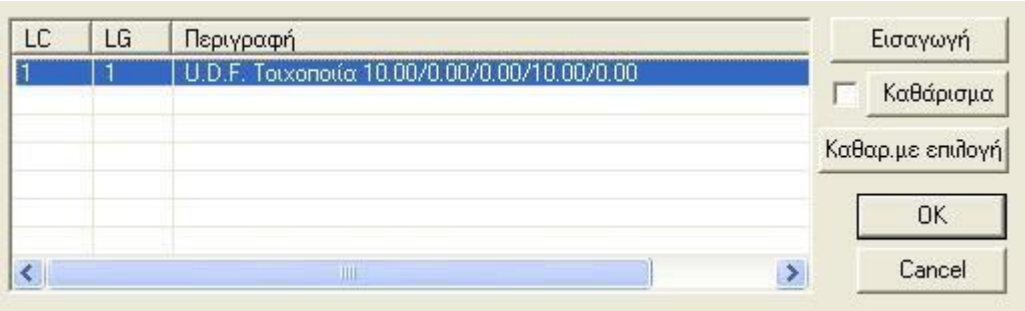
The loads on the nodes are given in terms of the universal axis system. This means that the "Apply to" option is always "General x,y,z".



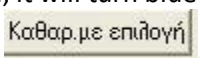
Positive loads are those whose buoyancy is homophonic and homomorphic on the universal axes.

- The next section of the load import dialog box is for viewing and deleting the loads you are importing.

By selecting the "Insert" button  and after you have specified a load in the way mentioned before, this load appears in the table with all its data.



EXAMPLE:

For example, enter a Uniformly Distributed Force (U.D.F.) that belongs to Load (LC) 1 (Permanent Loads) in Load Group (LG) 1. The numbers following the description (Masonry) are, in order, the value of the beginning load (10.00), the end load, the distance of the load from the beginning, the distance of the load from the end (10.00), and the angle of placement.
If you select a load in the viewer panel that you have already entered, it will turn blue and you can edit it. You can also delete it with the "Clear by selection" button  .

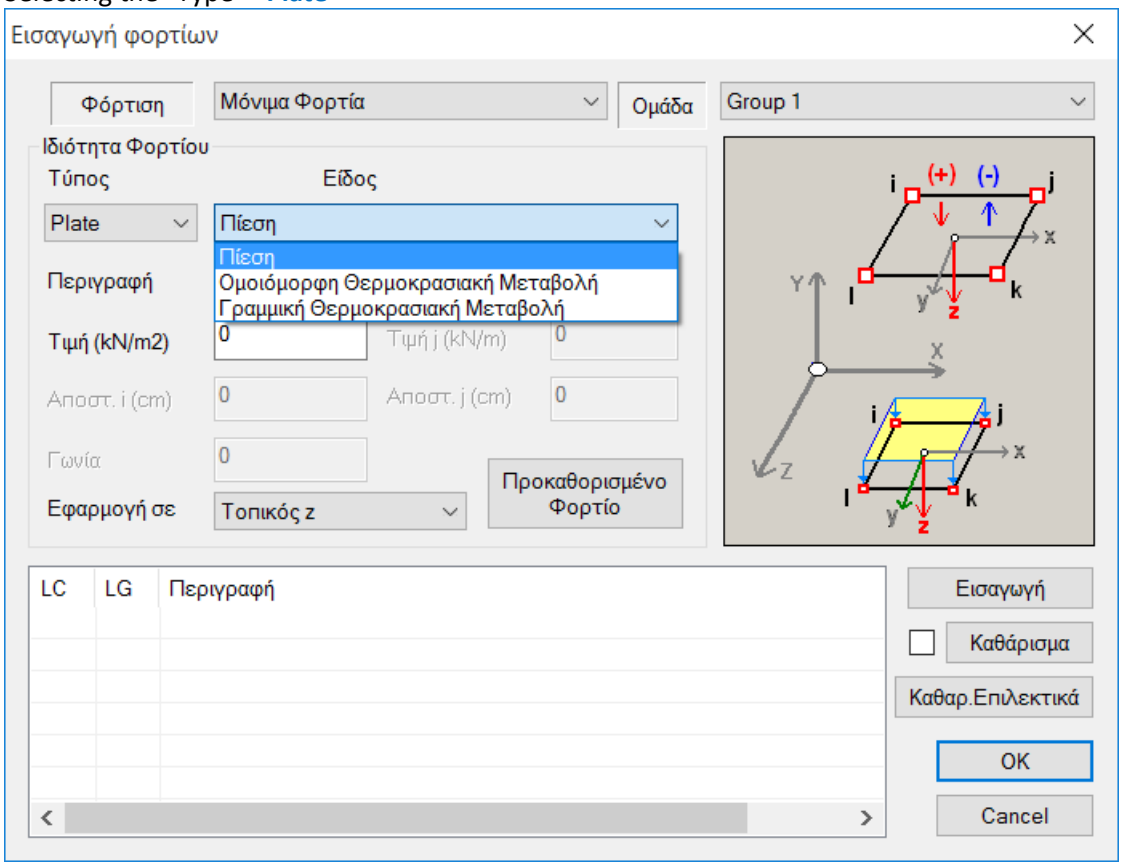
Finally, to delete all the loads that are in the dashboard, check the option next to the  "Clear" button.

Observation:

When a temperature load is applied to the nodes, only axial forces are generated and the existence of a diaphragm plate function is necessary

3.1.3 Loads in Plate elements

Selecting the "Type" **"Plate"**



You have the ability to set a **Pressure**, and the ability to enter a **Temperature Change** load for the finite surface elements has been added.

3.1.3.1 Temperature Change Plate

More specifically, for the Plate (shell) elements the Uniform Temperature Change charge and the Linear Temperature Change charge were added.

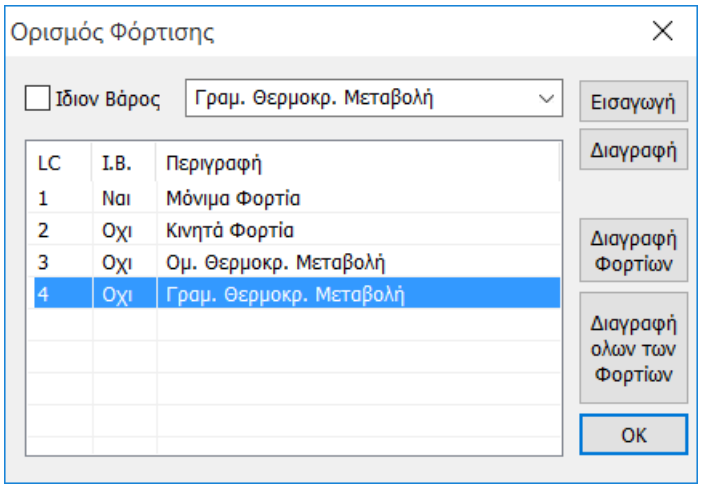
- The **Uniform Temperature Change** causes membrane deformation within the plane of the element, while
- The **Linear Temperature Change** causes bending deformation.

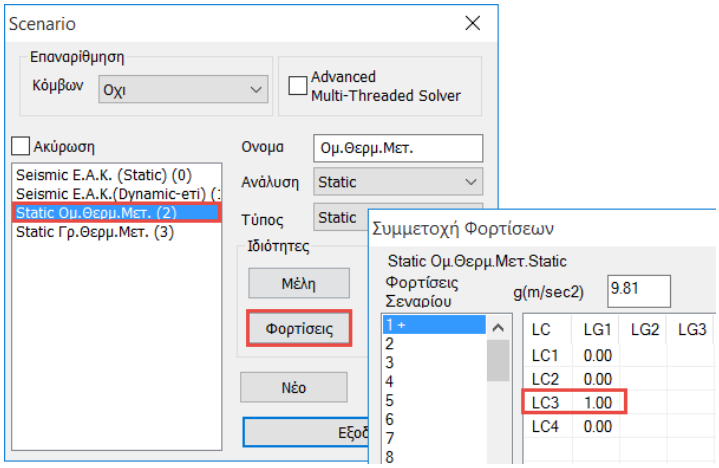
OBSERVATION:

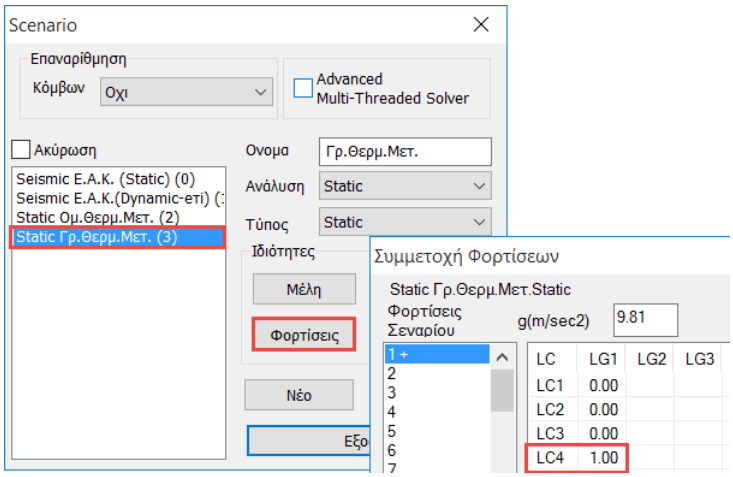
⚠ It should be noted that the two loads for the **plate** element, the Temperature Changes, can be included either in the same load or in two different loads.

- If you include both loads in the same analysis scenario you will get cumulative results but on one load (the first).
- If they join two different loads to obtain separate results, each load MUST go to a different analysis scenario.

The procedure to follow is as follows:







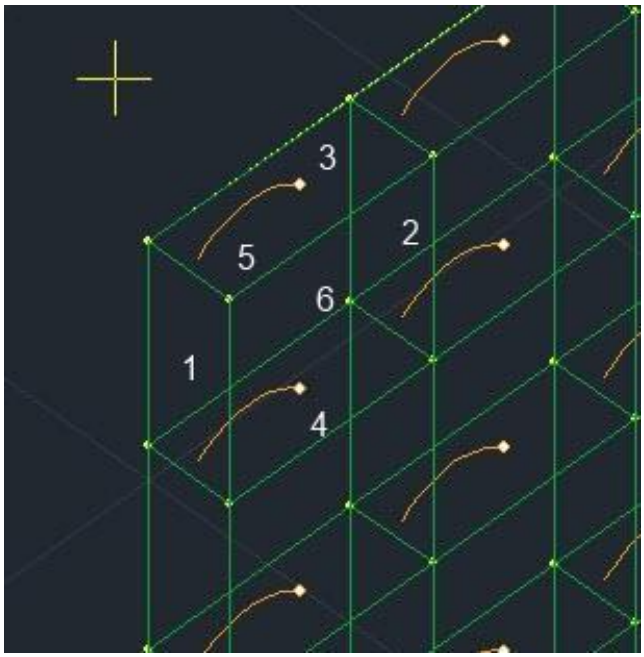
OBSERVATION:

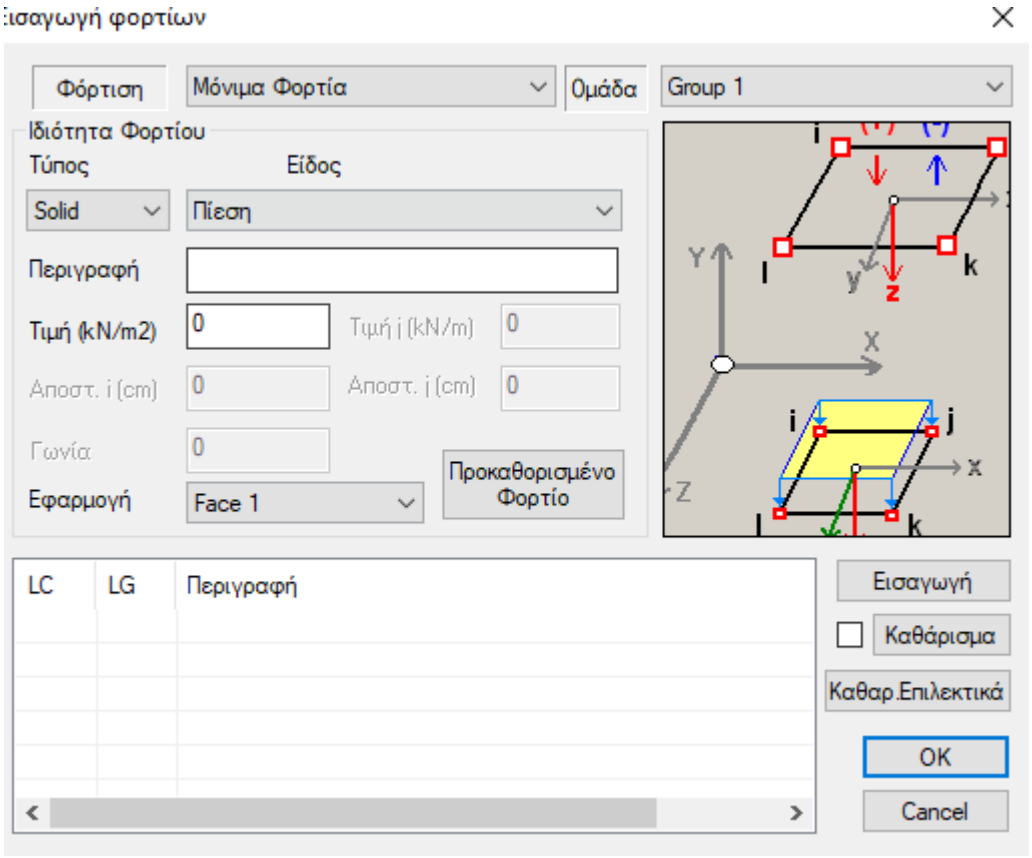
⚠ For the **Plane** elements (Stress, Strain, Axisymmetric) there is the possibility of imposing only **Uniform** Temperature Change.

3.1.4 Loads in Solid elements

Selecting the "Type" **"Solid"**

Regarding the loads, a new option has been added to the import of the load in the Solid element



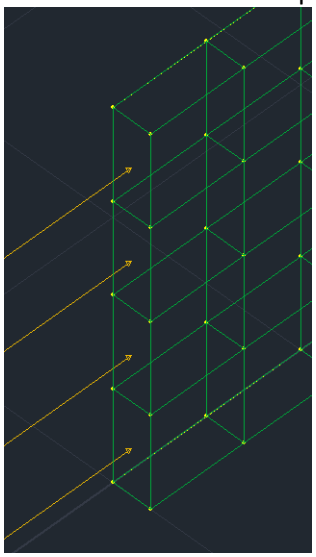


The load that can be introduced in Solid is a pressure load (as in Shell) except that the face to which it will be applied must be defined according to the following picture

OBSERVATION:

⚠ Grief is positive.

The vertical loads are placed on face 3 and the corresponding vertical loads on the wall surface on the inner face will be placed on face 6 and on the outer face on face 5.



In the picture above, a compressive load of 15 kN has been introduced on face 1.

OBSERVATION

The OUT file initially contains the Solid Elements data

```
*****************************************************
*           S O L I D   E L E M E N T S             *
*****************************************************
Αριθμός Στοιχείων              =   56
Αριθμός Γεωμετρικών Ιδιοτήτων  =    1

                Γ Ε Ω Μ Ε Τ Ρ Ι Κ Α    Σ Τ Ο Ι Χ Ε Ι Α

Αριθμ  Πυκνότητα      ALPHA(X)     ALPHA(Y)     ALPHA(Z)     G(XY)
Υλικ.     E(XX)        E(YY)        E(ZZ)        v(XY)        v(XZ)        v(YZ)
-------------------------------------------------------------------------------
    1   0.250E+02    0.100E-04    0.100E-04    0.000E+00    0.125E+08
        0.300E+08    0.300E+08    0.300E+08    0.200E+00    0.200E+00    0.200E+00


---------------- Φ Ο Ρ Τ Ι Α --------------
Αριθμ.      Είδος           Face      Τιμή
-----------------------------------------
    1     Κατανεμημένο       1        15.000
```

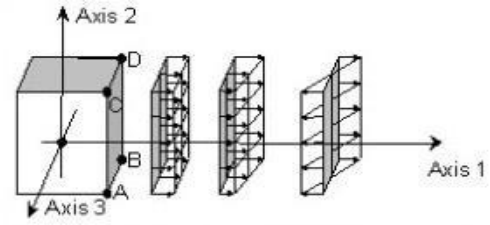
In the picture above, the material used is indicated with its characteristics and a load applied to face 1.
The following are the surface data of the surface

```
                    ΔΕΔΟΜΕΝΑ ΕΠΙΦΑΝΕΙΑΚΩΝ ΣΤΟΙΧΕΙΩΝ

Αριθμ. ----------------Συνδεσμολογία----------------  Ιδ.Κατανεμημενο Φορτιο
Στοιχ.   I      J      K      L      I*     J*     K*     L*   Υλ.   Φ1   Φ2   Φ3   Φ4
-------------------------------------------------------------------------------------
    1   185    183    152    154    186    184    153    155    1    0    0    1    0
    2   154    152    121    123    155    153    122    124    1    0    0    1    0
    3   123    121     90     92    124    122     91     93    1    0    0    1    0
    4    92     90     59     61     93     91     60     62    1    0    0    1    0
    5   183    181    150    152    184    182    151    153    1    0    0    0    0
```

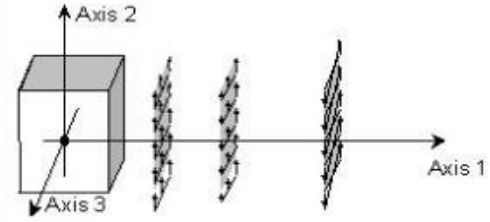
More specifically, the number of the element, its 8 nodes, then which material property it obeys (here it is all 1) and finally which load and which load it obeys. Here we see that the first 4 elements, under F3 have the value 1 which means that in loading 3 they have received a load with property 1 i.e. the distributed load with the value 15 kN applied to face 1.
changes in terms of the effects of the movements and rotations of the nodes.
What's new is in the trend results. More specifically, 6 stresses are calculated, 3 principal and 3 shear stresses **ALWAYS IN THE TOTAL SYSTEM OF CONDITIONS.**

```
                          N O D E   S O L I D   E L E M E N T   S T R E S S
ELEMENT   LOAD LOCATION          S-XX          S-YY          S-ZZ          S-XY          S-YZ          S-ZX


      1     1        1  -0.285032E+00  -0.327794E+01   0.329177E+01   0.113687E-12  -0.265389E+01  -0.710543E-13
                     2  -0.285032E+00  -0.327794E+01   0.329177E+01  -0.298946E-01  -0.265389E+01  -0.177321E+00
                     3  -0.285032E+00  -0.327794E+01   0.329177E+01   0.298946E-01  -0.265389E+01   0.177321E+00
                     4   0.669828E+00  -0.389153E+01   0.336003E+01   0.682121E-12  -0.318938E+01   0.284217E-13
                     5  -0.123989E+01  -0.266435E+01   0.322352E+01   0.227374E-12  -0.211840E+01  -0.182016E-12
                     6  -0.442730E+00  -0.358511E+01   0.191362E+01   0.703661E-12  -0.299481E+01  -0.170530E-12
                     7  -0.127334E+00  -0.297077E+01   0.466993E+01   0.227374E-12  -0.231297E+01   0.284217E-13
```

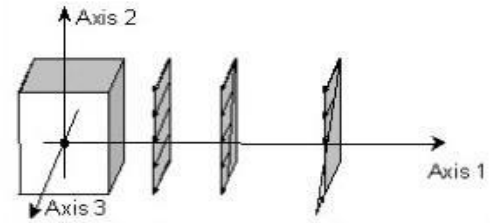
These stresses are calculated for each element (ELEMENT column) for each load (LOAD column) at 7 different points:
At the centroid (point 21 in the original figure) and at the centres of the 6 faces (22-27 in the original figure). That is, point 1 is 21, point 2 is point 22, etc.
For an easier understanding of the above trends and how they develop on the surface, I attach the figure below.
The convention for shear stresses is that their indices mean:
The first indicator is the axis along which the plane in which the shear stress acts is perpendicular. The second is the axis that is parallel to this trend. It is shown
that for shear stresses σij = σji



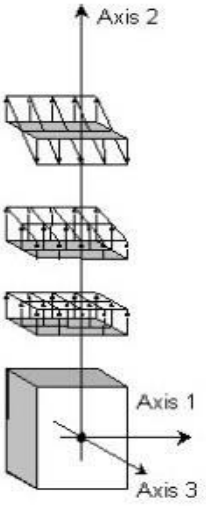
a. Examples of membrane direct stresses, S11



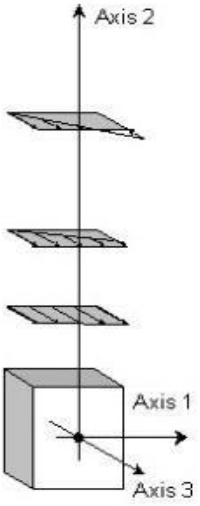
b. Examples of membrane shear Stresses, S12 (S21 stresses similar)



c. Examples of plate transverse shear stresses, S13



d. Examples of membrane direct stresses, S22



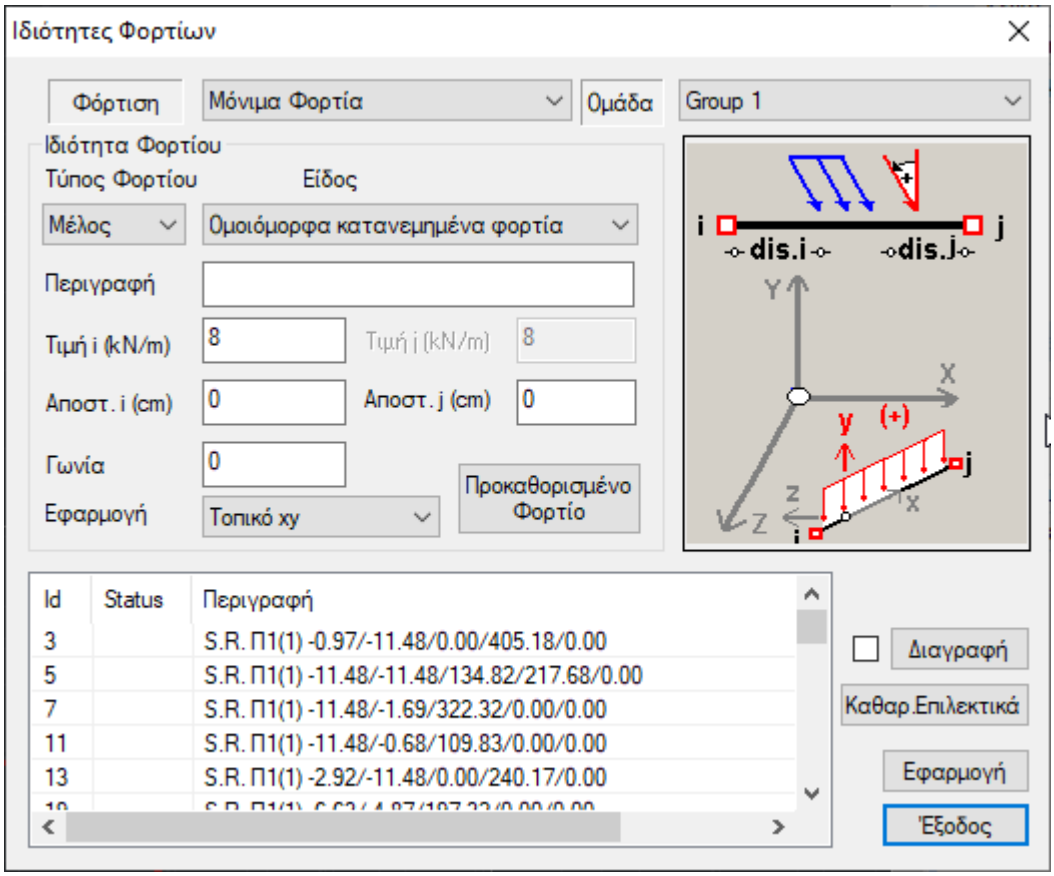
e. Examples of plate transverse shear stresses, S23

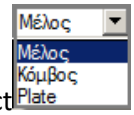
3.2 Edit

of existing loads on members, nodes and finite surface elements.

3.2.1 Overall


To process in total all the loads present in the active level. Select the command and in the dialog box:

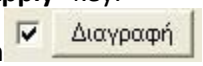




you are given the option to change or delete these loads altogether. Simply select item and from the list, the load you want to change and make the changes in the fields above.

For example, if you want to change the total load of a masonry in all the members where it has been applied, you simply select it and change it. After the changes are complete, you press the "**Apply**" button.

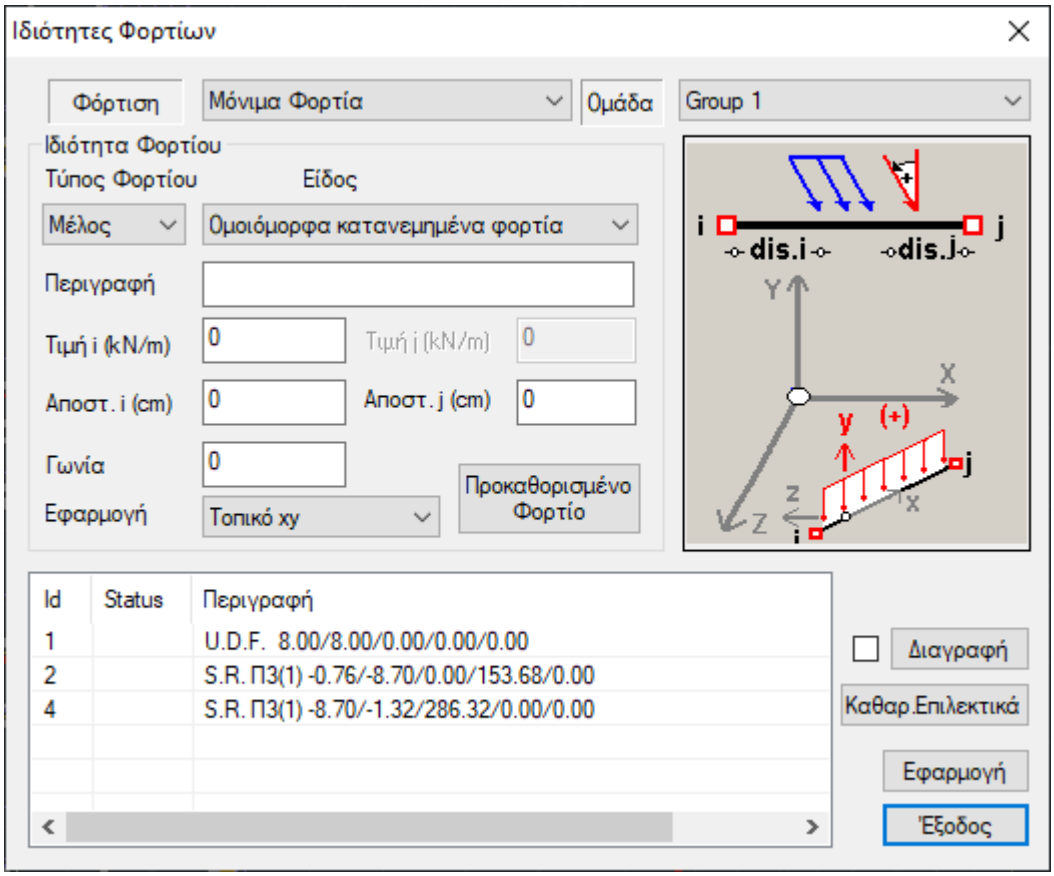
By using the "**Clear. Selectively**" to delete the load you have already selected.
Pressing the button does not automatically delete the load, but the word "Delete" is displayed in the "Status" column, indicating that it is to be deleted. If you press the "Clear" key again, the "Clear" key will be deleted. Selectively" the deletion is cancelled. The definitive deletion is done by pressing the "**Apply**" key.

The "Deleteoption 

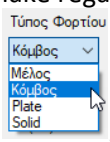
works in a similar way and refers to a mass deletion of the level loads. For its operation, it is necessary to check the option in front of the "Delete" button  . The deletion of loads is not immediate. The final deletion is done by pressing the "Apply" button.

3.2.2. Selectively

 To edit single member, node or surface loads.
Call the command select one or more members, nodes or surfaces and then press the right mouse button to indicate the end of the selection and the following dialog box appears:



The loads are displayed at the bottom of the window. The list contains the loads of the selection you make regardless of type of item and is displayed by selecting the



Press .

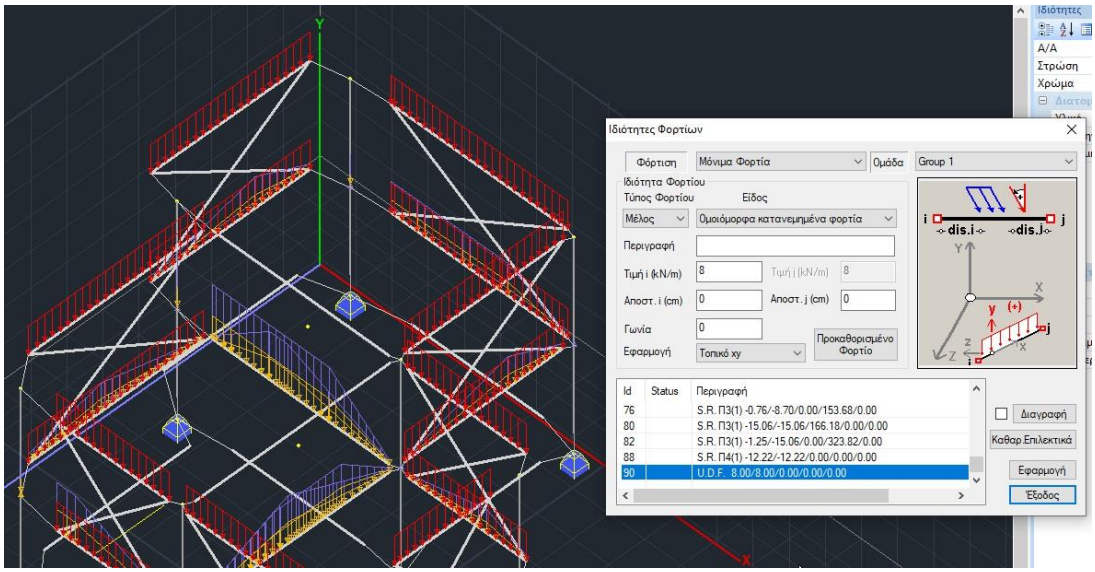
EXAMPLE:

In the example in the figure for a selected member, there are the reactions of the P1 slab (S.R. Slub Reactions) and a Uniformly Distributed Force (U.D.F.) masonry load.

Selecting a load displays its values at the top of the window where you can change them.

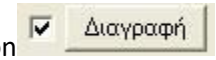
OBSERVATION:

In the new version of the program, by selecting a load in the list, all loads belonging to the same load and having the same value are automatically reddened in the 3D representation of the vector.



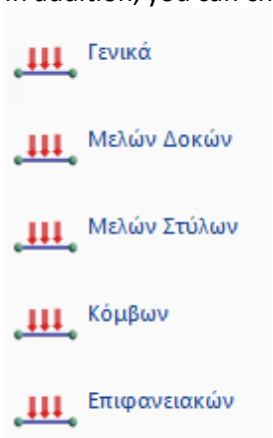
In this way, you can better control the loads that have been applied to the elements of the structure and that will be affected by a possible overall modification.

You can delete a load by selecting it and pressing the "**Clear"** button. **Selectively**". Pressing the button does not automatically delete the load, but the word "Delete" is displayed in the "Status" column, indicating that it is to be deleted. If you press the "Clear. Selectively" the deletion is cancelled. The definitive deletion is done by pressing the "**Apply**" key.

The "Deleteoption 

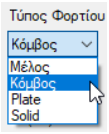
works in a similar way and refers to a mass deletion of the level loads. For its operation, it is necessary to check the option in front of the "Delete" button  . The deletion of loads is not immediate. The final deletion is done by pressing the "Apply" button.

In addition, you can choose to edit loads by:

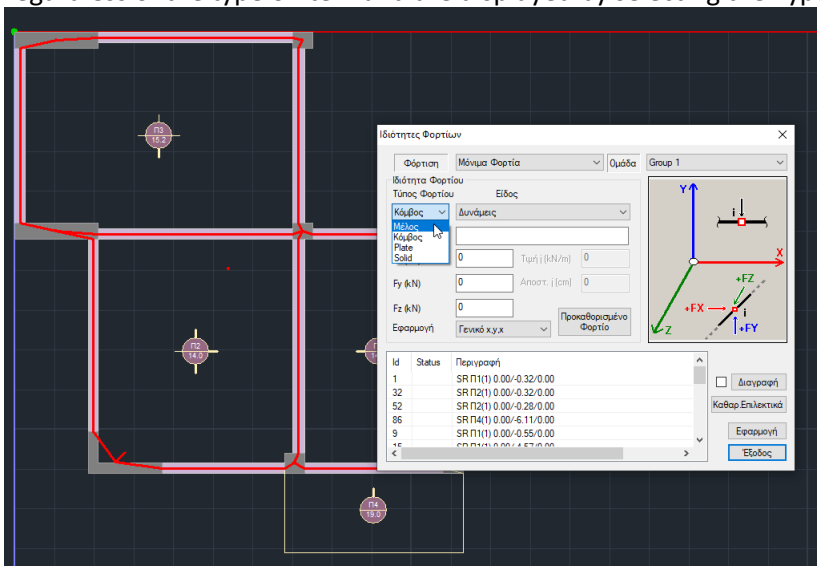


So by selecting the loads of one item category only, even if you select other item categories in total (e.g. with a window), the loads that will appear in the list will only be for the "per" selection.

General is the same as the command  and displays the loads of the selection you make



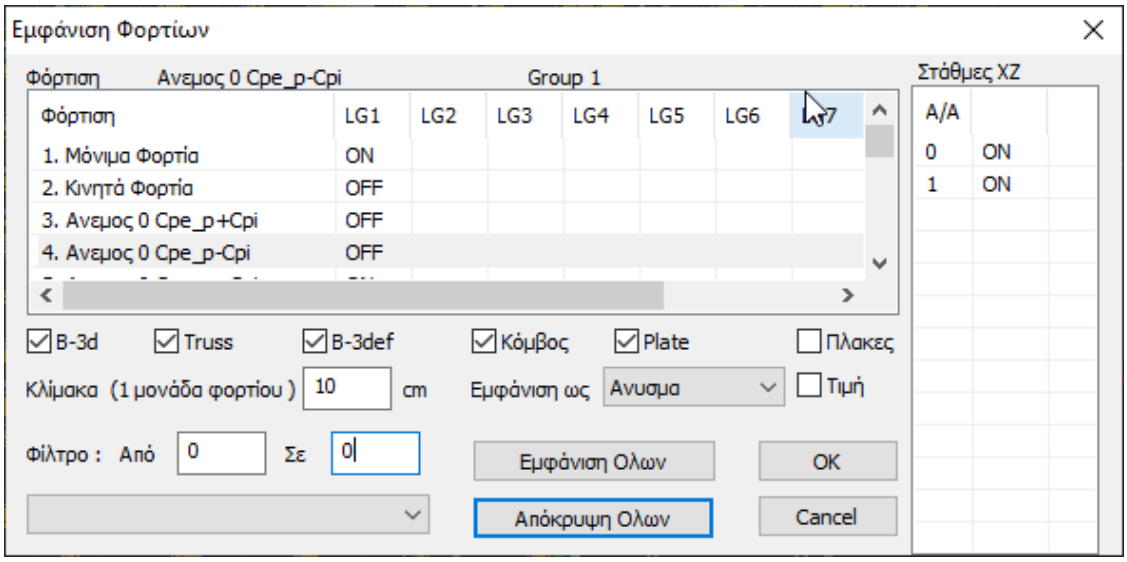
regardless of the type of item and are displayed by selecting the Type



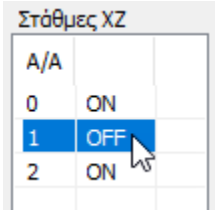
3.3 Show

to display the loads on members, nodes and surfaces, either by level or in total, either by vector or by number.

The display of numbers can be done in the plan view and in the three-dimensional mathematical model while the graphical display of the vectors is done only in the three-dimensional mathematical model.
Select the command and in the dialog box:

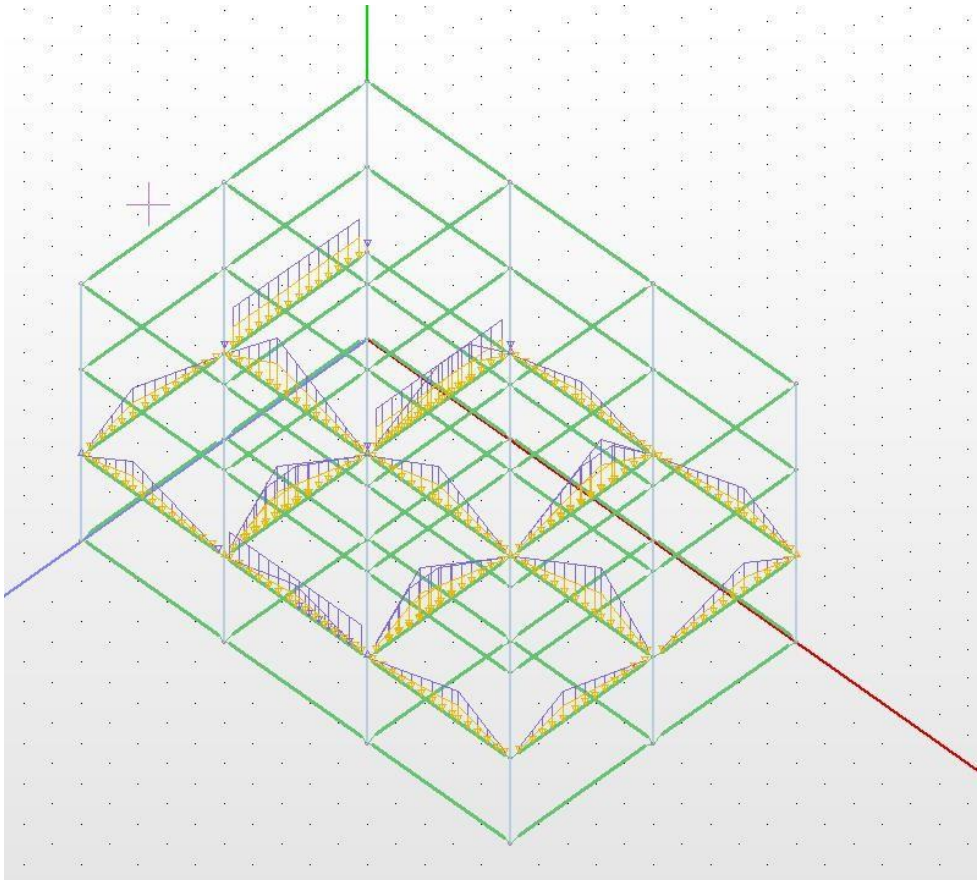


at the top and in the Load column, the loads you have defined are displayed, with the corresponding description for better and immediate monitoring. The remaining columns refer to the Load Groups (LG Load Groups) that each load contains.
Each column contains an ON or OFF switch in the charge groups you have already defined. The ON option is changed to OFF by clicking on it.
The above image shows four loads Permanent, Mobile and two winds. Each load contains a default LG1 group which includes all the loads of each load. For Permanent loads the LG1 display is ON while for Mobile and Winds it is OFF. In this state only the Permanent loads will be displayed.

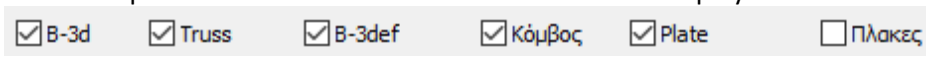


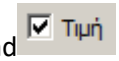
In the "Levels" section you can display the loads per level by clicking on the ON and OFF of each level respectively.

Select the 3D view of the model to display the loads on the screen.

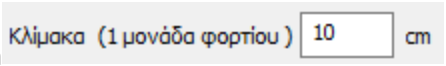


The next options concern which items the loads will be displayed on.

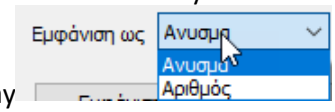


and 

to display the price of the loads.

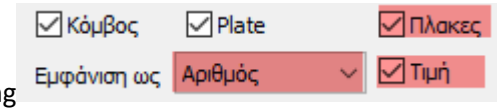
The next option 

concerns the scale of the graphical representation of the load cells. Enter how many cm a unit of load corresponds to.

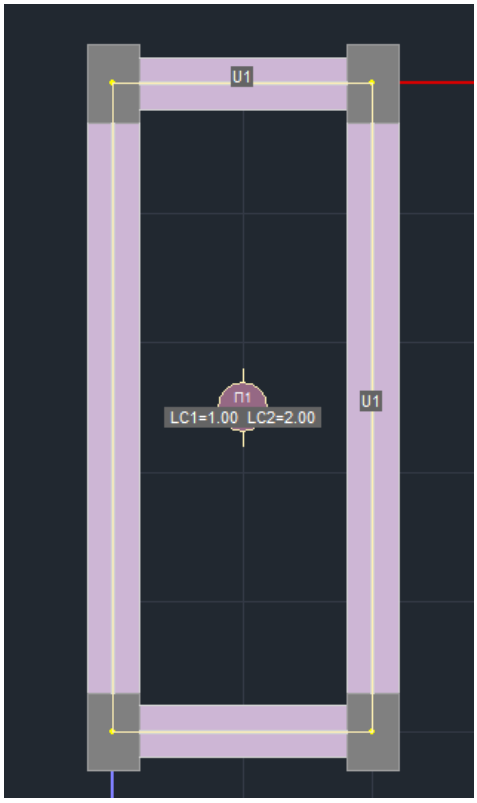
The next option is how to display 

loads.

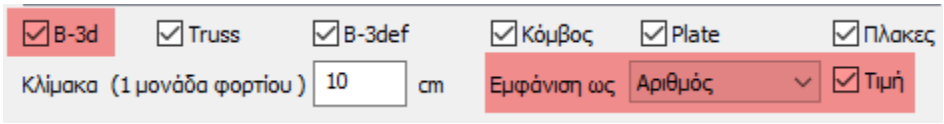
You can choose to display a vector or a number. The vector is only displayed in the 3D mathematical model. If you also check the "Value" option then values are also displayed in the graph of the loads with the vectors.

In addition, by selecting 

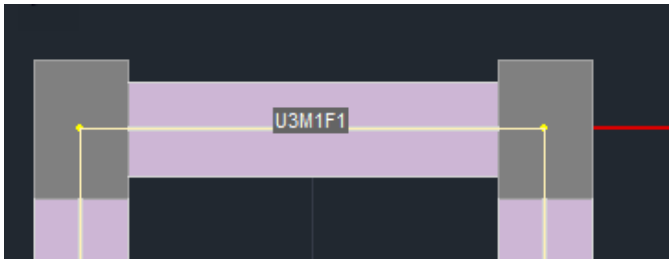
, inside the plates, in the 2D visualization, the values of the plate loads are displayed.

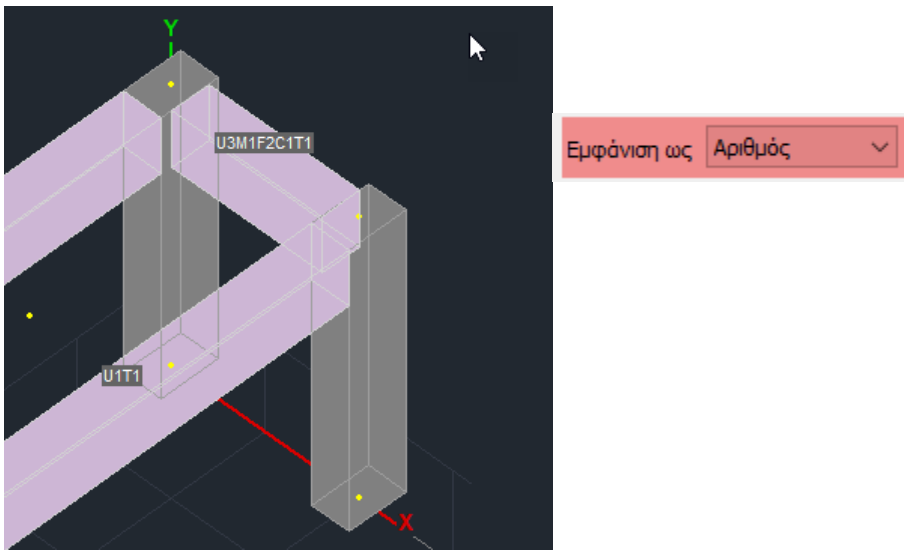


Similarly for members, with B-3d, Number and Price selected,

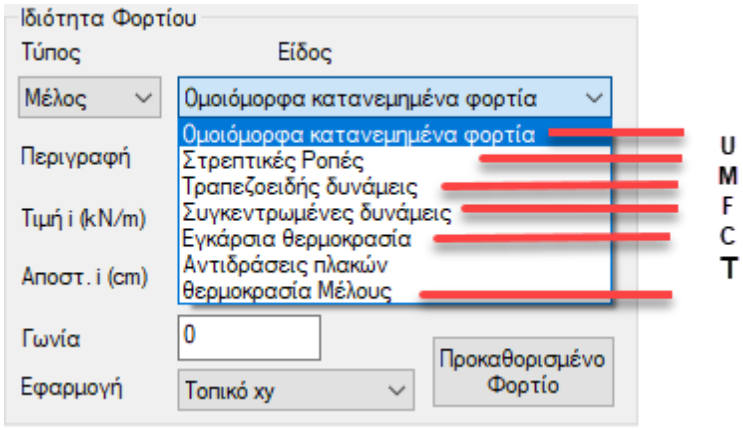


the load presence indicator is displayed on the member in letters and numbers,

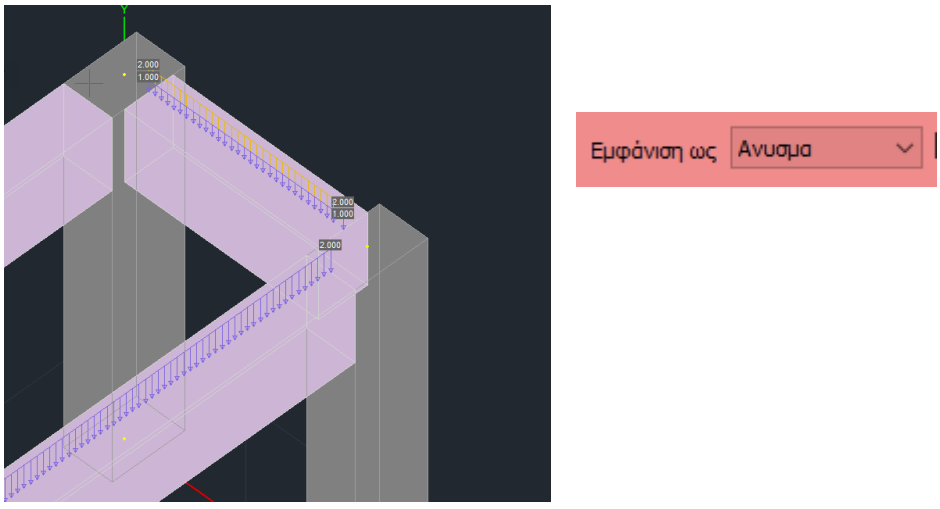


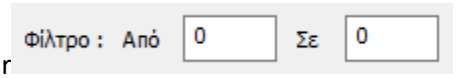


depending on the type of load (U,M,F,C,T):

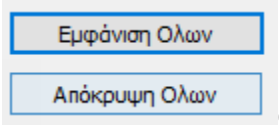


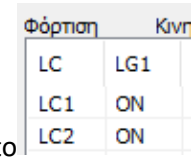
And the number indicating how many shipments of that species there are.



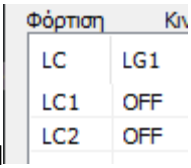
Finally, in the Filter 

option you can specify a range of values for the loads you want to display.



commands allow the loads to be shown or hidden globally, converting all the loads in the list to 

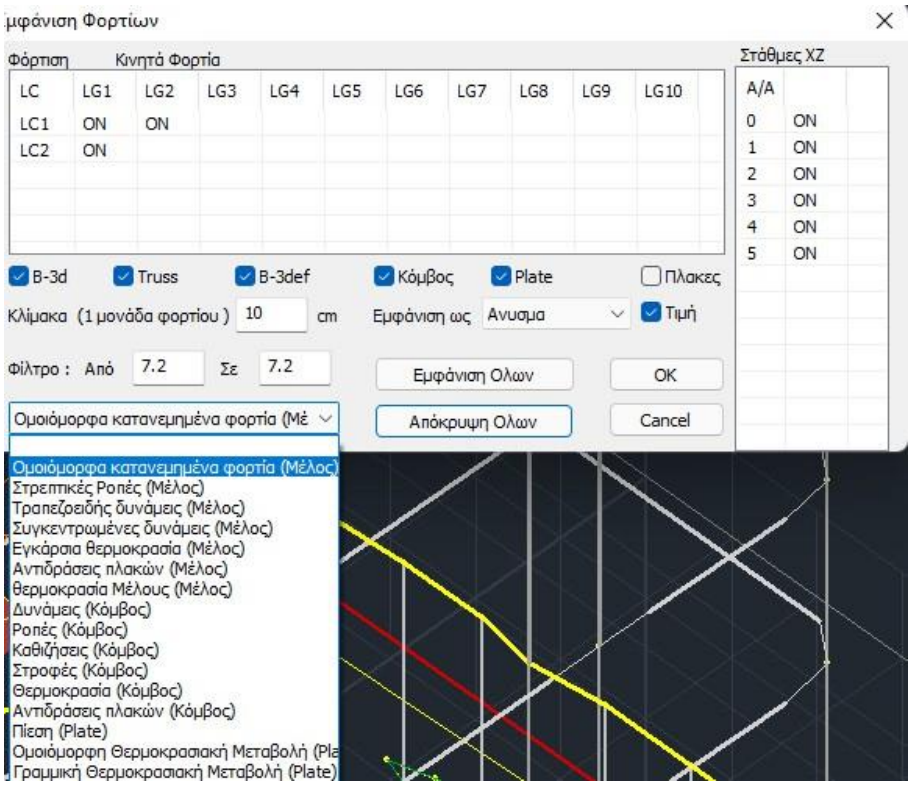
Φόρτιση	Κινη
LC	LG1
LC1	ON
LC2	ON

and 

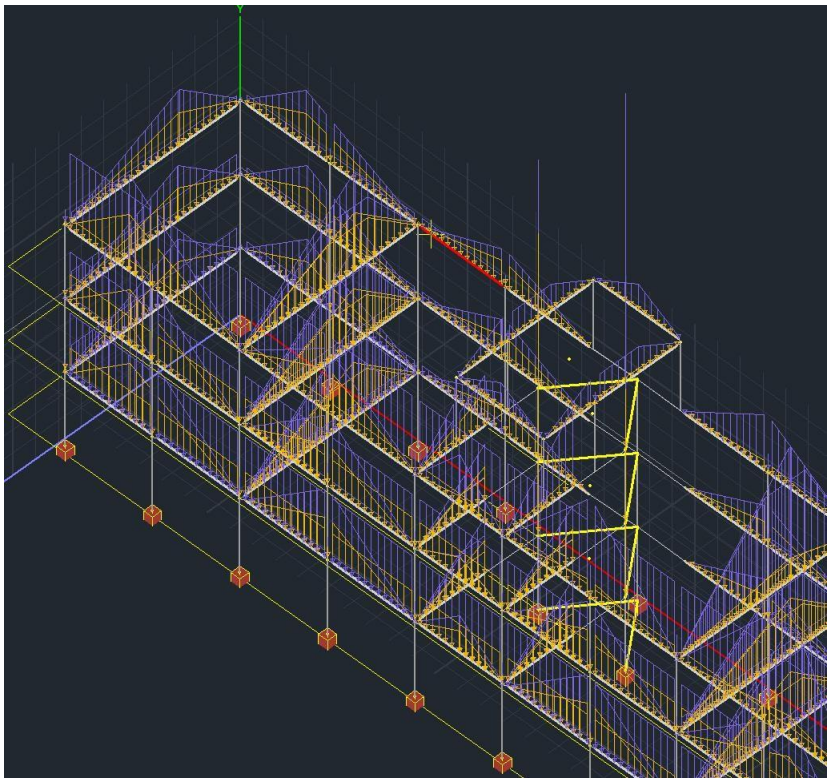
Φόρτιση	Κι
LC	LG1
LC1	OFF
LC2	OFF

respectively.

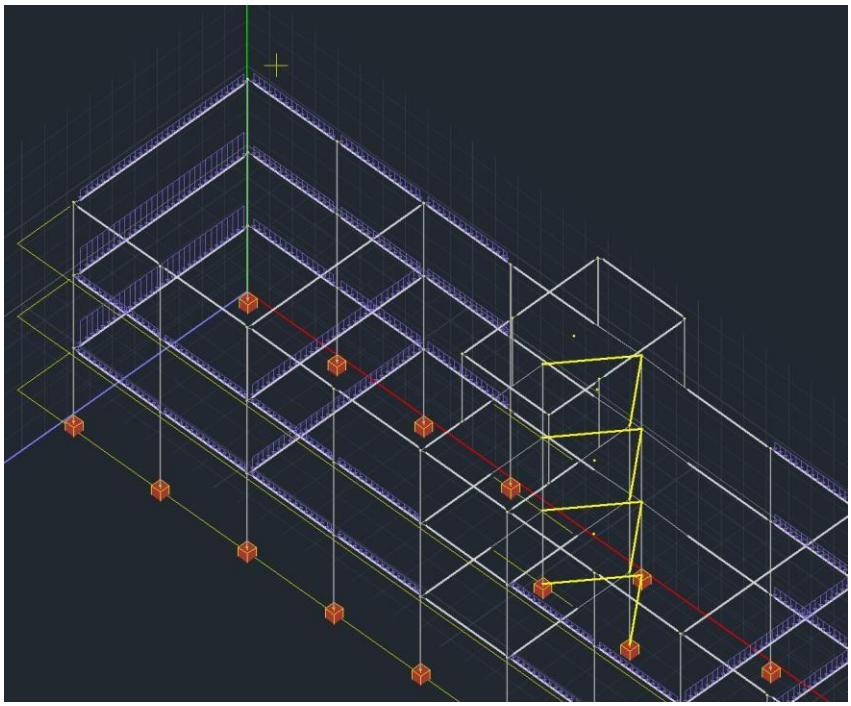
NOTE:
In the dialog box of the cargo display, two new possibilities have been added: depending on the type of cargo and on the range of values of the specific type.



For example, in the following vector, all types of loads are initially selected to be displayed:



By choosing to display only the uniformly distributed loads, only the corresponding loads are displayed:



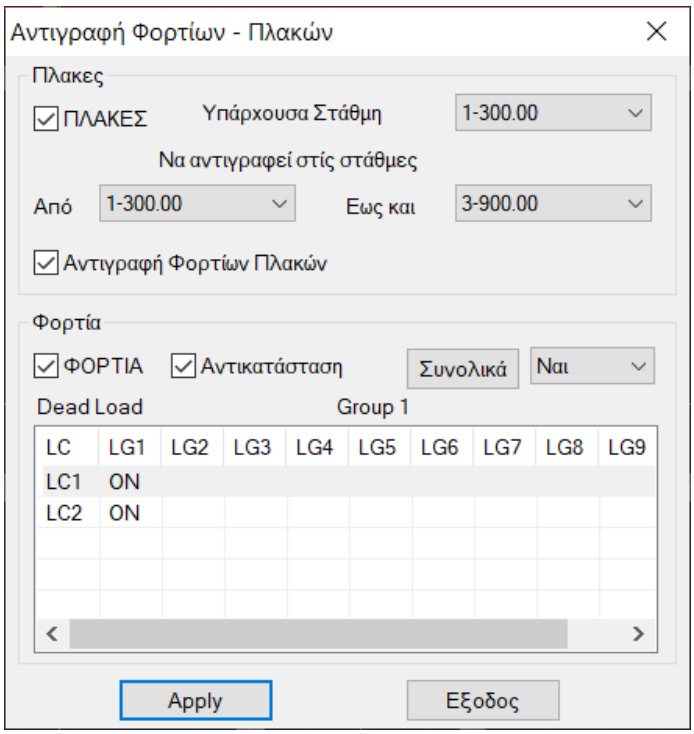
Finally, in the "Filter" indicator you can enter a minimum and a maximum load value in order to display only the loads with values in this range.

3.4 Copy

to copy plates and loads from one level to another.

OBSERVATION:
Use this command only when you have a standard floor, i.e. when the floors are exactly the same. Otherwise, copying will only be applied to the slabs that are exactly the same as those of the original floor.

Call the command and in the dialog box:



The upper part of the dialogue box is about the plates and their loads.
Specifically check the "**PLATES**" option if you want to copy the plates from one level to another.
You also specify the level you want to copy ("Existing Level"), as well as the level or levels to which the copy will be made.
The "**Copy Plate Loads**" option allows you to copy the plate loads as well.

The bottom part of the dialog box refers to the additional loads you have entered (masonry, linear, concentrated, etc.).
Check the "Loads" option if you want the loads to be copied and select ON on the loa
you want to copy.

LC	LG1
LC1	ON
LC2	ON

Using the "**Replace**" option will replace the loads, if any, on the other floors.

OBSERVATION:
If you do not select it, the loads of the level will be added to the existing ones.

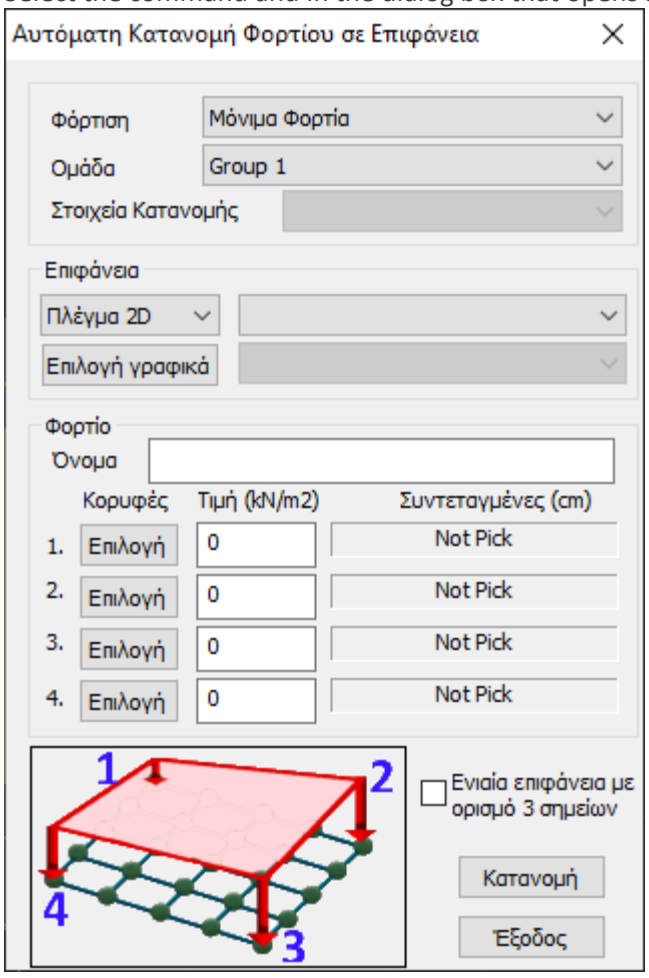
With the option "Total: YES OR NO" you copy the level loads in total or selectively per Group and per charge (LC).

3.5 Tools

3.5.1 Load Distribution on Surface

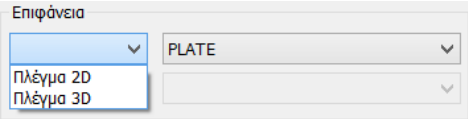
Command for automatic load distribution to surface elements

The new version of SCADA Pro includes a new tool for automatic load distribution and performance on surfaces simulated with finite surface elements.
Select the command and in the dialog box that opens set:



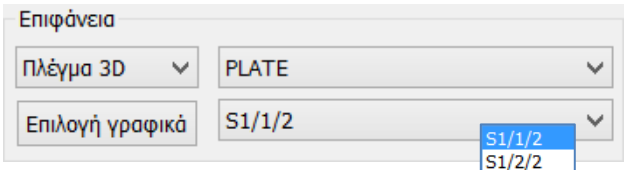
The type of load by selecting from the already defined loads and the corresponding group.

In the Surface field

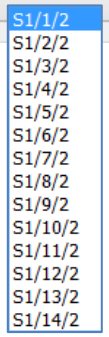


Select the type of surface to which the surface or surfaces be loaded belong.

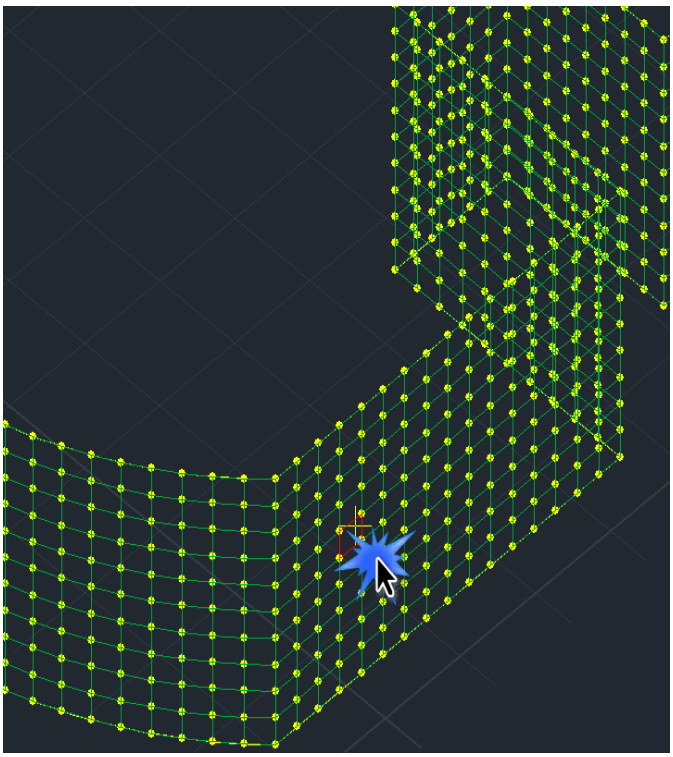
'When more than one surface has been defined with surface elements then you select the corresponding grid.



When a 3D Grid includes it appear in the list: more subgroups, these



The selection of the grid to be loaded is done graphically. Press the Επιλογή γραφικά button.



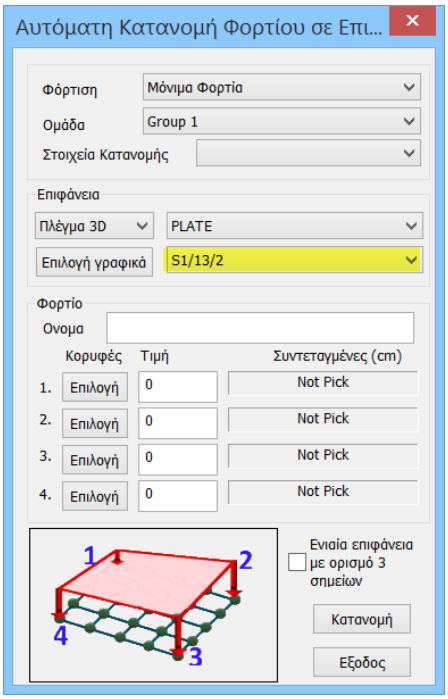
The dialog box is automatically closed and you are asked to point to the surface to which the load will be applied by left-clicking on a surface element.

The dialog box reopens with the graphically selected surface recognized

In the Load field give a characteristic name for the load. You are then prompted to specify how the loads are to be distributed on the selected surface.
The definition can be done graphically:

- With its 4 vertices and the corresponding values of the load.
- With any 3 points of which the first two define a line where one value of the load will be applied and the third point the altitude where the other value will be applied.

The points may not be coplanar and its outline may include lines, arcs and circles.

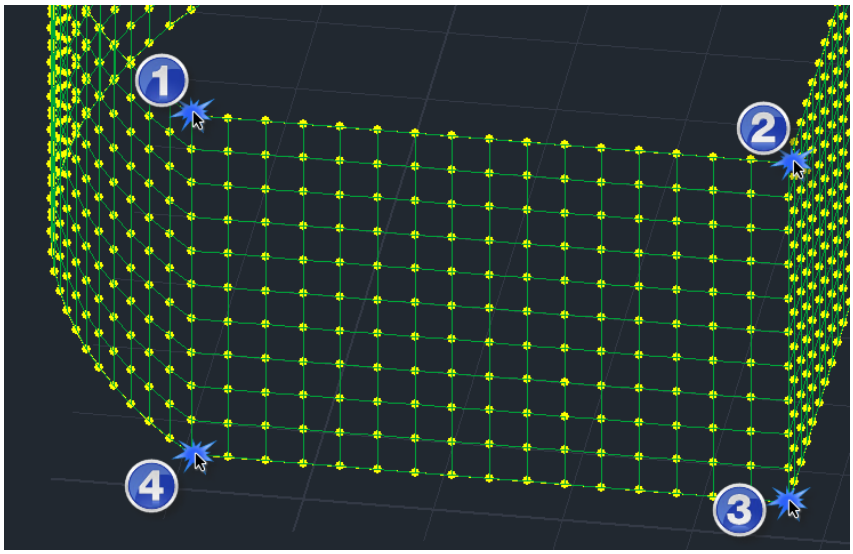


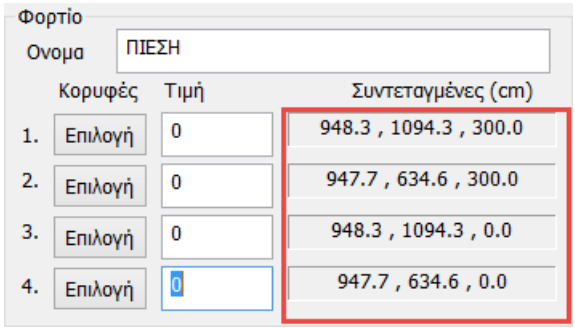
In detail in:

3.5.1.1 Flat surfaces:

EXAMPLE 1:

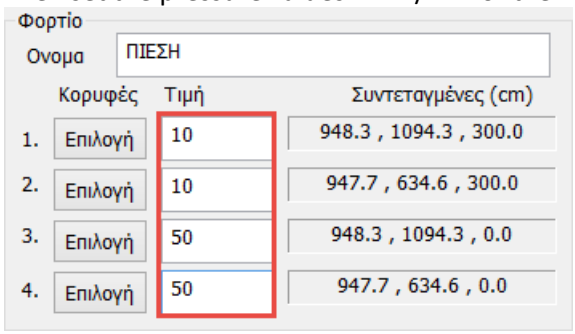
Set the 4 vertices that define it by pressing the keys in succession Επιλογή for each vertex, as described in the picture

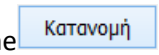


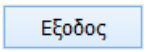


In this way the coordinates of the 4[s] points are automatically recognized and complete the list of coordinates.

Then set the pressure values in KN/m2 for the 4 points

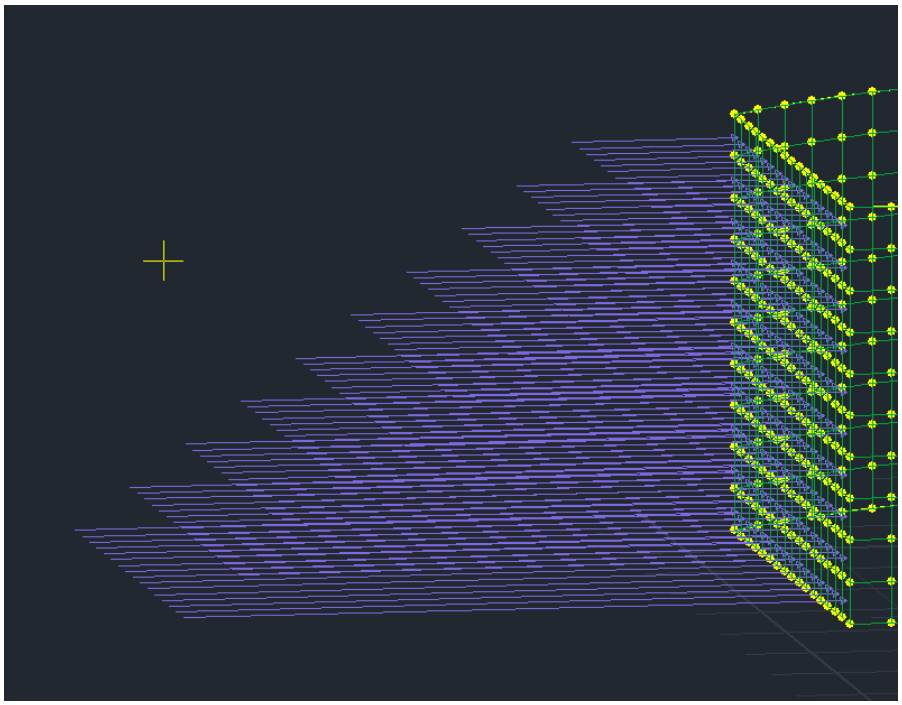


Finally press the 

and 

button.

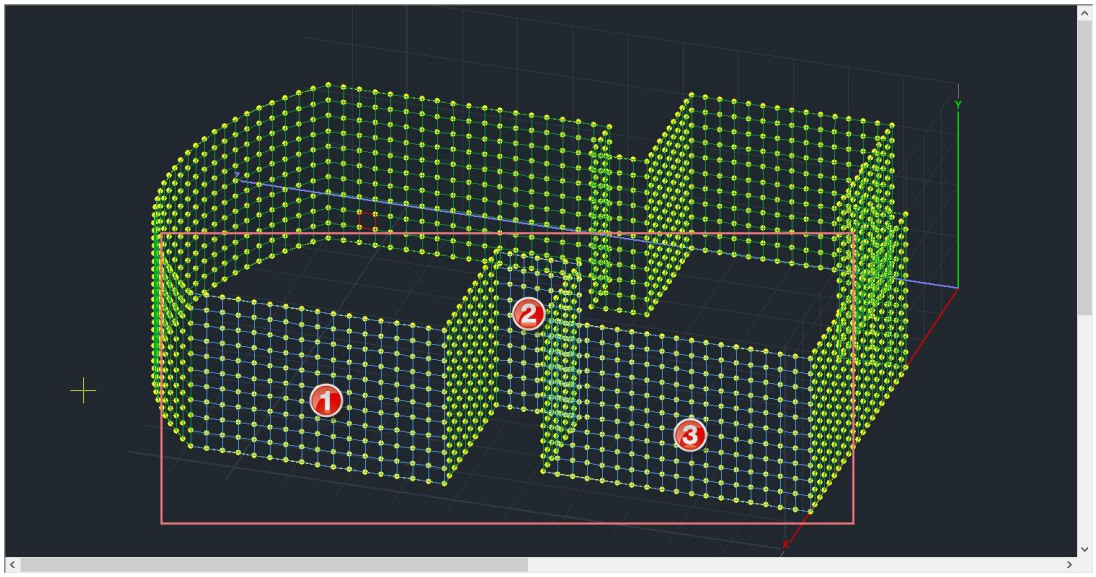
The distribution of loads on the selected surface is completed and the graphical representation is displayed on the surface mesh elements that simulate it.



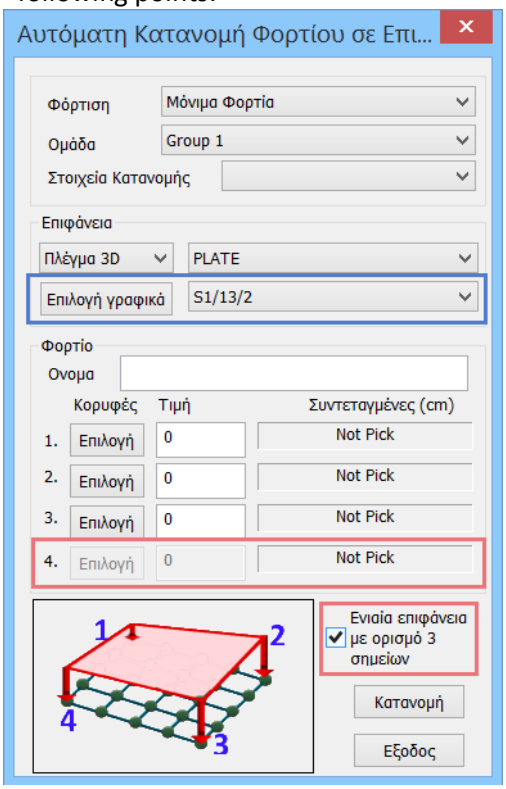
3.5.1.2 Successive surfaces:

EXAMPLE 2:

It is also possible to automatically distribute the pressure to successive surfaces.

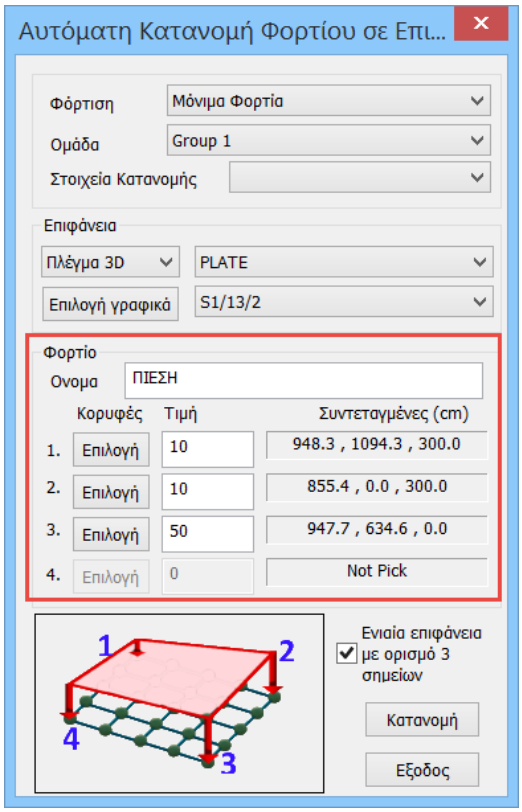


The procedure followed is very similar to the previous one and the differences can be found in the following points:



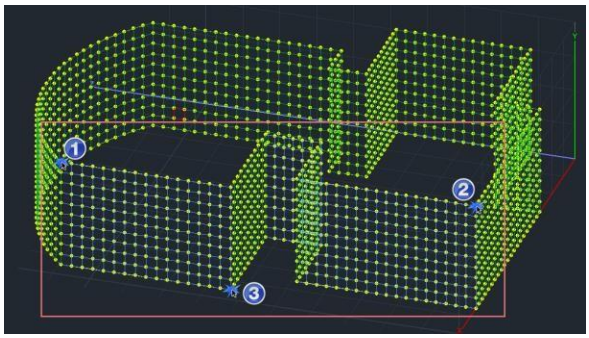
- With the Graphics option you show as before an element of one of the successive surfaces.

- Check the "Single surface with 3-point definition" option and the 4th vertex is automatically disabled.



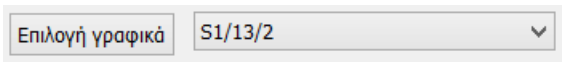
- Similarly to before, via the Επιλογή button you show the 3 vertices that define the single surface.

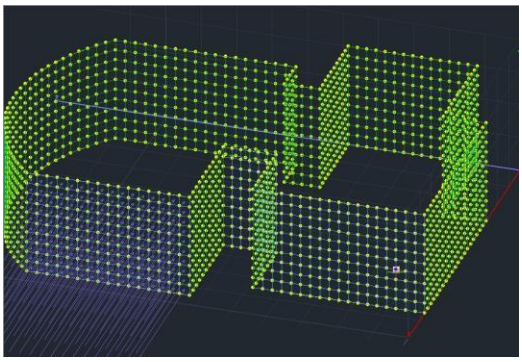
- Then set the pressure values in KN/m2 for these 3 points.



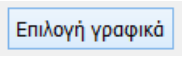
- Finally press the Κατανομή and Εξοδος button.

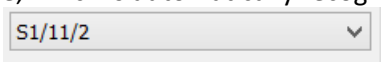
- The distribution of loads on the selected (with the Graphics option) surface is completed and the graphical representation is displayed on the elements of the surface mesh that simulates it.

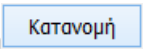


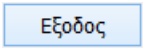


- In order to distribute the loads to the next remaining surfaces, select the command 

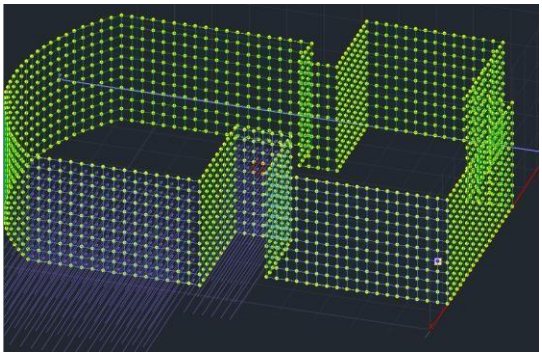
again and from the dialog box 

and show a surface element of the next surface, which is automatically recognized and displayed in the window 

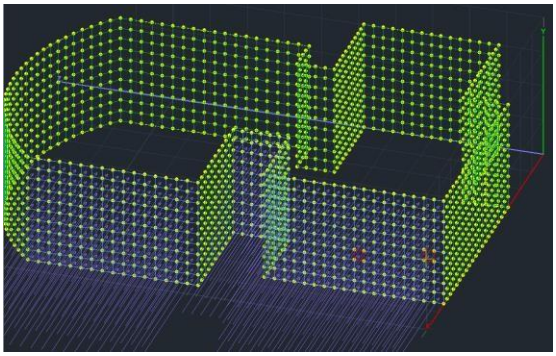
. Again 



.

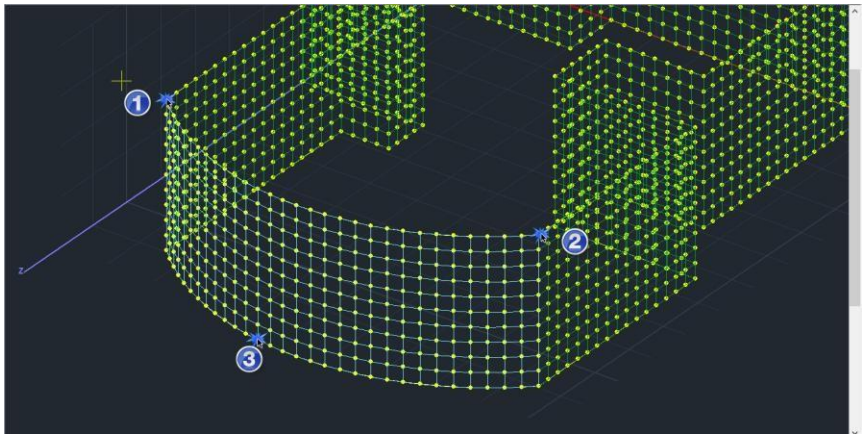


- Follow the same procedure for the third surface in the row.

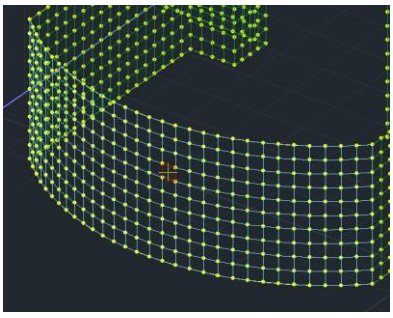


3.5.1.3 Curved surfaces:

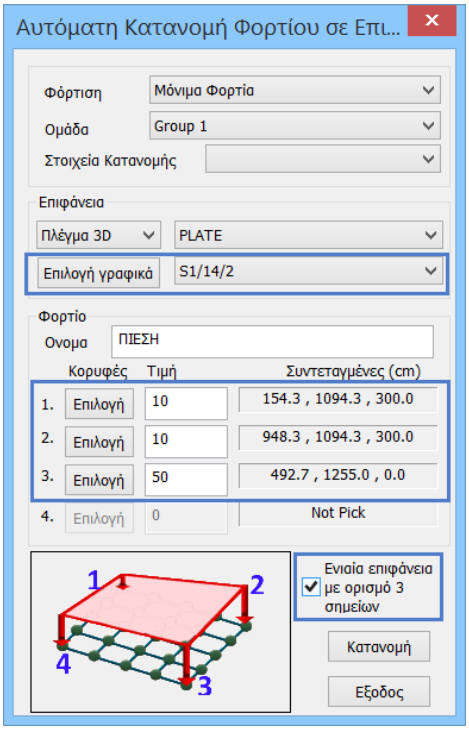
EXAMPLE 3:



Follow the same procedure:



Select graphics with left click.



Check the "Single surface with 3-point definition" option and the 4th vertex is automatically disabled.

Set the 3 vertices that define the single surface via the Επιλογή button.

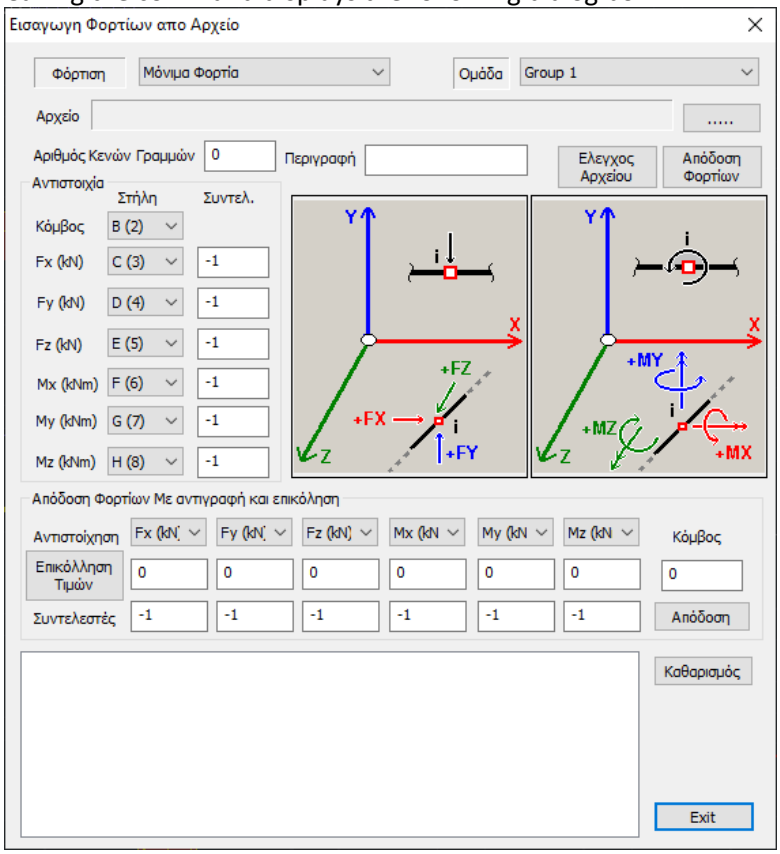
Set the pressure values in KN/m2 for these 3 points.

Κατανομή and . Εξοδος

3.5.2 Import loads from file

NEW command that now allows you to import into your vector epic loads from an Excel file that can be in the format *.csv, *.xls, *.xlsx.
This function is quite useful, especially in cases of foundation analysis where superstructure loads are applied as epicyclic loads at the beginning node of the columns leading to the foundation.
NOTE: Recall that the epicometric loads in SCADA Pro are applied in the global coordinate system.

Calling the command displays the following dialog box



There are two ways to import loads from the file you have: a. The automatic way
β. By copying and pasting

α. The automatic way

A REQUIRED condition in the automatic mode is: before importing the file to match the original nodes included in Excel with the nodes of the operator in SCADA Pro.

For example, the following is a prop reaction file from SAP2000.

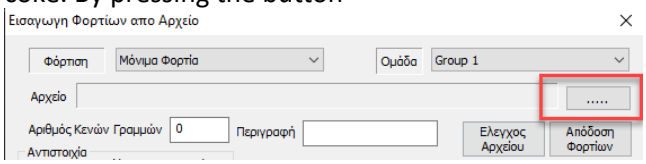
TABLE: Joint Reactions							
Joint	F1	F2	F3	M1	M2	M3
SAP	KN	KN	KN	KN-m	KN-m	KN-m
1	6,255	7,641	759,589	-10,5952	8,4675	-0,0124
2	-1,519	9,266	1088,696	-12,6444	-1,5622	-0,0124
3	1,286	10,464	895,447	-14,1497	2,0563	-0,0124
4	10,854	-3,341	998,819	3,5719	14,4527	-0,0124
5	6,444	-2,451	535,851	2,4243	8,7985	-0,0124
6	-1,817	-3,118	749,784	3,3327	-1,8589	-0,0124
7	13,02	-9,788	1033,945	11,9775	17,2825	-0,0124
8	-16,535	13,395	906,307	-17,8875	-20,8914	-0,0124
9	-17,992	22,366	1025,771	-29,4264	-22,7368	-0,0124
10	20,9	-19,025	1022,569	23,9284	27,4834	-0,0124
11	-7,468	0,665	588,194	1,7425	-6,9445	-0,0086
12	1,565	-7,644	569,862	5,7798	-1,9269	-0,0086
13	-14,994	-18,432	3454,846	-15,8908	310,1293	-0,0261

The first column is the numbering of the nodes of the operator in SAP2000. So before you import the file, you need to create a new column, for example immediately after the nodes column, and there next to each SAP2000 node you will write the corresponding SCADA Pro node.

TABLE: Jo TABLE: Joint Reactions							
Joint	Joint	F1	F2	F3	M1	M2	M3
SAP	SACDA	KN	KN	KN	KN-m	KN-m	KN-m
1	5	6,255	7,641	759,589	-10,5952	8,4675	-0,0124
2	6	-1,519	9,266	1088,696	-12,6444	-1,5622	-0,0124
3	7	1,286	10,464	895,447	-14,1497	2,0563	-0,0124
4	8	10,854	-3,341	998,819	3,5719	14,4527	-0,0124
5	4	6,444	-2,451	535,851	2,4243	8,7985	-0,0124
6	3	-1,817	-3,118	749,784	3,3327	-1,8589	-0,0124
7	9	13,02	-9,788	1033,945	11,9775	17,2825	-0,0124
8	2	-16,535	13,395	906,307	-17,8875	-20,8914	-0,0124
9	1	-17,992	22,366	1025,771	-29,4264	-22,7368	-0,0124
10	10	20,9	-19,025	1022,569	23,9284	27,4834	-0,0124
11	11	-7,468	0,665	588,194	1,7425	-6,9445	-0,0086
12	13	1,565	-7,644	569,862	5,7798	-1,9269	-0,0086
13	12	-14,994	-18,432	3454,846	-15,8908	310,1293	-0,0261

In the above table, node 1 of SAP2000 corresponds to node 5 of the vector in SCADA Pro, the 2 with the 6

coke. By pressing the button



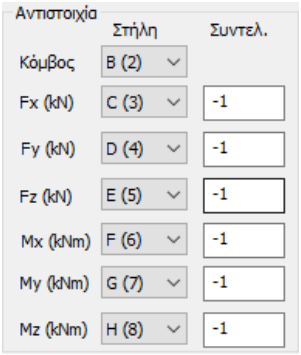
enter the file.

Then, from the "Load" and "Group" options, you select the load and the group to which the loads to be imported will belong. In the "Description" field you optionally write a description for this load.

ATTENTION!

Excel file loads can only be assigned to one load. If you have loads that you want to import into multiple loads, you must have a different Excel file for each load.

Then and in the section below



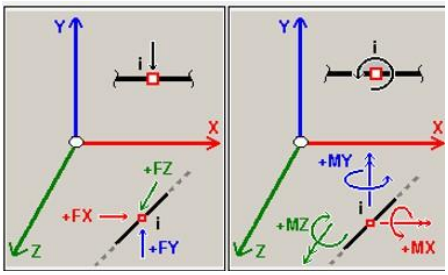
indicate from which column of Excel the nodes, forces and moments will be read in order to match them with those of SCADA Pro.

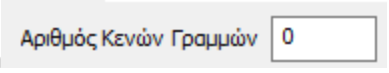
More specifically:

- In the "**Node**" option you indicate the column of Excel where the nodes (Attention!) of SCADA Pro that have been mapped in the previous step with those of SAP2000 are located. It is the additional column that you previously created in Excel. Correspondingly for each size you declare the column where the sizes from SAP2000 are located.

- In the column "**Co.**" you define a numerical coefficient by which the load magnitudes will be multiplied.

NOTE: Important here is the sign, in order to transfer the loads to the nodes with the correct times based on the convention of epicomponent forces and moments on the universal axes in SCADA Pro that you see in the following picture:



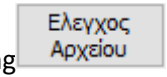
- In the "**Number of Blank Lines**option 

you enter the number of first rows in Excel where they are rows that do not include the intensive quantities.

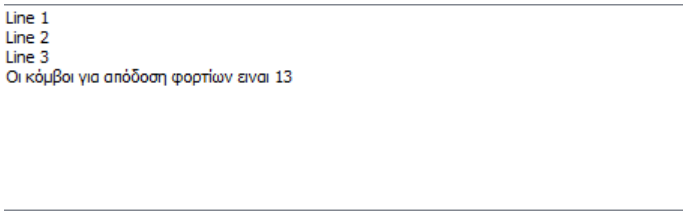
For example, for the following Excel

	A	B	C	D	E	F	G	H
1	TABLE: Jo TABLE: Joint Reactions							
2	Joint	Joint	F1	F2	F3	M1	M2	M3
3	SAP	SACDA	KN	KN	KN	KN-m	KN-m	KN-m
4	1	5	6,255	7,641	759,589	-10,5952	8,4675	-0,0124
5	2	6	-1,519	9,266	1088,696	-12,6444	-1,5622	-0,0124
6	3	7	1,286	10,464	895,447	-14,1497	2,0563	-0,0124
7	4	8	10,854	-3,341	998,819	3,5719	14,4527	-0,0124
8	5	4	6,444	-2,451	535,851	2,4243	8,7985	-0,0124
9	6	3	-1,817	-3,118	749,784	3,3327	-1,8589	-0,0124
10	7	9	13,02	-9,788	1033,945	11,9775	17,2825	-0,0124
11	8	2	-16,535	13,395	906,307	-17,8875	-20,8914	-0,0124
12	9	1	-17,992	22,366	1025,771	-29,4264	-22,7368	-0,0124
13	10	10	20,9	-19,025	1022,569	23,9284	27,4834	-0,0124
14	11	11	-7,468	0,665	588,194	1,7425	-6,9445	-0,0086
15	12	13	1,565	-7,644	569,862	5,7798	-1,9269	-0,0086
16	13	12	-14,994	-18,432	3454,846	-15,8908	310,1293	-0,0261

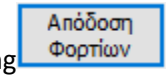
the number of first lines containing headings and not sizes is 3. The first line of intensive magnitudes is the fourth line. So the number of blank lines is 3.

- By selecting 

the program checks the Excel data and in case of errors they are displayed in the following window



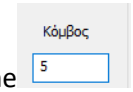
For example, the above messages show 3 lines that are not recognized by the program (the parameter for the number of empty lines was probably set incorrectly) and that 13 nodes were found to which epicomponent loads will be assigned.

- Finally, by pressing 

the program automatically assigns the loads to the nodes and a confirmation message appears

β. By copying and pasting

This is how to copy the loads from each row Excel and paste them here in the dialog box. This process is done for each node separately.

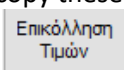
In the 

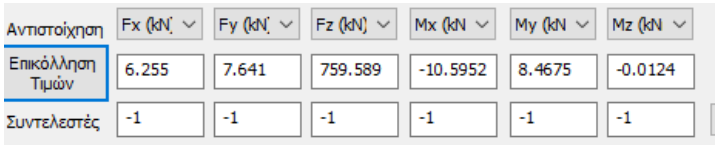
field enter the number of the node in SCADA Pro to which you want to assign the loads. This is the node number based on the SCADA Pro and SAP2000 node mapping mentioned earlier.

Then copy the line with the 6 intensive sizes - loads.

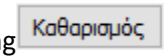
For example in the following picture

TABLE: Jo TABLE: Joint Reactions							
Joint	Joint	F1	F2	F3	M1	M2	M3
SAP	SACDA	KN	KN	KN	KN-m	KN-m	KN-m
1	5	6,255	7,641	759,589	-10,5952	8,4675	-0,0124
2	6	-1,519	9,266	1088,696	-12,6444	-1,5622	-0,0124
3	7	1,286	10,464	895,447	-14,1497	2,0563	-0,0124
4	8	10,854	-3,341	998,819	3,5719	14,4527	-0,0124
5	4	6,444	-2,451	535,851	2,4243	8,7985	-0,0124
6	3	-1,817	-3,118	749,784	3,3327	-1,8589	-0,0124
7	9	13,02	-9,788	1033,945	11,9775	17,2825	-0,0124
8	2	-16,535	13,395	906,307	-17,8875	-20,8914	-0,0124
9	1	-17,992	22,366	1025,771	-29,4264	-22,7368	-0,0124
10	10	20,9	-19,025	1022,569	23,9284	27,4834	-0,0124
11	11	-7,468	0,665	588,194	1,7425	-6,9445	-0,0086
12	13	1,565	-7,644	569,862	5,7798	-1,9269	-0,0086
13	12	-14,994	-18,432	3454,846	-15,8908	310,1293	-0,0261

the number of the node in SCADA Pro to which the loads will be transferred is 5. So copy these values from Excel and then, after writing node 5 in the corresponding field, press the 



The values were transferred to the corresponding fields. It goes without saying that here too you have to assign the loads (Forces and moments per direction) from the menus above the value fields. In case of different units, you also set the appropriate coefficient.

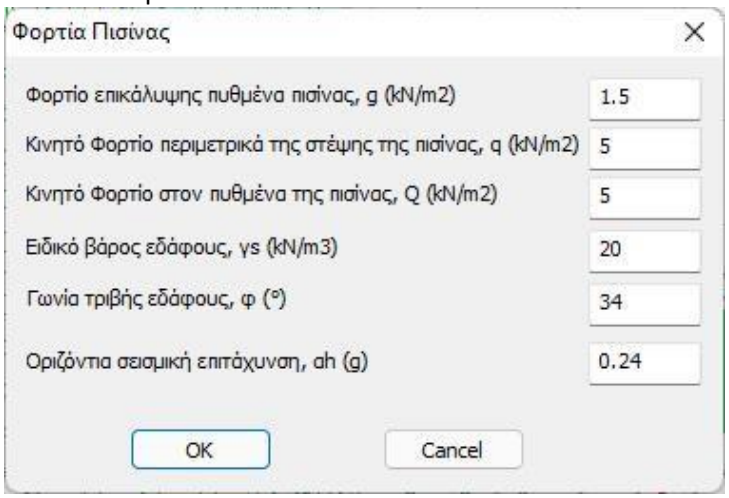
Finally, selecting 

deletes all messages that appear in the message area (control and confirmation).

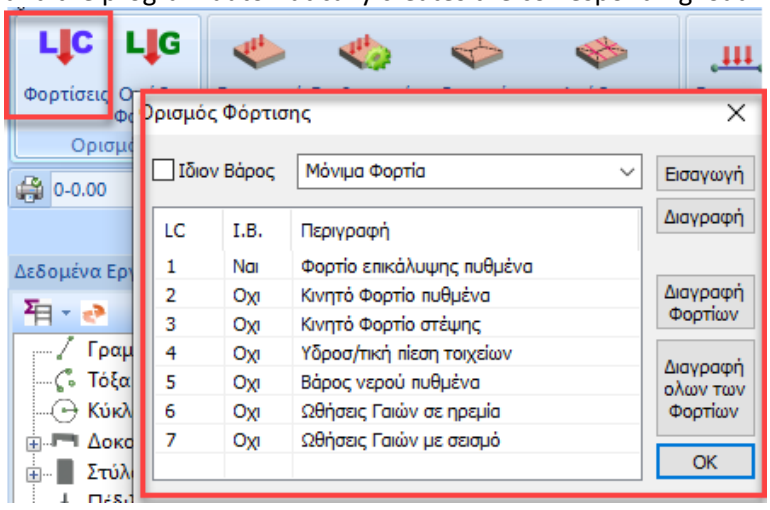
3.5.3 Pool loads

In the new version of the program there is now the possibility of automatic creation and input of pool loads.
ATTENTION: For the correct direction of the loads, you must make sure that the local axes of all the pool surfaces are in the correct direction.

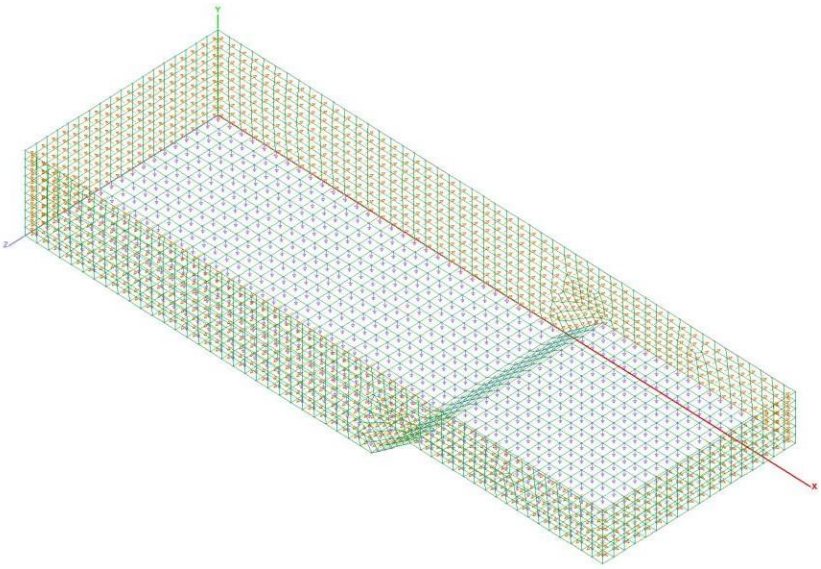
You set the parameters of the loads



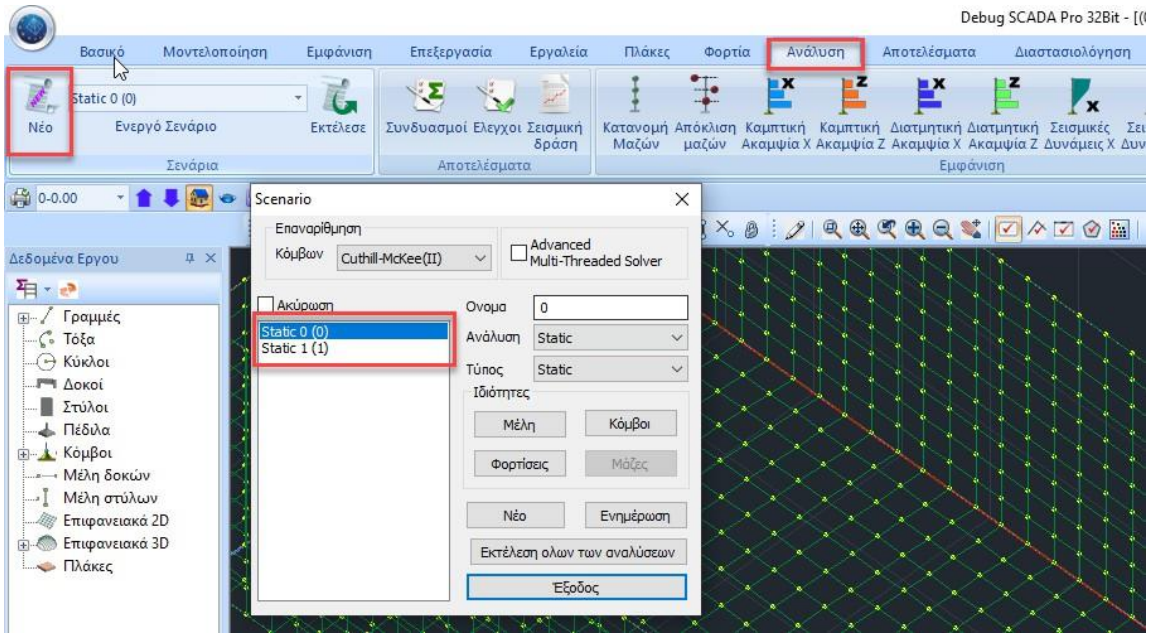
and the program automatically creates the corresponding loadings



and applies the loads to the pool.

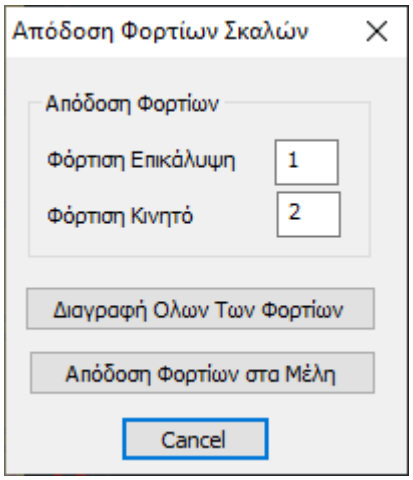


The 7 Loadings that relate to the in-ground pool are automatically generated, while the corresponding static scenarios of the analysis that include them are also automatically generated.

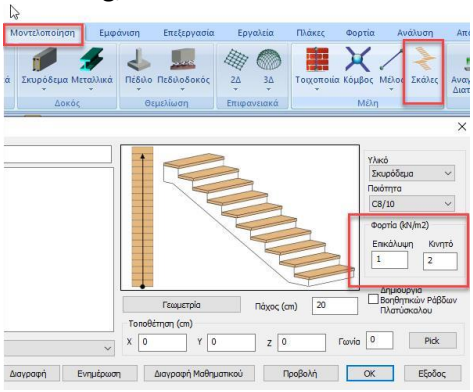


In the first scenario *Static 0 (0)* the first 3 loads are included and in the second scenario *Static 1 (1)* the other 4**.**

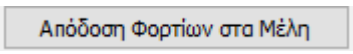
3.5.4 Scale loads

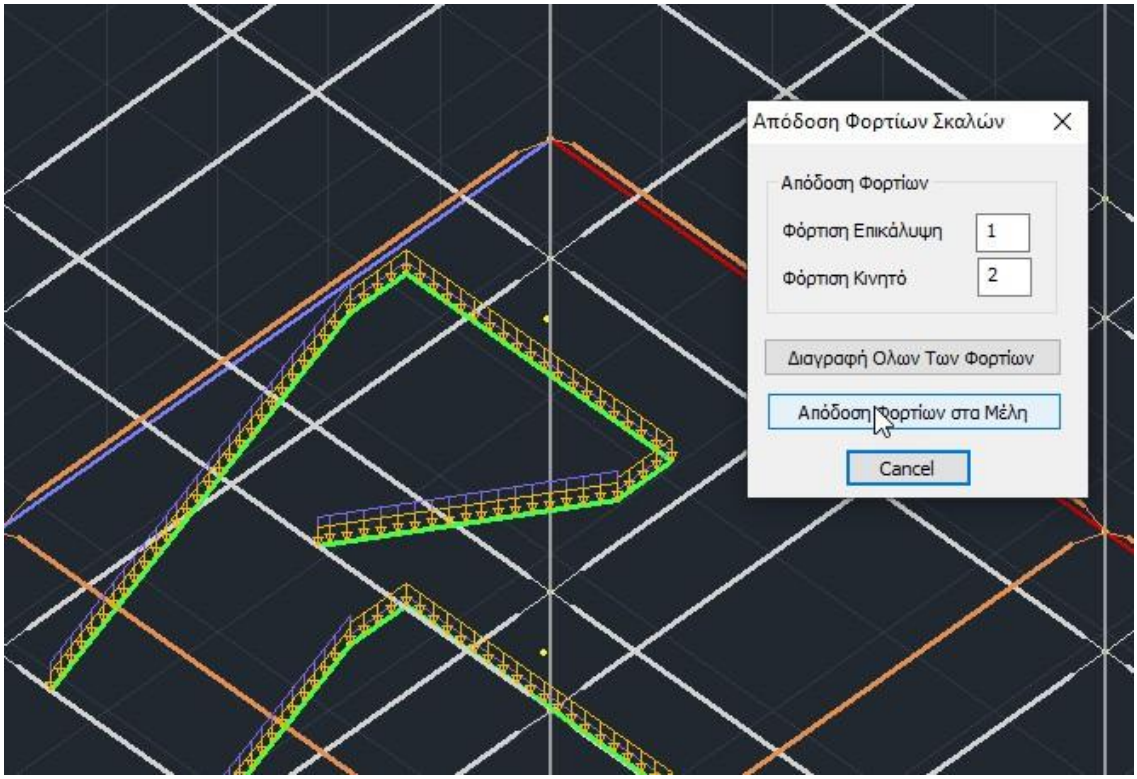


Selecting the command opens the dialog box, where you set the number of the corresponding load (LC1: Permanent, LC2: Mobile), for the performance of the loads that you have defined when creating the stairs in the Modeling,

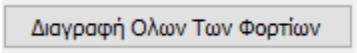


to the respective Members with the mandate

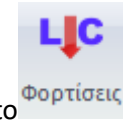




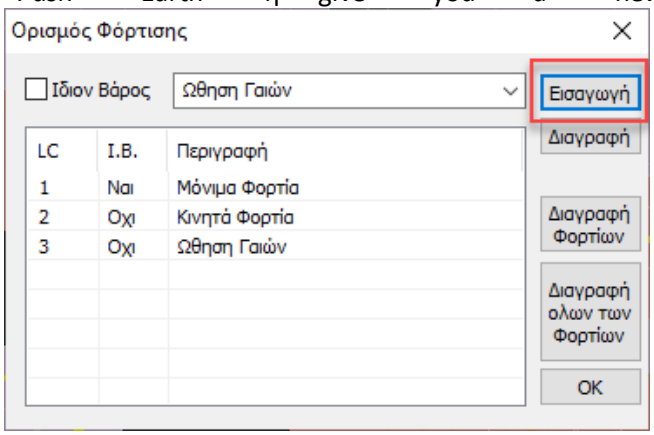
You can also modify the values of the loads or delete them by selecting



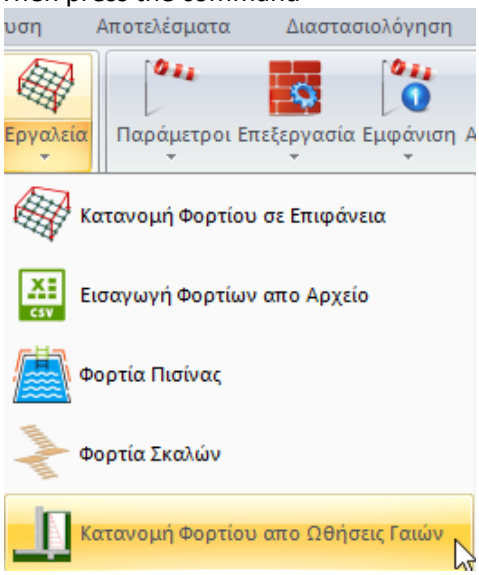
3.5.5 Load Distribution from Earth Thrusts

You set a new charge by going to 

, open the scroll down menu, select "Push Earth" ή give you a new name and then import.

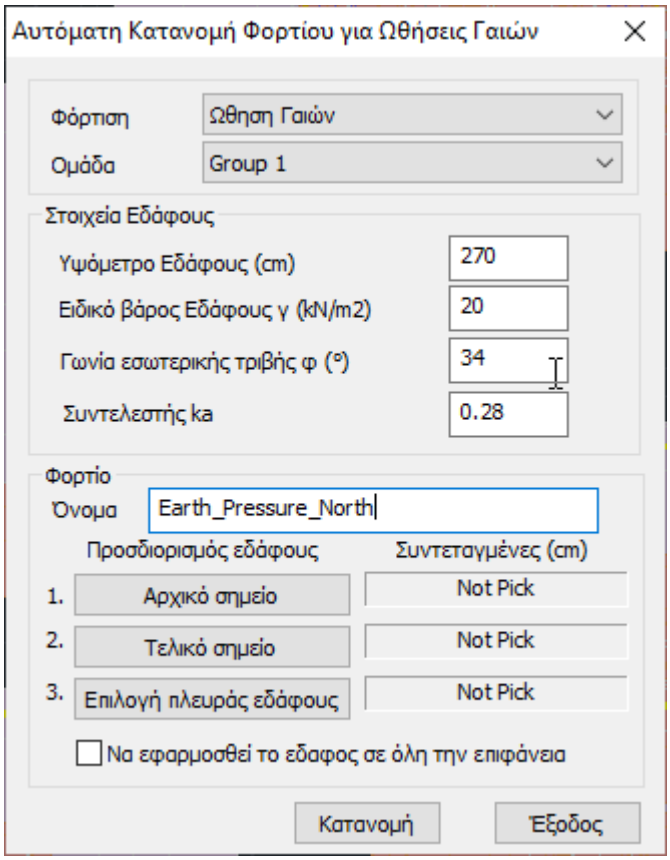


Then press the command



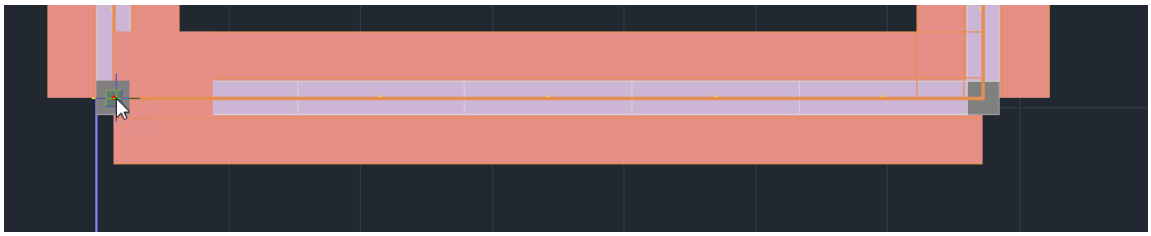
The automatic load sharing mask opens for earth thrusts. There

1. Fill in the ground elevation (in relation to level 0)
2. Modify specific gravity
3. You modify the angle of internal friction and the coefficient ka
4. You give a name to the shipment.



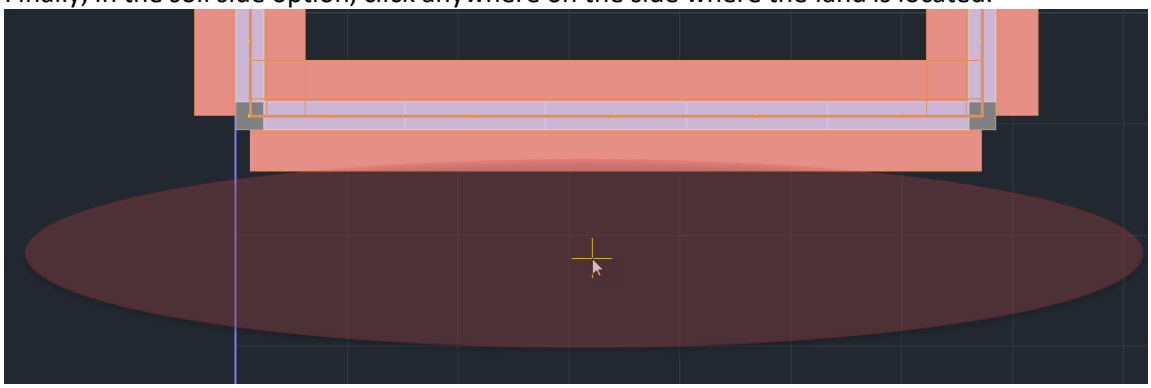
You are then asked to select the points in the model where the above will be applied to make the distribution.

By pressing the "start point" button make sure you are in plan view at level 0 and then select the one node that defines the start

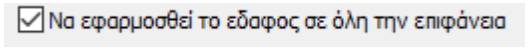


Successively click on "end point" and select the other end node

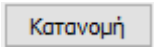
Finally, in the soil side option, click anywhere on the side where the land is located.



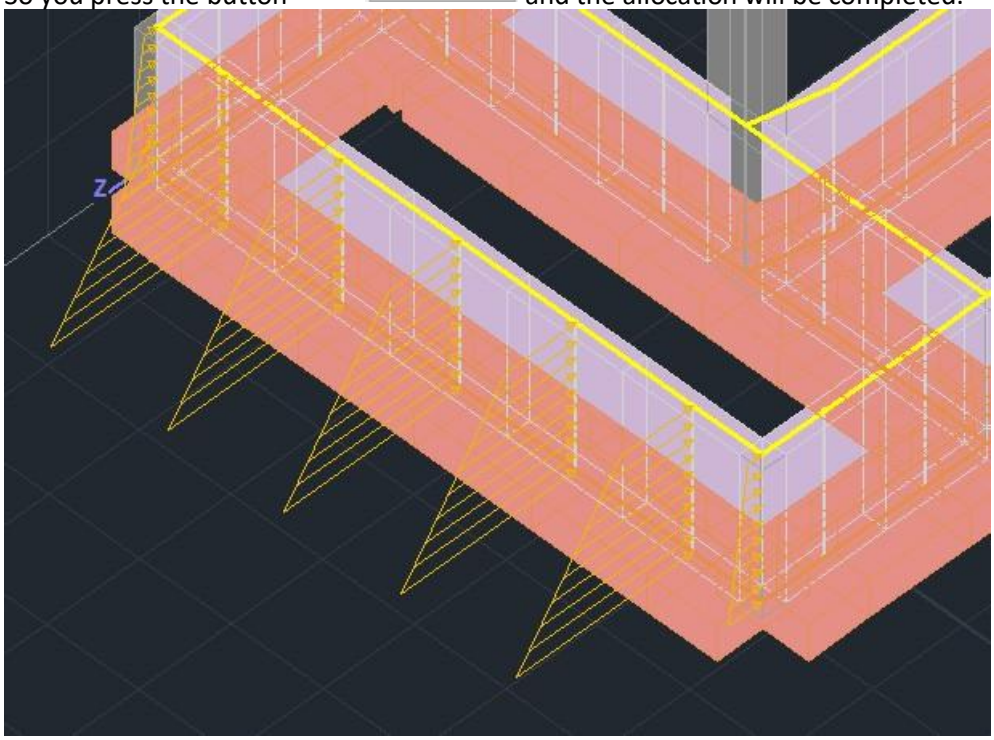
Notes:



selecting this checkbox will have the following effect result to allocate ground loads with the above data to any element that is conjugate to the points defined.

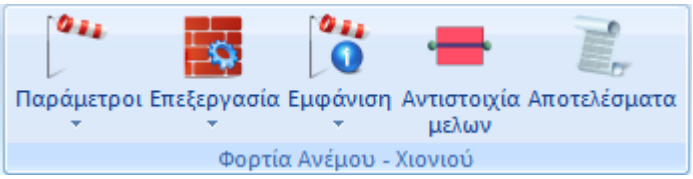
So you press the button 

and the allocation will be completed.



You can continue for the rest of the building without leaving the mask and when you are done press exit.

4. Wind and Snow loads



The "Wind and Snow Loads" command group contains the tools for the automatic calculation of wind and snow loads, and their distribution to members, based on various regulations including Eurocode 1 (including the Greek Appendix) and the Saudi Arabian regulation (SBC 301) It also includes the appendices of Eurocode 1 of Italy, Germany, Poland and the Italian regulation NTC18.

This is an excellent tool that includes:

- Automatic calculation of the characteristic values of the snow load on the ground and roofs based on the above regulations and for all types of roofs: flat, single, double, double, four-roofed, vaulted, with roof adjacent to a taller building, drifting on overhangs and obstacles.
- Automatic calculation of roof shape coefficients.
- 2D and 3D visualization of snow load distribution.
- Automatic calculation of the basic wind speed.
- Automatic calculation of average wind speed VM (z) at height z (according to ground roughness and topography)
- Categories and soil parameters
- Wind turbulence
- Maximum speed
- Distribution of wind pressure on surfaces
- Wind forces
- Pressure coefficients for buildings (vertical walls or roofs)

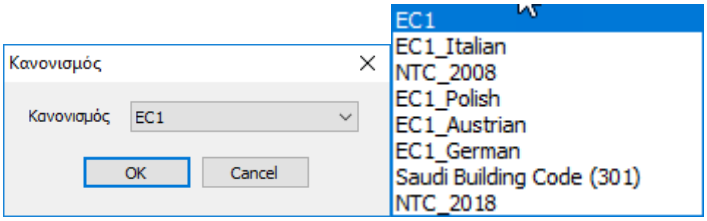
The procedure for calculating the wind and snow loads and their distribution to the members is included in the 5 team orders:

1. **Parameters**: Regulation Selection, General Wind-Snow Parameters
2. **Treatment**: wall-roofing
3. **Appearance**: wind-snow
4. **Membership Match**
5. **Results**

4.1 Parameters

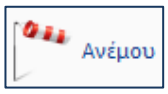
4.1.1 Regulation

In the dialog box that appears

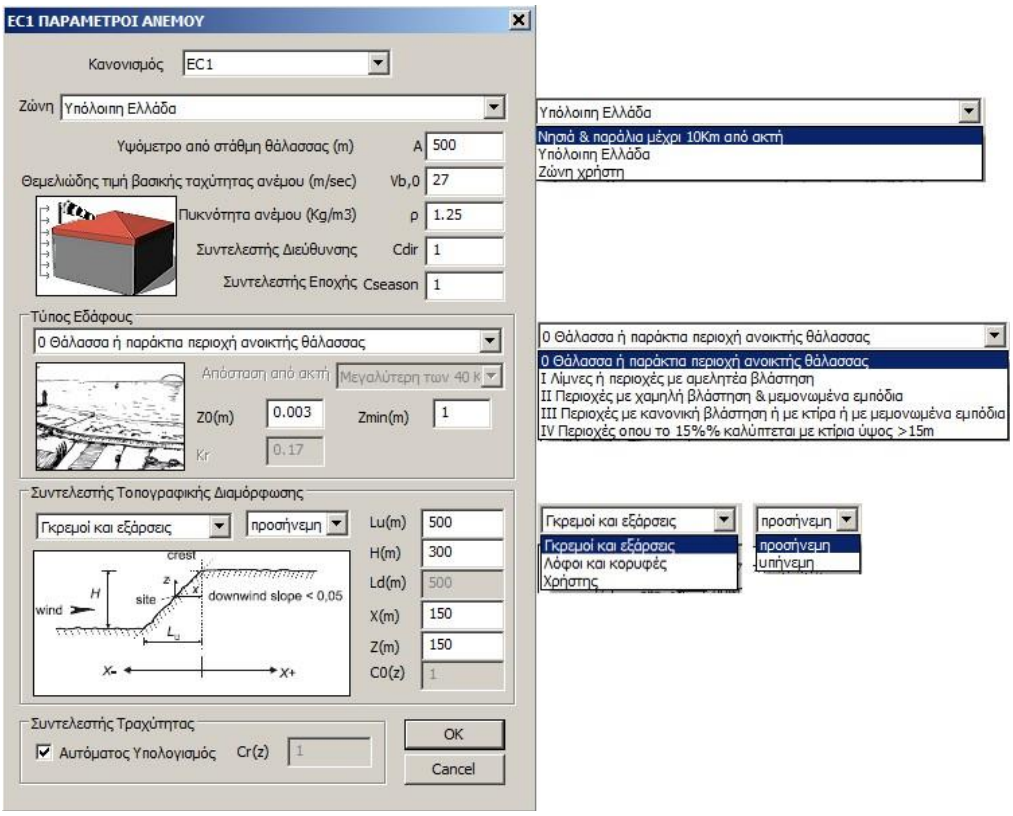


select the regulation on the basis of which the wind and snow loads will be calculated.

4.1.2 Wind parameters



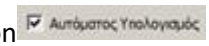
If you select **Eurocode 1** as regulation, you set the wind parameters in the following dialogue box, according to Eurocode 1 and the corresponding Greek Appendix (EC1):



Select from the lists: the "Regulation" report and the "Zone" report and the corresponding fields are automatically updated.

In the "Type of Soil" field, select the type, category and distance from the coast from the list.

In the "Topographic Modulation Factor" field, select the topography and wind direction from the list. The other fields are filled in automatically according to the previous options.

In the "Velocity Coefficient" field turn on 

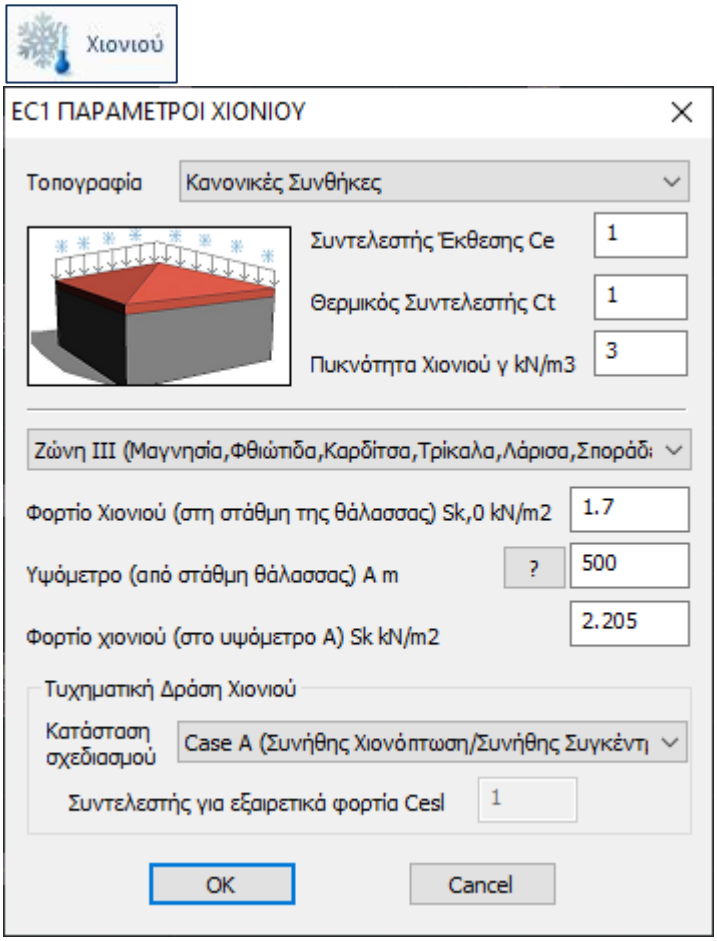
and the program automatically calculates Cr(z) or turn off and enter a value . 

Select "OK" to save the parameters.

⚠ *The user can modify the values automatically entered by program by typing his own values in the fields.*

4.1.3 Snow parameters

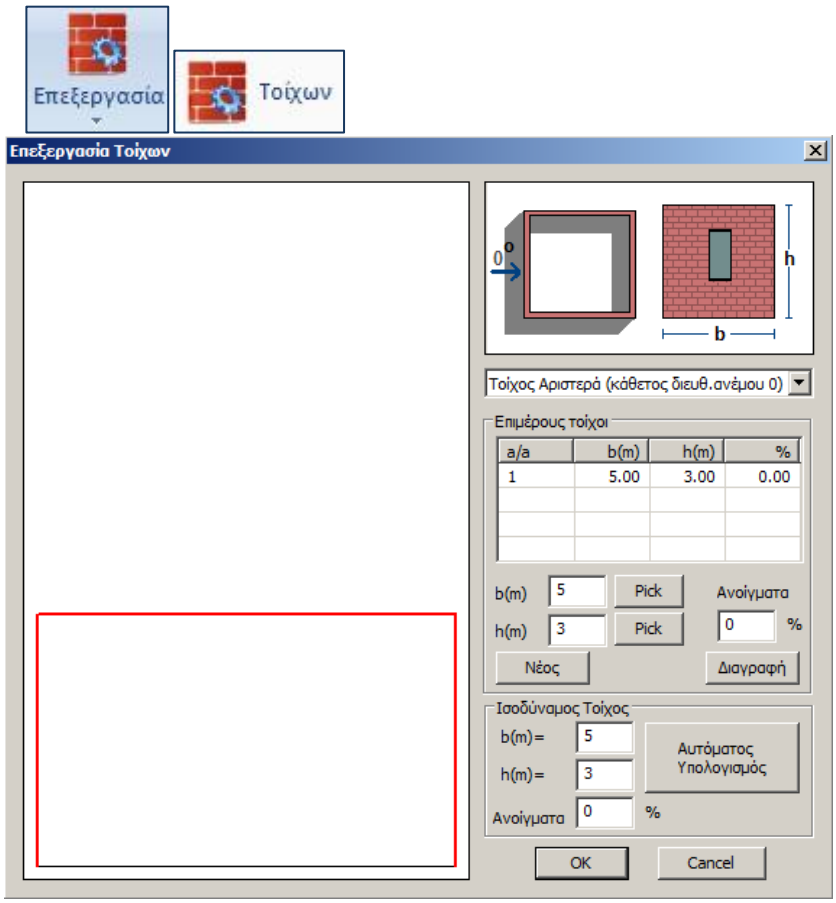
Set the snow parameters, according to **Eurocode 1** and the Greek Appendix (EC1), by filling in the dialogue box:



Select from the lists: the reference "Legislation", "Topography" and "Zone" and the corresponding fields are automatically updated.
In the "Random Snow Action" field, select the condition. "OK" to save the parameters.

4.2 Edit

4.2.1 Wall Treatment



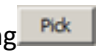
Taking advantage of the advantage offered by "Standard Structures", the user can save a lot of time and work since all the geometric characteristics of the walls are automatically filled in by the program.

4.2.1.1 Without using the "Standard Constructions"

Select from the list the wall according to the wind direction.

⚠ *Remember that the direction of the walls is clockwise.*

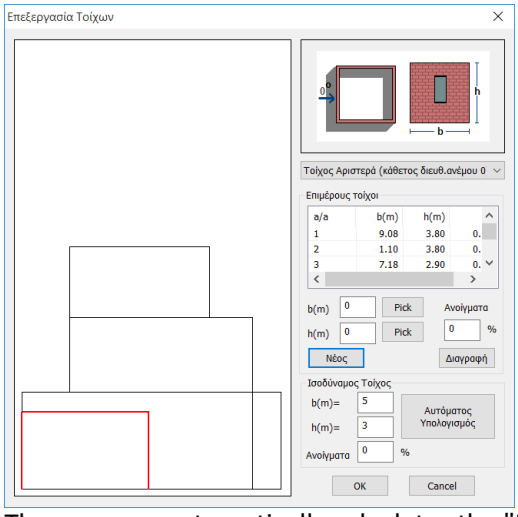
Select the Pick button next to b(m) and left-click on the desktop to point to the start and end points that define the length of the selected wall.

Repeat by selecting  next to  and left-click on the desktop to point to the starting and ending points that define the height of the selected wall.

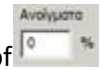
OBSERVATIONS:

⚠ The height of the wall below is <u>always</u> defined starting from <u>level 0</u> even if the metal structure starts from a higher level.

⚠ If the face consists of more walls on one or more levels, press the "New" button and repeat the previous procedure until you have defined the whole face.

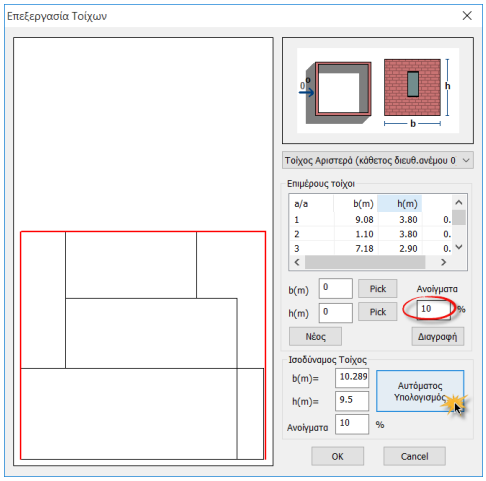


In this way you complete the label with the geometric characteristics of the "Individual walls".

Finally, enter the percentage of  openings for each direction and select  each time.

The program automatically calculates the "Equivalent Wall".

⚠ *The entire face must be bounded by the red rectangle.*

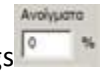


"OK" to save the parameters.
Repeat for all four directions of the walls.

4.2.1.2 Using the "Standard Constructions"

Select from the list the wall according to the wind direction.

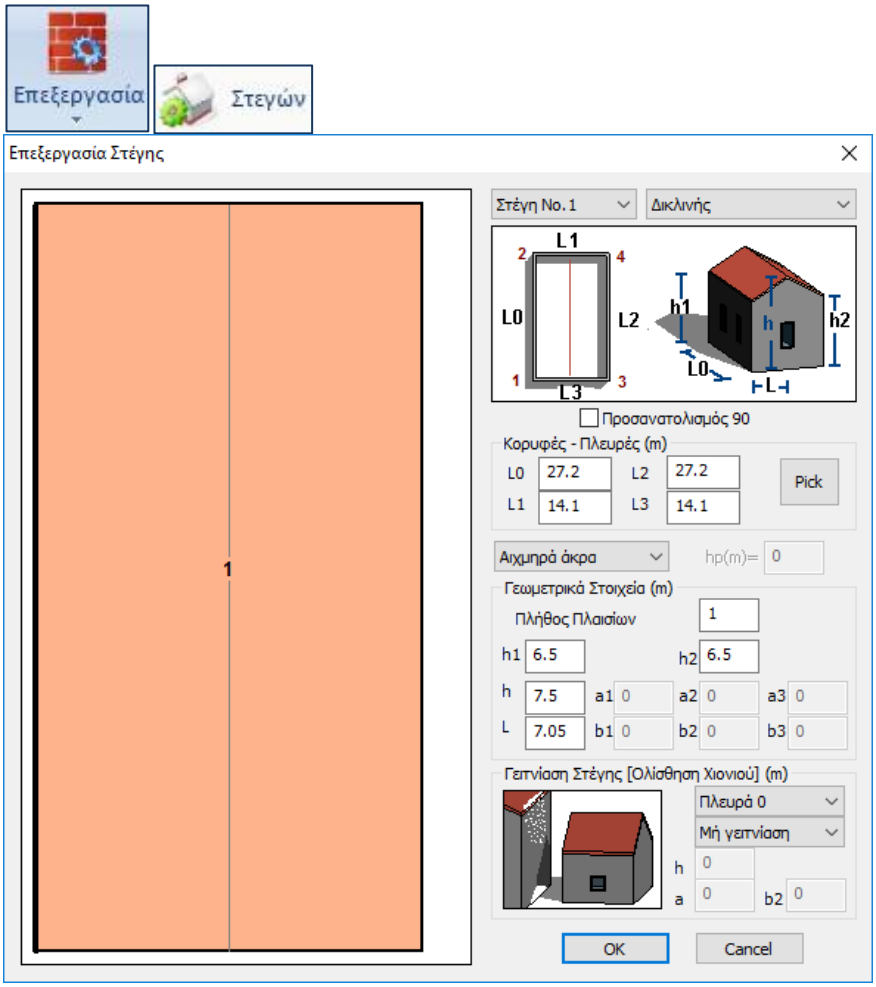
⚠ *Remember that the direction of the walls is clockwise.*

The tab with the geometric characteristics of the "Individual walls" is automatically filled in by the program. The user only has to enter the percentage of openings 

for each direction and select 

each time.
The program automatically calculates the "Equivalent Wall".
"OK" to save the parameters.
Repeat for all four directions of the walls.

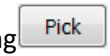
4.2.2 Roof treatment



4.2.2.1 Without using the "Standard Constructions"

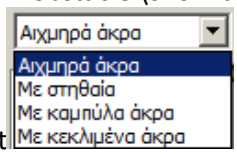
Select from the lists the number and shape of the roof.

⚠ *Remember that the direction of the roof is set clockwise.*

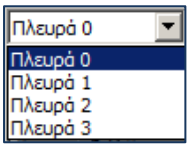
define the type of roof, its orientation and the dimensions Lo,L1,L2,L3, by pressing 

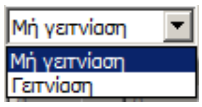
and selecting each time with the mouse the 4 edges of the roof.
The label with the geometric characteristics "Tops-Sides" is automatically filled in by the program.

NOTES:

If your roof has an obstacle (snow accumulation point) select the type of obstacle from the corresponding list 

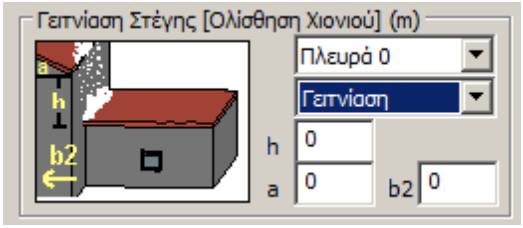
and enter its height in m.

If the structure in question is adjacent to another taller one, in the field "Roof adjacency" select the adjacent side "Kneeling". 

and from the list 

the

The field "Roof alignment" is modified accordingly to enter the necessary geometric characteristics.



"OK" to save the parameters.

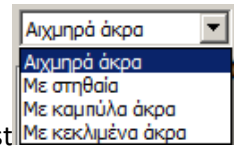
Repeat the process for all four directions of the roof.

4.2.2.2 Using the "Standard Constructions"

Select from the lists the number and shape of the roof.

⚠ *Remember that the direction of the roof is set clockwise.*

The label with the geometric characteristics "Tops-Sides" is automatically filled in by the program.

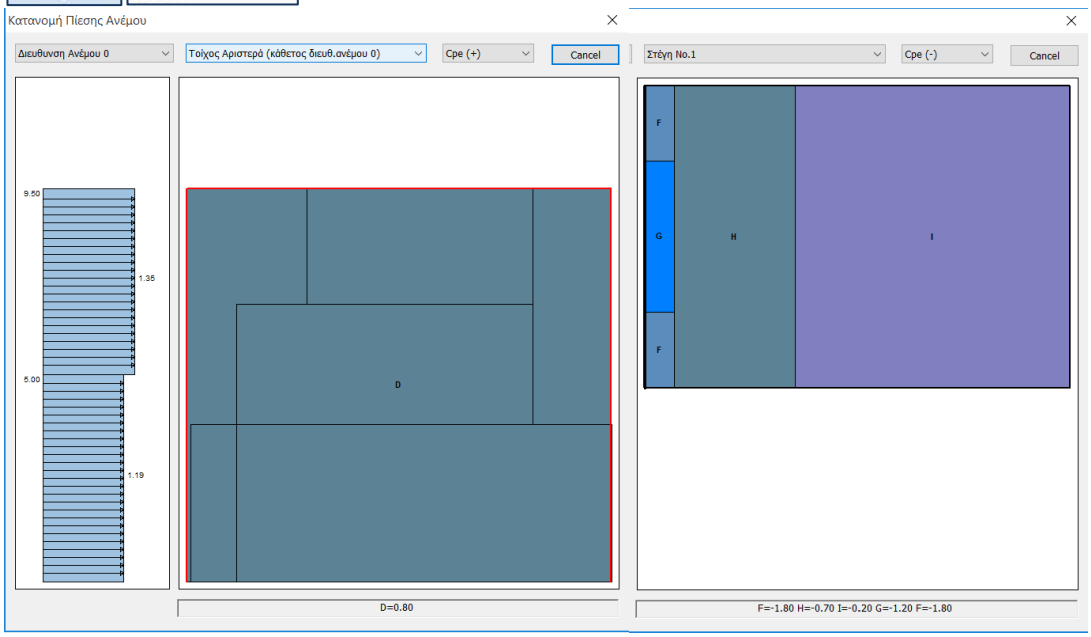


The user only has to select from the list , enter the height of the obstacle in m and possibly define the characteristics of the neighbourhood as before.
"OK" to save the parameters.

4.3 Show

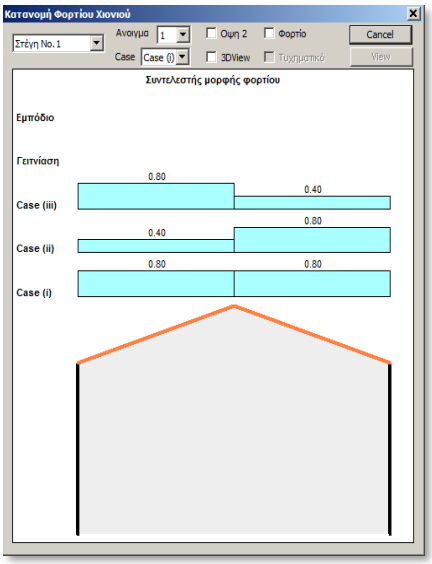
4.3.1 Wind Show

to see the wind distribution on the walls and roofs of the structure.
In the dialog box, from the first list on the top left, select the address of the wind, from the second the wall or roof and from the third the pressure in its direction.
The distribution is automatically displayed in colours. Zones with different pressure are defined by a different colour.

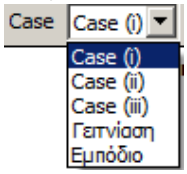


4.3.2 Snow Show

to see the distribution of snow on the roofs of the structure.

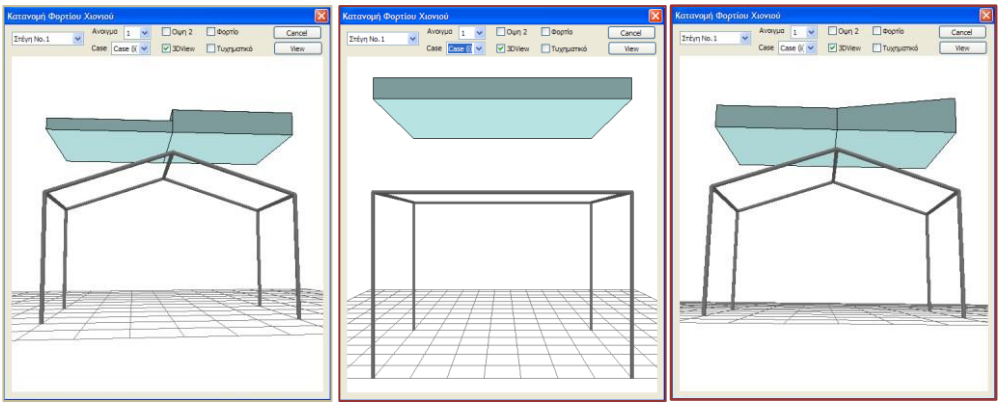


In the dialog box, select from the lists above left the number of the "roof", the "opening" meaning the number of the frame, in case you have more than one, and the "Case"

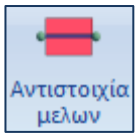


for snow load distribution.

Also enable the checkbox next to the "Load" to see the load values and side "3DView" to get the snow distribution in the illustration below.



4.4 Matching members



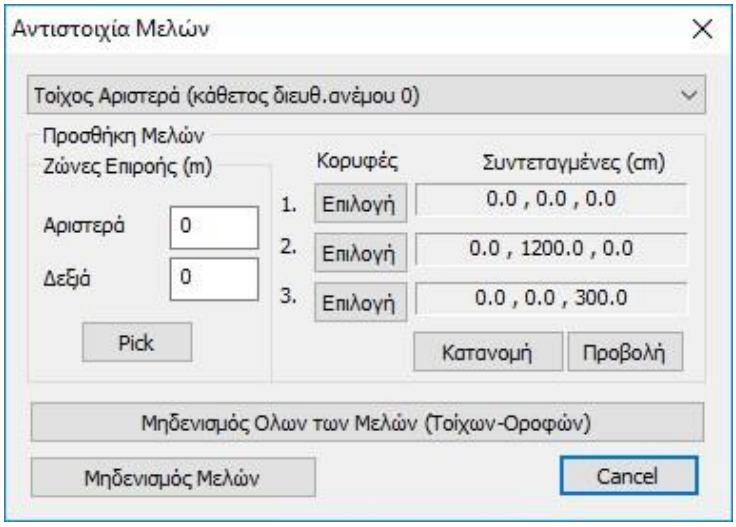
to assign the calculated loads to the corresponding members, through the zones of influence.
Select the command and in the dialog box you select one by one the walls and/or roofs for the distribution.

In SCADA Pro, the automatic calculation of the influence surfaces for the linear members was completed and integrated in order to distribute the wind and snow loads.

OBSERVATION:

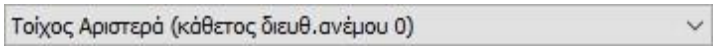
⚠ *Recall that until now automatic allocation was only done for constructions coming from the standard ones. It is now possible to perform this allocation on any surface defined by the designer.*

Let's take a closer look at **Manual**, **Semi-Automatic** and **Automatic**:
Selecting the command opens the following dialog box



In the part concerning the old definition of influence surfaces nothing has changed as well as the function of the "Pick" button where it hides the dialog box and displays the existing influence surfaces, has remained the same.

However, a part has been added on the right concerning the definition of the surface with three points. The definition of the surface is always done on the specific wall that is active in the window

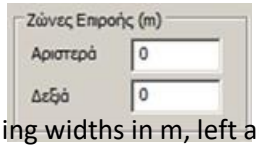


*It is advisable before starting either **manual** or **semi-automatic**, to reset everything that exists by pressing the "Reset Members" button.*

ATTENTION:

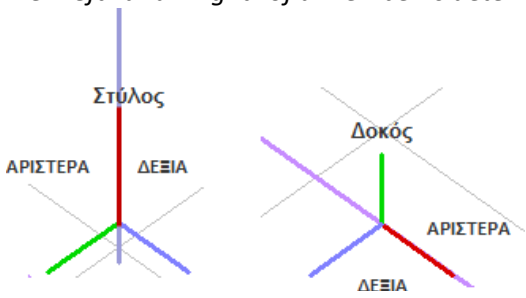
⚠ *In the **automatic** process that comes from the Standard Constructions DO NOT press the "Zero Members" button, because the automatic distribution of the loads to the members will be deleted!!!*

4.4.1 Manual Process - Without using the "Standard Constructions"

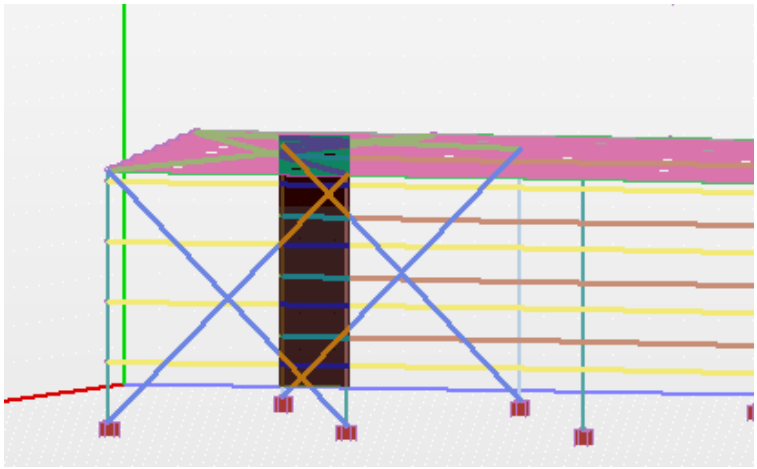
In the left field 

define the zones of influence of a member by entering the corresponding widths in m, left and right of it, for both wall and roof members

⚠ *The "Left" and "Right" of a member is determined based on the local x-axis (red).*

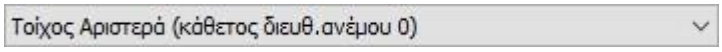


"Pick" and left click on the member or parts of the member. The zone of influence is displayed on the screen as in the figure below.



4.4.2 Semi-automatic Process - Without using the "Standard Constructions"

A part has been added on the right concerning the manual definition of the three-point surface. The surface definition is always done on the specific wall that is active in the window



OBSERVATION:

⚠ *It is advisable before starting the process to reset everything that exists by pressing the "Reset Members" button.*

The points are shown graphically with the following peculiarity:

- The first two points define the direction in which the automatic calculation of the influence surfaces is performed for the elements that are parallel to this direction.
  Note also that the distribution is done for all linear members belonging to this plane and as we said it is parallel to the first line.
- After defining the 3 points, we press the "Distribution" button and the program automatically executes the distribution and displays it.

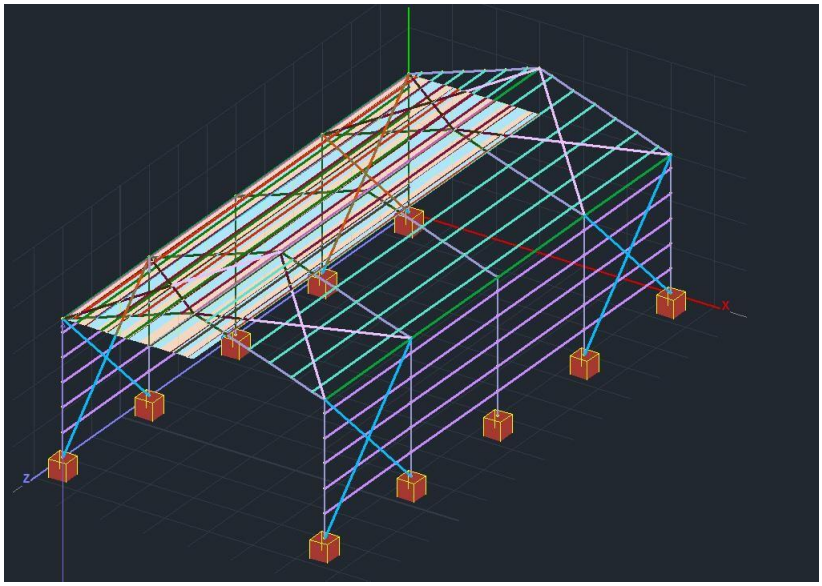
The same definition is made for the other walls.

As far as roofs are concerned, the definition can be done sequentially, i.e. after I first choose the roof to define

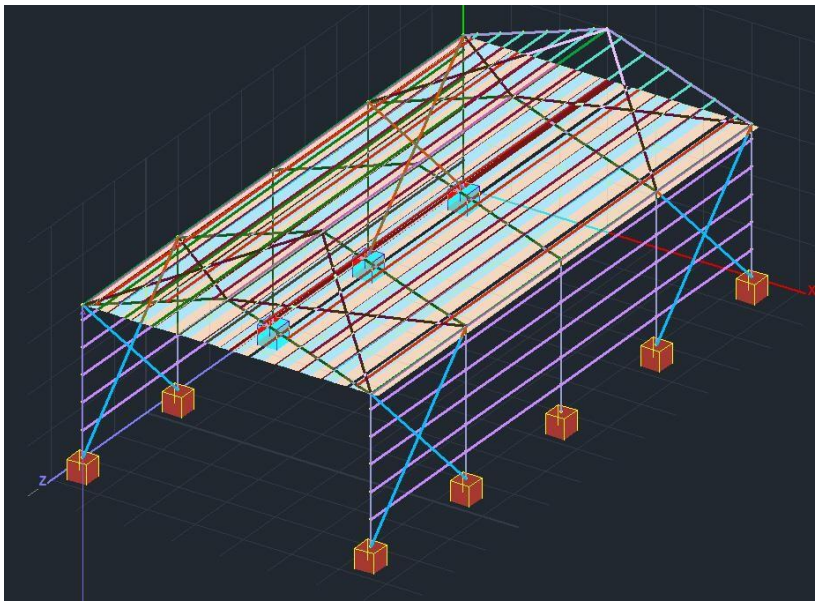
it is obligatory to select the individual levels, i.e. in a two-pitch roof to select successively the two levels of the slopes, because as we said the logic is to define with three points a level for which and for those members belonging to it the calculation of the influence surfaces will be done automatically.

EXAMPLE:

For example, I first set the left slope



and then the right. The overall result is as follows



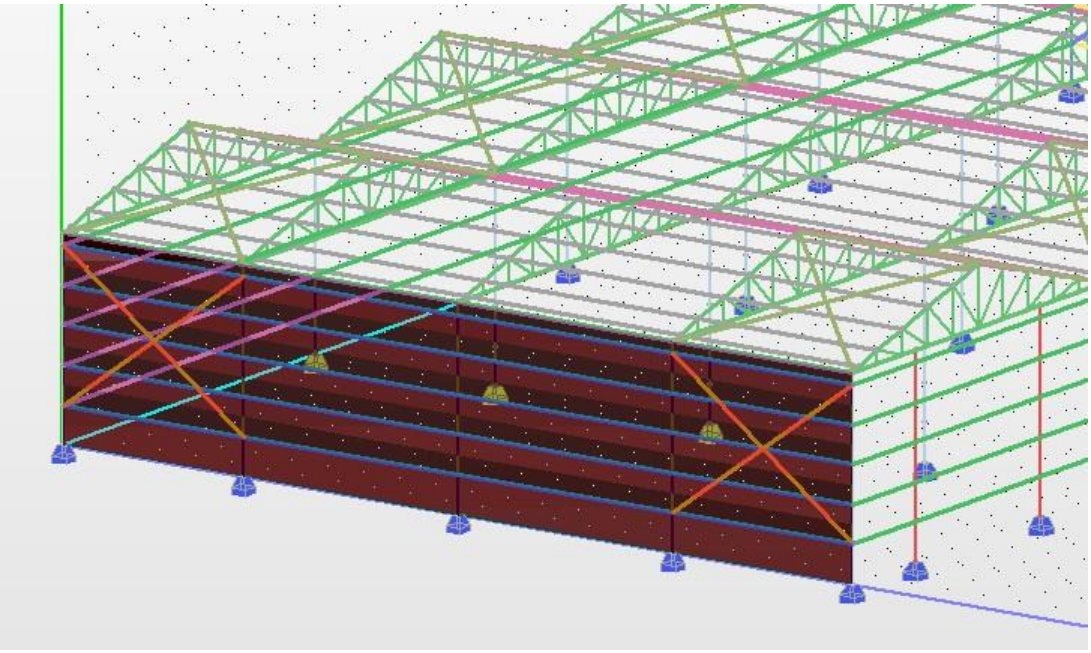
- Finally, it is worth noting that if the walls are correctly defined, there is NO need to define the levels. Simply select each wall and press the button "Allocation" is made and at the same time the allocation to the linear members belonging to this wall is displayed.
- The same applies to roofs that, attention is on one level. For the rest of them (e.g. two-plane roofs), however, the procedure of defining the individual planes described above is needed.

4.4.3 Automatic Process - Using the "Standard Constructions"

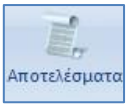
With "Tects" and "Mects" activated, the "Load Performance" field of the

Απόδοση Φορτίων	
Τεγίδες	Ναι
Μηκίδες	Ναι

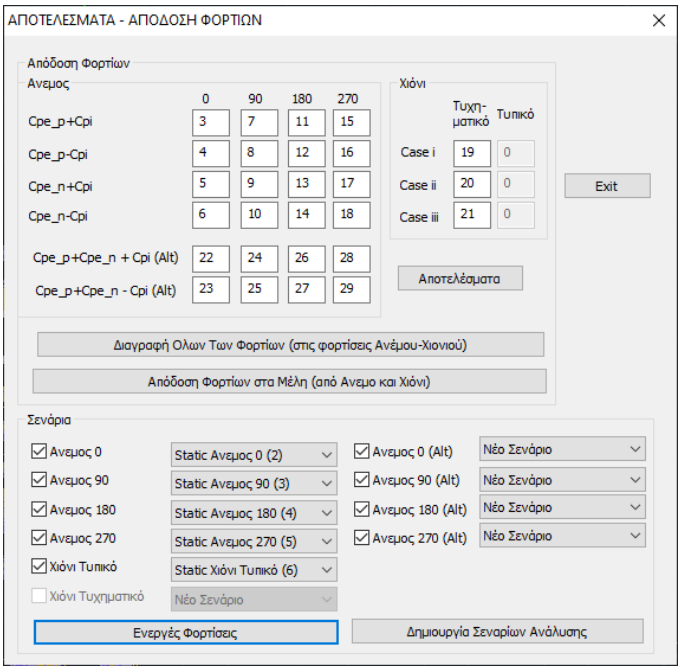
"Standard Construction" , just select "Pick" and the program automatically calculates the zones of influence by distributing the stresses to all the tectiforms and mectiforms.



4.5 Results

Last command, the "Results" command. 

In the dialog box, in the "Load Performance" field are there two tabs?

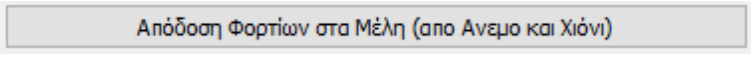


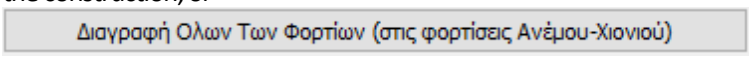
- wind loads, 4+2 cases for 4 directions,

- loads of snow, 3 cases for typical snowfall (random is not applicable in Greece).

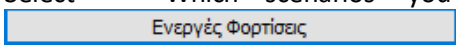
The numbers shown on the labels are the numbers of the charges.
Recall:
- Charge 1: Permanent
- Charge 2: Mobiles

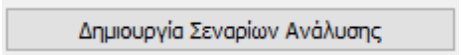
and now add another 24 wind charges (from 3 to 18 and 22 to 29) and 3 for snow (19, 20 and 21)

Select the command 

to attribute loads of wind and of snow to members the construction , or members the construction, or 

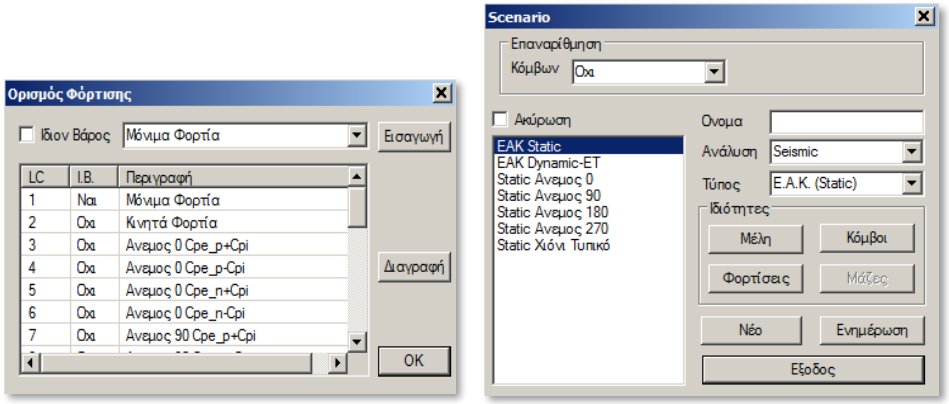
to delete them all.

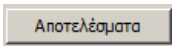
The "Scenarios" field contains a list of all possible analysis scenarios.
In the right column there are alternative scenarios in which the pressures and subpressures are overlapped to find the worst (for the context) combination.
Select Which scenarios you want to activate and click at 

Then on to created these the scenarios click 

OBSERVATION:

⚠ *So SCADA Pro, besides automatically calculating the distribution of wind and snow loads, automatically creates all the analysis scenarios, saving the user a lot of work and time.*



The order 

opens a txt file of the results, detailing all the data and calculations resulting from each Eurocode 1 group command.