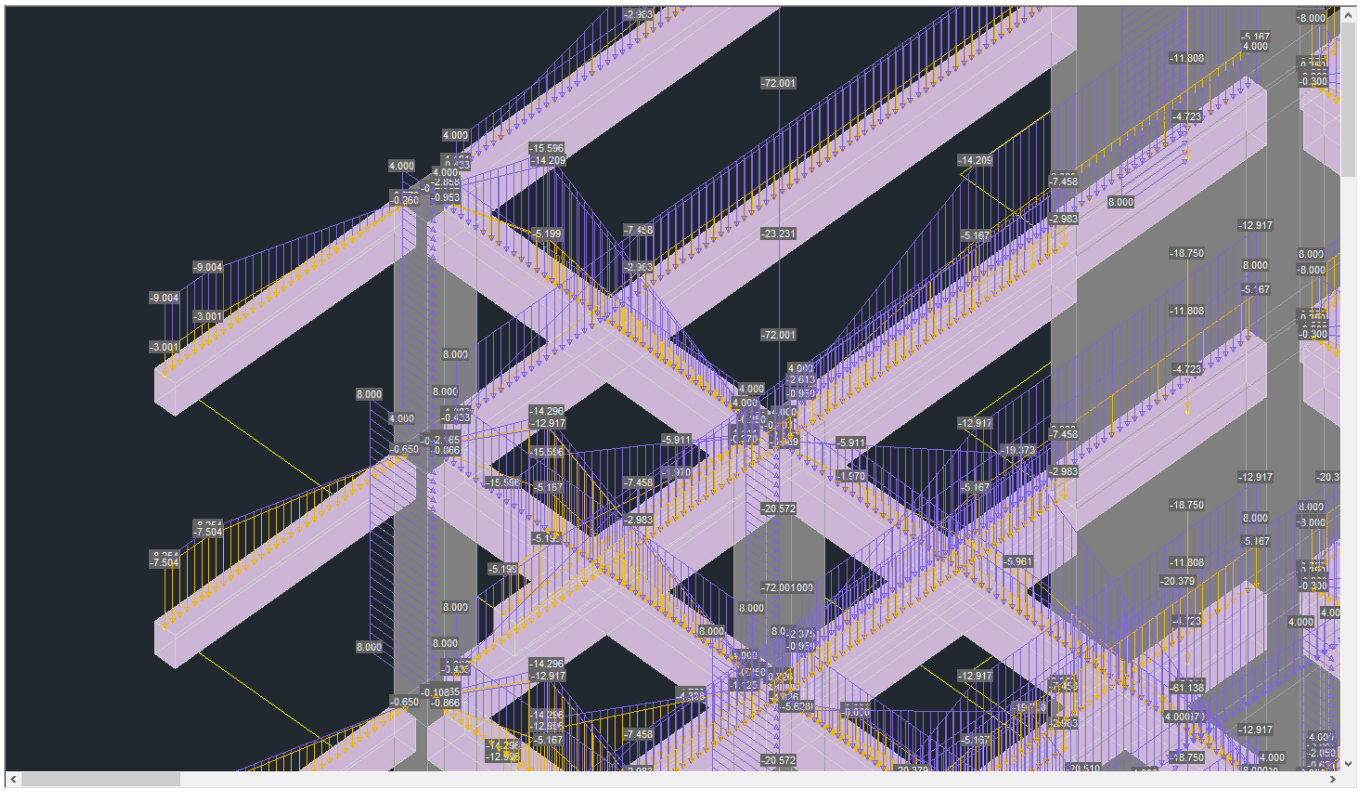
SCADA Pro™
Structural Analysis & Design

User's Manual

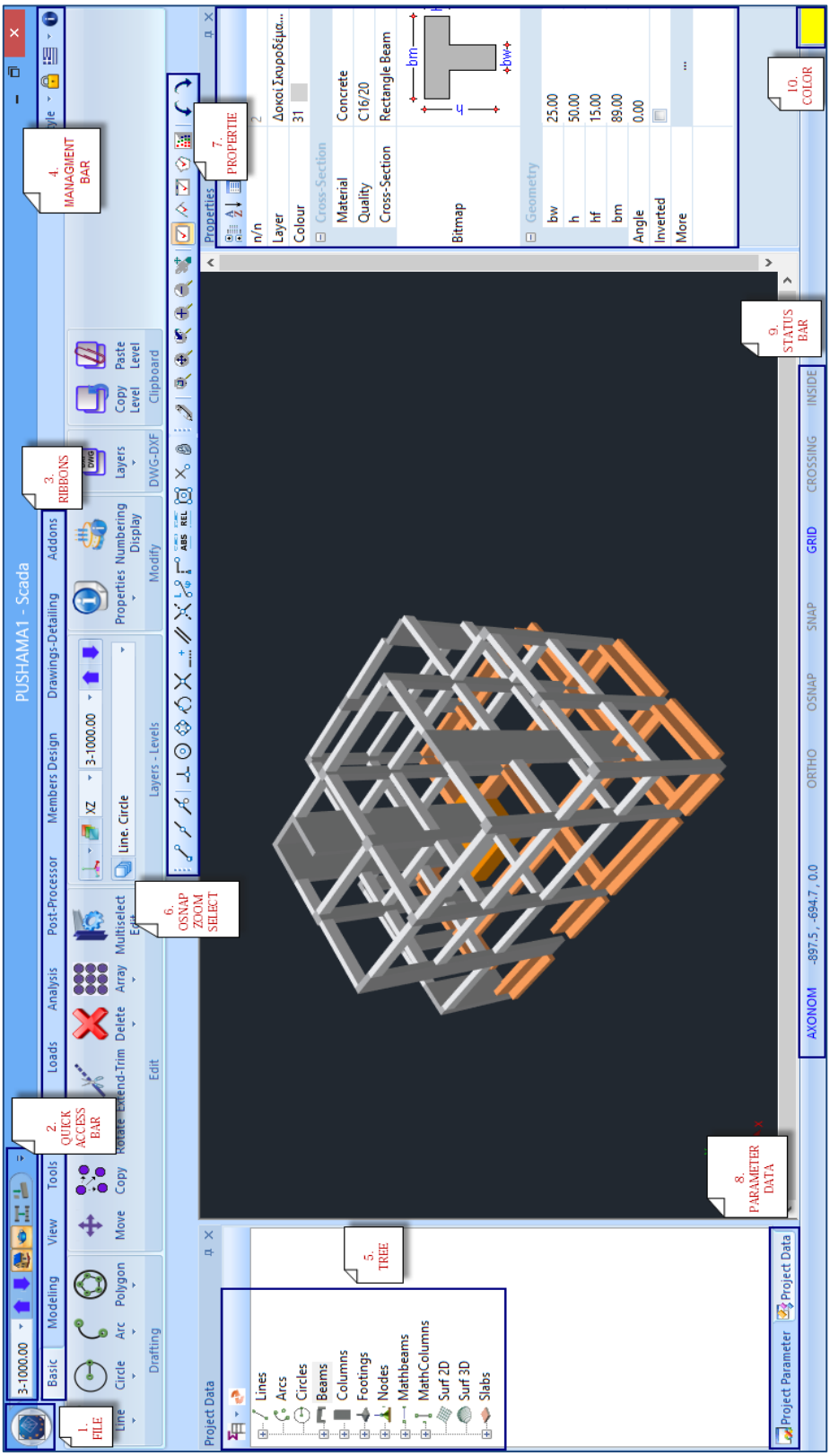
6.LOADS



ACE HELLAS
INTEGRATED SOLUTIONS

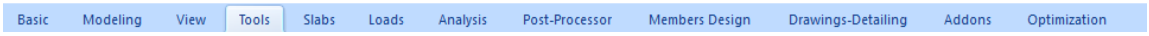
CONTENTS

I. THE NEW UPGRADED INTERFAFE OF SCADA Pro

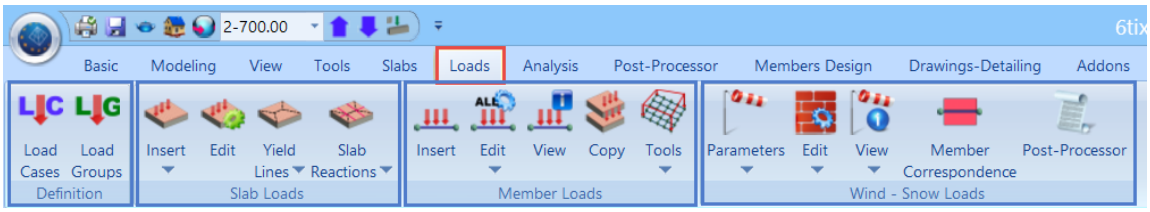


II. DETAILED DESCRIPTION OF THE NEW INTERFACE

In the new upgraded SCADA Pro, all commands are grouped in 12 Units.



LOADS



The 6th Unit is called “Loads” and includes the following four groups of commands:

1. **Definition**
2. **Slab Loads**
3. **Member Loads**
4. **Wind and Snow Loads**

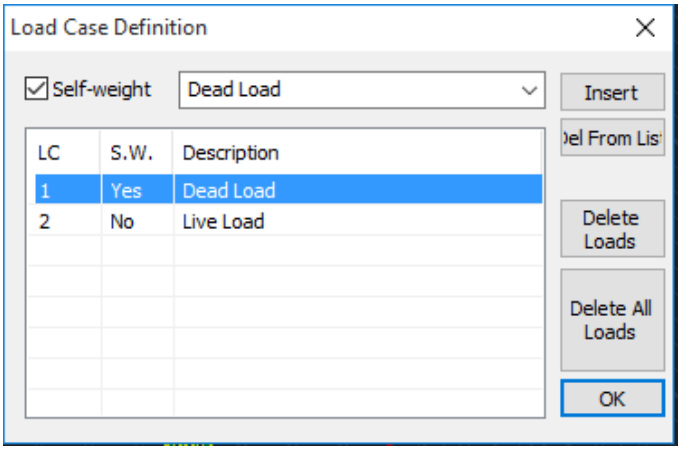
1. Definition

“Definition” command group allows the definition of the loads and their corresponding groups.

⚠ *The basic condition for loads application is the definition of the respective load cases. Each load will belong to one of those cases.*

1.1 Load Cases

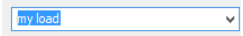
To define the Load cases use the “Load Cases” command. In the dialog box:

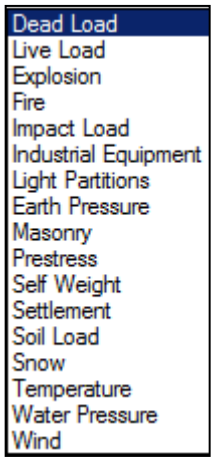


There are two default load cases:

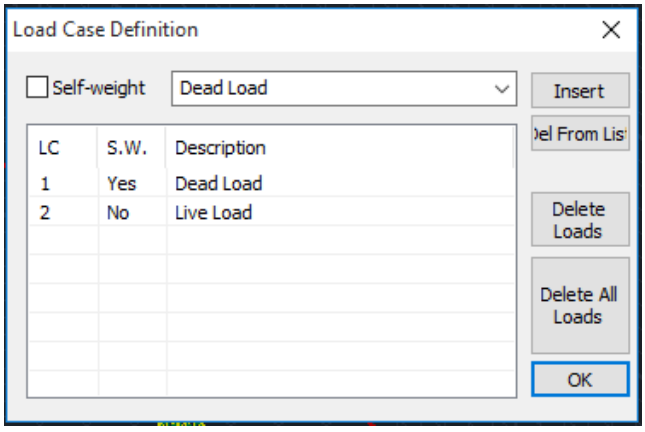
1. Dead Loads (L.C.=1)
2. Live Loads (L.C.=2)

⚠ *The S.W. column indicates the participation of the self-weight in the specific load case.*

In addition to dead and live default loads, you can also define other loads. You can choose loads from the list, or you can set your load by typing a name  and then click the "Insert" button.

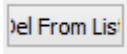


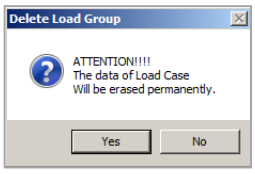
The imported load, takes a serial number, in LC (Load Case) column and a "Yes" or "No" indication, depending on, whether or not, the self-weight is including.



Press OK to save and close the window.

⚠ *To include the self-weight of the structure in a load case, check "Self Weight" selection*  .

- To delete a load case from the list, first select it and then press  The program will ask you to confirm your choice. Click "Yes", to delete.



⚠ *You can delete a load case only if it does not include loads.*

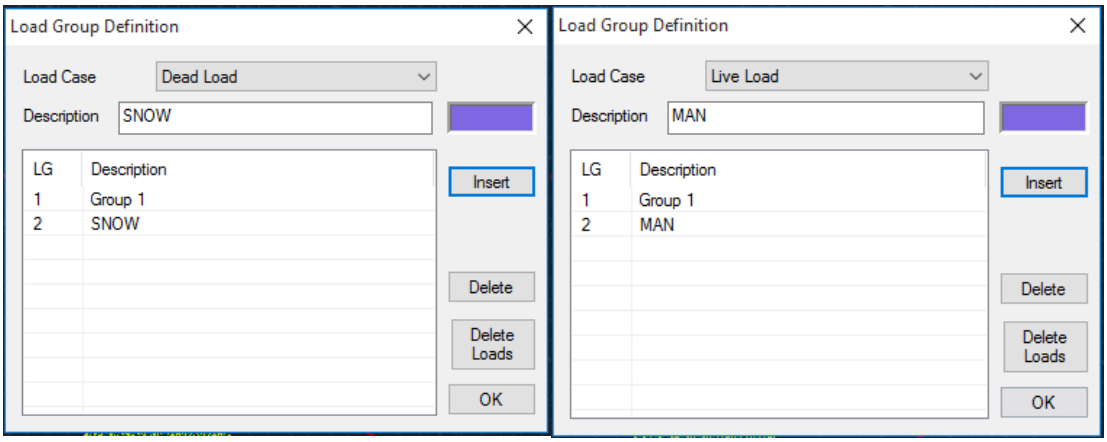
- In order too delete the loads from a case first select the load and then click Delete Loads.
- To delete all the loads from all cases click Delete All Loads

1.2 Load Groups

Load Groups Use "**Load Group**" command to create load groups that belong to an existing load case as an optional procedure. Dead and Live Loads contain as default a predefined group "Group1".

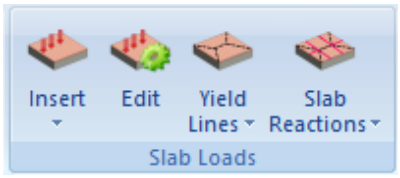
EXAMPLE:

In case you want to define a new group, for example, a SNOW group in Dead Load and a MAN group in Live Load,
select at first Dead Load from the list, then on "Description" type "SNOW", select a color (press on) and click "Insert". So, SNOW Load becomes (LG) 2.
Do the same to define MAN Group in Live Load as (LG) 2, while Group1 refers to the standard live loads.



- Press "Delete" to delete an existing group.
  Prerequisite for deleting a group is that it does not include loads.
- To delete the loads included in a group (loads already entered), choose from the list of the groups this group (to turn blue) and then press Delete Loads
- To display loads of each group of the same load case use the "View" command View.

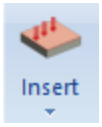
2. Slab Loads



"**Slab Loads**" command group contains the commands for inserting, editing and assigning the slab loads on members and nodes.

2.1 Insert

The command contains two kinds of loads input:

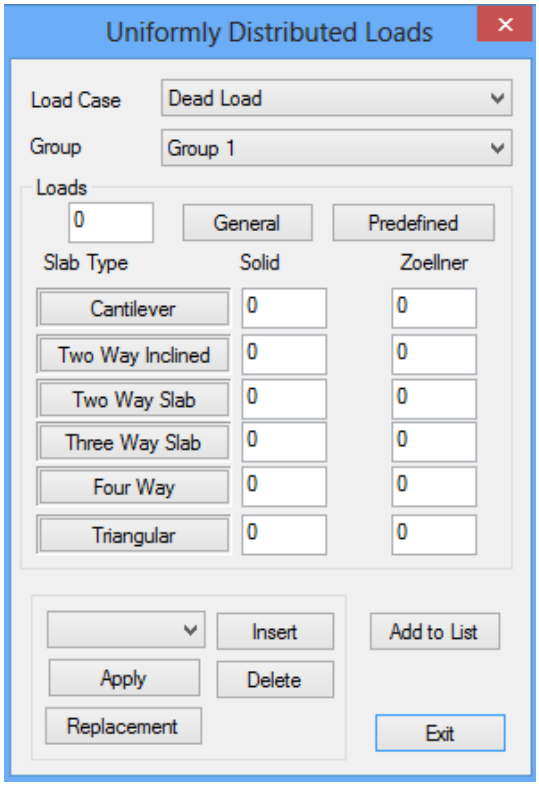


to all current level slabs

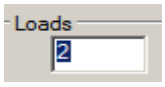
only on the selected slab

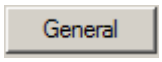
2.1.1 Overall

Overall: Select the command and in the dialog box:

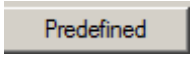


First, define the Load Case and Group and then type the load value for each type of slab.

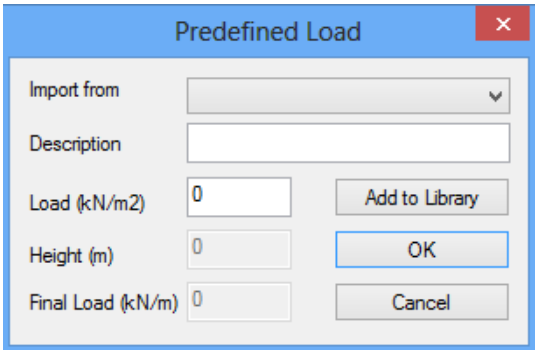
- You can proceed by typing in 

a value in kN/m^2 and then press 

to fill in all fields with the same value.

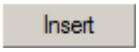
- You can also use predefined values by clicking on the 

button that opens the next dialog box:

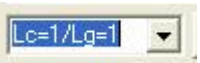


Select from the "Import from" list the predefined load to define loads directly, or define your load, by typing a "Description", a value "Load (kN/m^2)" and in case you want to save the load in the library click the button 

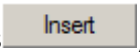
.

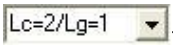
Define the loads and press 

.

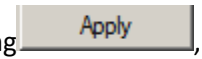
The load case and group 

are displayed in the list (Lc=1: Load Case 1/Lg=1: Group 1) automatically.

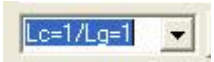
Follow the same procedure for the other load cases (ex. Live Load) and press 

, 

to display the Load case and group (Lc=2: Load Case 2/Lg=1: Group 1).

By selecting 

, all defined loads are applied to all current level slabs.

The loads' assignment for the first time means that the loads in the list 

will be applied to all current level slabs.

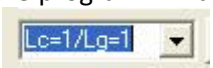
But, in case of slabs containing existing loads, by clicking on the 

button, the existing loads will be replaced.

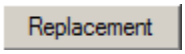
EXAMPLE 1:

Suppose that you have already assigned loads in all current level slabs with dead and live loads.

- If you define a new value for the dead loads and press 

, the program will apply the new dead load value and 0 live loads (the list contains only dead load 

and no live load).

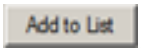
- But, if you want to replace dead loads and keep the existing live loads, then press 

- Press 

to cancel an inserted load from the list.

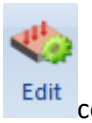
EXAMPLE 2:

Suppose that a dead load of 1 kN/m^2 is already applied on a slab and you want to add another dead load 2 KN/m^2.

Define the load and press 

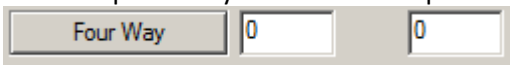
.

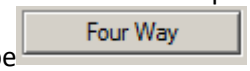
Type	LC	LG	Value	Status	
Uniform	1	1	1.00		
Uniform	1	1	2.00		

Select 

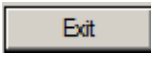
command to confirm.

EXAMPLE 3

You can also replace only the value of a specific Slab Type. Type the value in the corresponding space 

and then click the type 

. This value will replace the first one to all slabs with the same type.

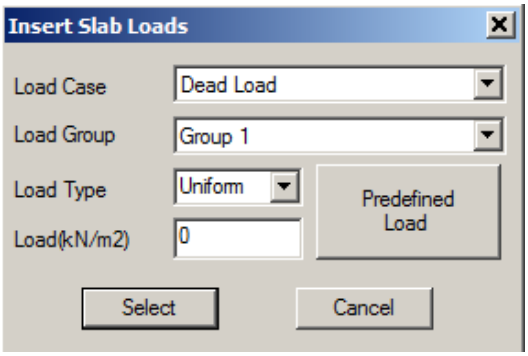
Press 

to close the dialog box without saving, or press 

to save the changes.

2.1.2 By pick


select the command and then left click inside a slab. In the dialog box:



Define “Load Case”, “Load Group”, and type the value in KN/m2. Then select “Load Type”.

There are 3 “Load Types”:

- **Uniform**

Insert uniform loads over the entire surface of the slab. Define the load and left click inside the slab.

- **Partial**

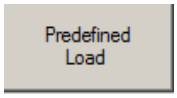
Insert partial load in a specific area of the slab. Define the load and left click inside the slab. Then select a side to identify the direction and then left click to indicate a vertex and movthe e mouse to describe the load area.

- **Linear**

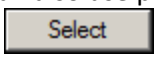
Insert linear load over the slab and follow the same procedure as described in partial loads. To define the position of the load, left click to identify the two ends of the line (start and end point).

⚠ *Partial and Linear loads, regarding the attribution to the slab members, will be replaced by an equivalent uniform load on the entire slab*

⚠ *Define "Load" in kN/m^2.*

You can also use predefined values using 

button, as previous.

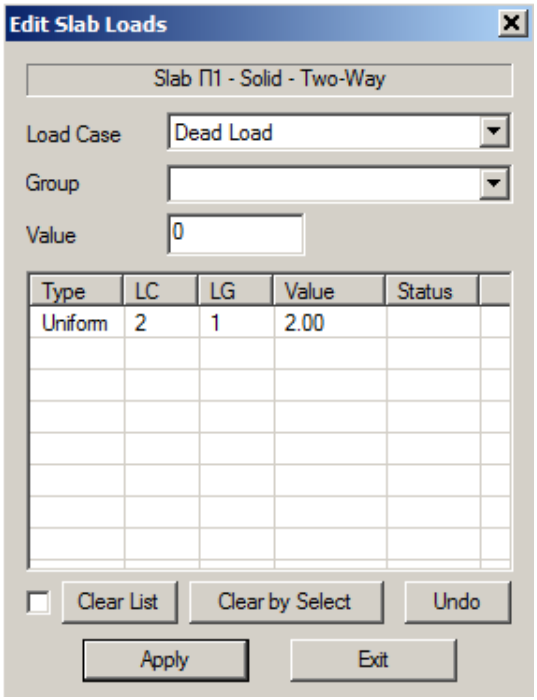
Press 

to close the dialog box and click inside one or more slabs' area to apply the load.

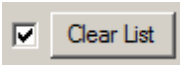
2.1.3 Edit

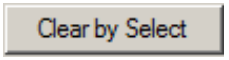
To edit and modify slab loads use/command.

Select the command and click inside a slab. In the dialog box:



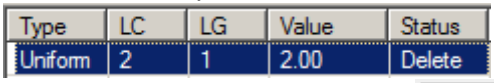
Select Load Case and Group, then from the list select the load for editing.

Activate 

and all the loads will be deleted from the list. Otherwise, press 

to delete only the selected load from the list.

⚠ *Loads are not deleted immediately, but first "Delete" is displayed on "Status" column, which means "ready to delete".*



Type	LC	LG	Value	Status
Uniform	2	1	2.00	Delete

To delete them permanently, press 

.

Undo command invalidates the previous action (cancels "Delete" designation in Status column)

Press Exit to close the dialog box without saving, or Apply to save the changes.

2.1.4 Yield Lines

Load areas' calculation resulting from the geometric partitioning of the slab, and then used to calculate the design forces for beams (slab loads which will be imposed on beams),
The calculation is automatically made by the program according to the support conditions, either Overall or By Pick.

Overall: select the command (Load areas calculation for all current level slabs)
By Pick: select the command and then left click inside one or more slabs (Load areas' calculation for the selected slabs)

- *Yield Lines calculation concerns rectangular slabs or slabs with equivalent rectangular modeling.*
- *Yield Lines calculation is made according to the support conditions*
- *Yield Lines calculation is applied to conventional slabs and not to slabs with surface elements.*

2.1.5 Slab Reactions

By selecting Slab Reactions, slab loads are assigned on beam members as reactions. More specifically, loads are distributed from slabs to beams and columns, based on the geometric partitioning done previously (Yield lines).

2.1.6 Overall:

 select the command (Load distribution from all the current level slabs).

2.1.7 By Pick:

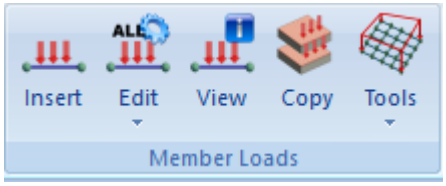
 select the command and then left click inside one or more slabs (Load distribution from the selected slabs)

2.1.8 Equivalent

means: to assign (Overall  or By Pick  , respectively) the slab loads on the connected members, without considering the yield lines evaluation (rectangular and triangular areas). Instead, the assignment is implemented by the convertion of the entire area corresponding to the member, in an equivalent rectangle.

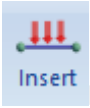
⚠ *Slabs' load distribution as a reaction of the connected members, for slabs without any required simulation, is implemented without the generation of equivalent models (uniform, triangular, trapezoidal loads, etc.).*
⚠ *In slabs identified by mathematical members entered by the user (e.g. beams with high rigidity for the simulation of basement walls), loads are assigned to the mathematical members, regardless of whether linear members (either selecting the mathematical member or the physical cross section) are chosen for the modeling and matching of the sides of the slab.*

3. Member Load



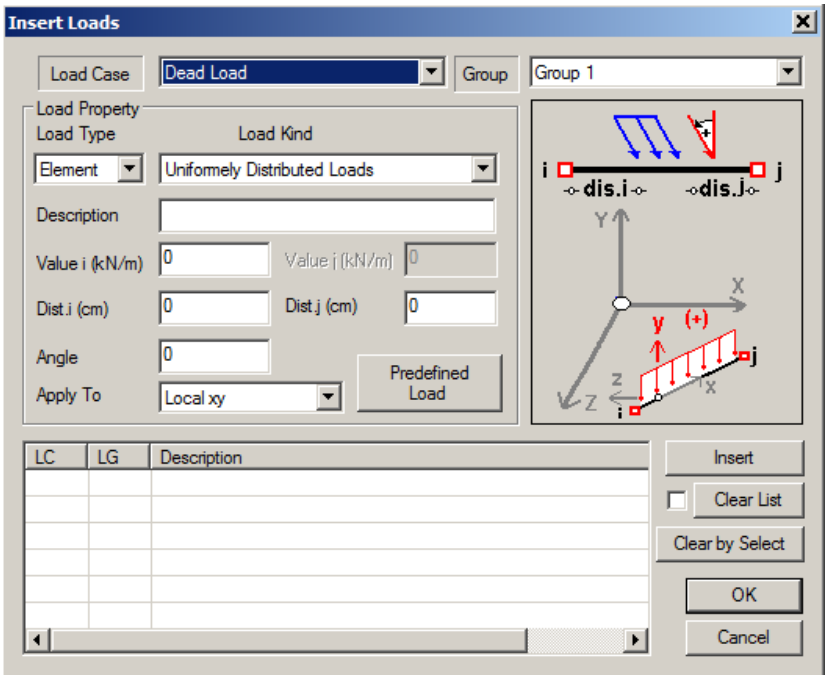
"Member Loads" commands' group contains the commands for inserting, editing, viewing and copying loads of members, nodes and surfaces finite elements.

3.1 Insert



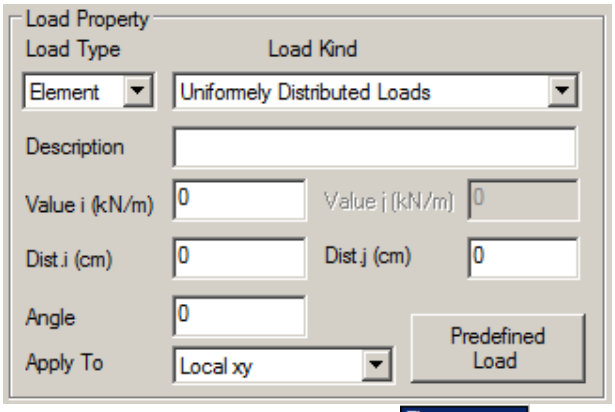
loads on members, nodes and surface finite elements.

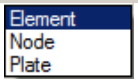
Select the command and define the elements (member, node, and surface) to insert the loads to. To select the elements to use  . Complete selection by pressing the right mouse button and then the following dialog box appears:

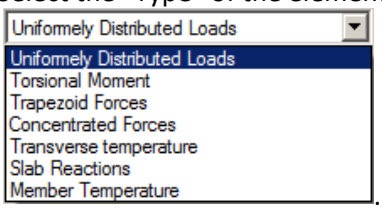


The lists on top, contain all the possible load types depending on the element type.

“Load Property”:



Select the “Type” of the element 

and the “Type” of the load 

.

According to the element “Type” and the load “Type”, the “Load Property” is modified. Fill in the fields according to the drawing, type a description, the values, and the corresponding distances.

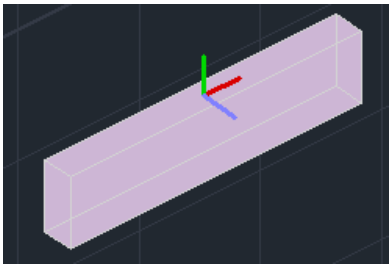
Member Load Sign :

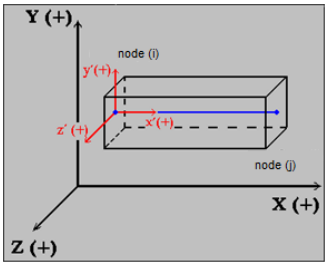
Loads' sign convention is made versus the local coordinate system of each member, which is based on the rule of the "right hand".

Specifically:

1. BEAMS :

x-x is the local axis directed from the start to the end point (red vectors),
y-y is the vertical axis (perpendicular to the local axis x-x) parallel to the height of the slab (green vector). It is always directed like Y Absolut axis (bottom-up).
z-z is the third vertical axis, perpendicular to the plane defined by the xx and yy local axes (blue vector).



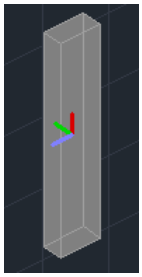


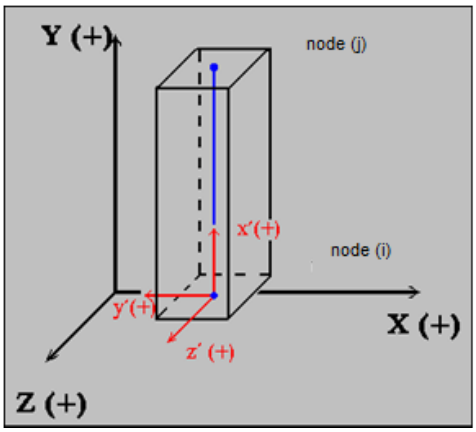
2. COLUMNS :

x-x is the local axis directed from the start to the end point meaning bottom-up direction (red vectors),

y-y is the vertical axis (perpendicular to the local x-x) directed like X Absolut axis (green vector).

z-z is the third vertical axis, perpendicular to the plane defined by the xx and yy local axis (blue vector).

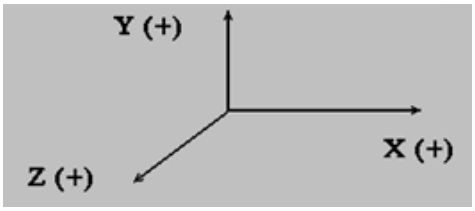




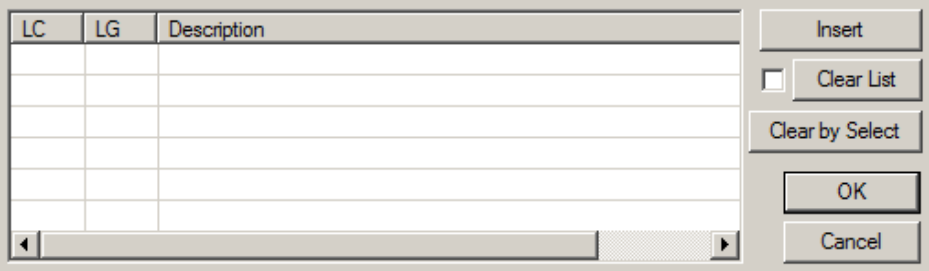
⚠ *Beams and columns local axes can also be defined using the rule of the right hand with your thumb along the positive axis xx, the index finger along the positive yy and the middle one along the positive zz.*

Nodes Load Sign :

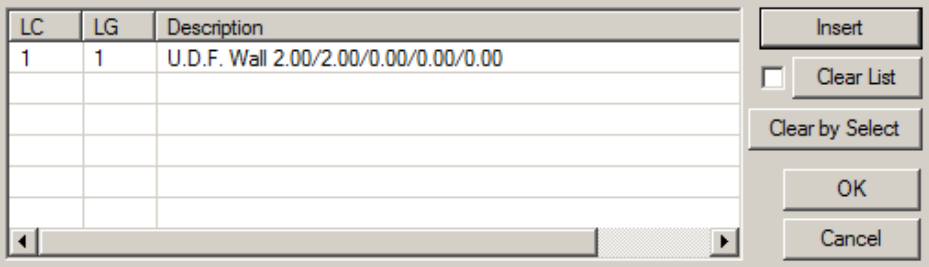
Nodes' loads are always directed according to the Absolute X, Y, Z axes.



The next dialog box section is the load list.

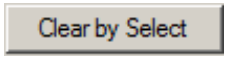


The list is filled in by defining the loads and selecting the "Insert" command.



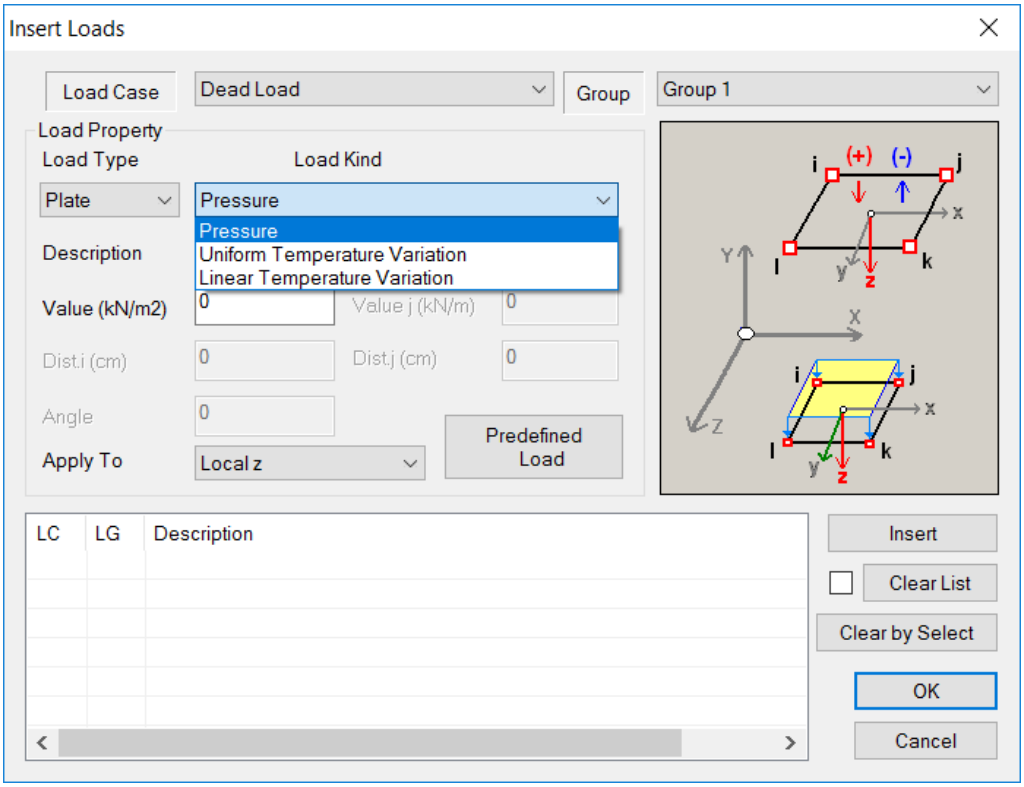
EXAMPLE:

Insert a uniform distributed load (U.D.F. Uniformly Distributed Force) that belongs to Load Case (LC) 1 (Dead Loads) and Load Group (LG) 1. The numbers after the description (Wall) are: start load value, end load value, the distance of the load from the beginning, the distance of load from the end and the angle.

By activating/, all the loads on the list will be deleted. Otherwise, press 

to delete only the selected load from the list.

Plate Loads



Allows you to define a **Pressure**, and also the possibility to enter **Temperature Variations** load for finite surface elements is added.

More specifically, for Plate (shell) elements **Uniform Temperature Variations** and **Linear Temperature Variations** loads are added.

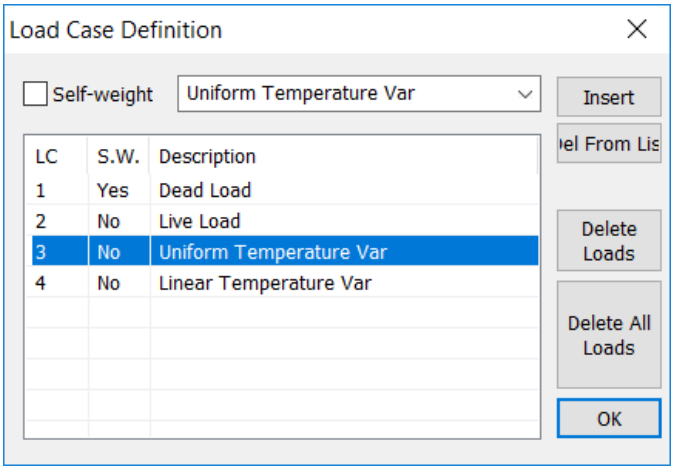
- **Uniform Temperature Variations** causes membrane deformation in the plane of the element, while
- **Linear Temperature Variations** causes deflection.

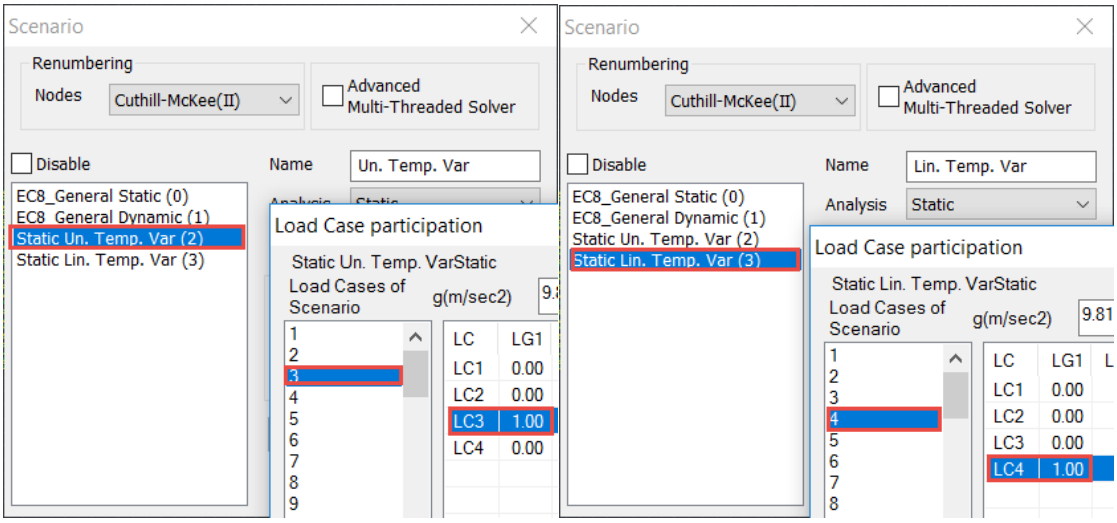
NOTE:

⚠ We have to note that the two loads of the **plate** element can be integrated either on the same loading or in two different loadings.

- Integrating both loads at the same analysis scenario, you will get aggregated results in one load (the first).
- Considering two different loadings, to obtain individual results, each load MUST go to a different analysis scenario.

The procedure to follow is:

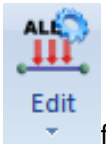




NOTE:

⚠ For **Plane** elements (Stress, Strain, Axisymmetric) only **Uniform Temperature Variation** is possible.

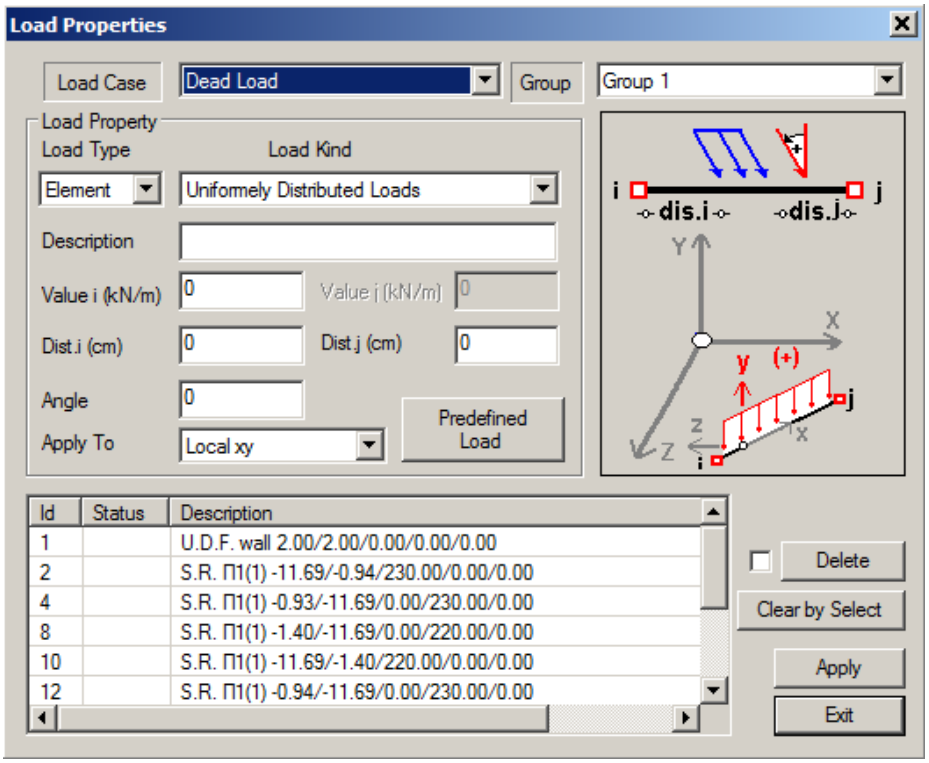
2.2 Edit:



for editing the existing loads' properties.

2.2.1 Overall:

for editing existing loads' properties on the current levels. Select the command and the dialog box opens:



In the load list, you can see all the imported loads according to the selection.
For example, select Dead Load/Group 1/Element/ Uniformly Distributed Loads.
The list describes all the existing loads according to the selection.
(U.D.F. Uniformly Distributed Force, S.R. Slab reactions)

When you choose a load, the values appear on the top of the window where you can modify them.

Press the Apply command to save the changes.

By activating ☑ Delete, all the loads on the list will be deleted. Otherwise, press Clear by Select to delete only the selected loads.

Loads are not deleted immediately but first, the "Delete" indication is displayed on "Status" column, which means “ready to delete”.

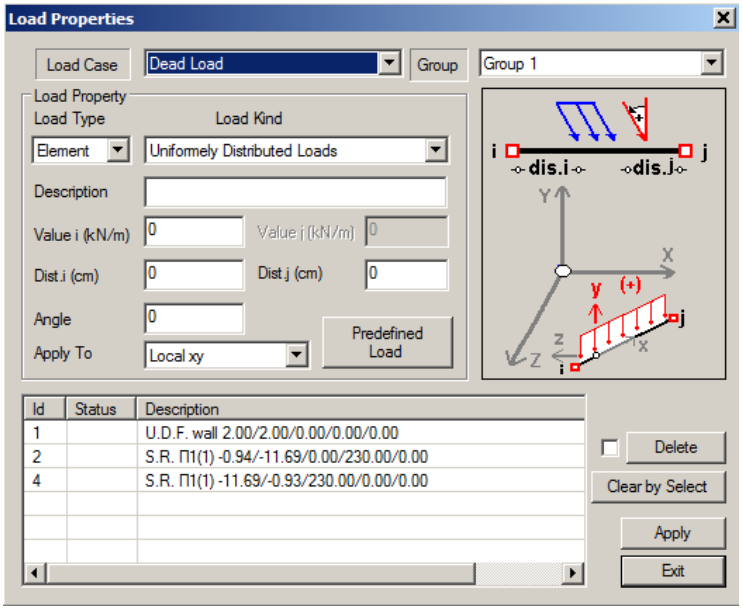
To delete them permanently, press Apply.

Id	Status	Description
9	Delete	U.D.F. 1.00/1.00/0.00/0.00/0.00
11	Delete	S.R. Π1(3) -0.32/-5.93/0.00/109.27/0.00
13	Delete	S.R. Π1(3) -5.93/-1.01/215.73/0.00/0.00
16	Delete	S.R. Π1(3) -1.01/-5.93/0.00/305.73/0.00
18	Delete	S.R. Π1(3) -5.93/-5.93/109.27/215.73/0.00

Press Exit to close the dialog box without saving, or Apply to save the changes.

2.2.2 Pick


for editing the load properties of the selected element. Select the command and left click on a member, node or surface finite element and the dialog box appears:



In the load list, you can see all the imported loads of the selected element.

For example, select Dead Load/Group 1/Element/ Uniformly Distributed Loads.
In the list, all the existing loads according to the selection will be descripted.
(U.D.F. Uniformly Distributed Force, S.R. Slab reactions)
When you choose a load the values appear on the top of the window where you can modify them.
Press Apply command to save the changes.
In the list, loads of the specific member are displayed. For example, the Uniform Distributed Force and the Slab Reactions of the selected member. By choosing a load the values appear on the top of the window where you can modify them. Then press “Apply” to save.

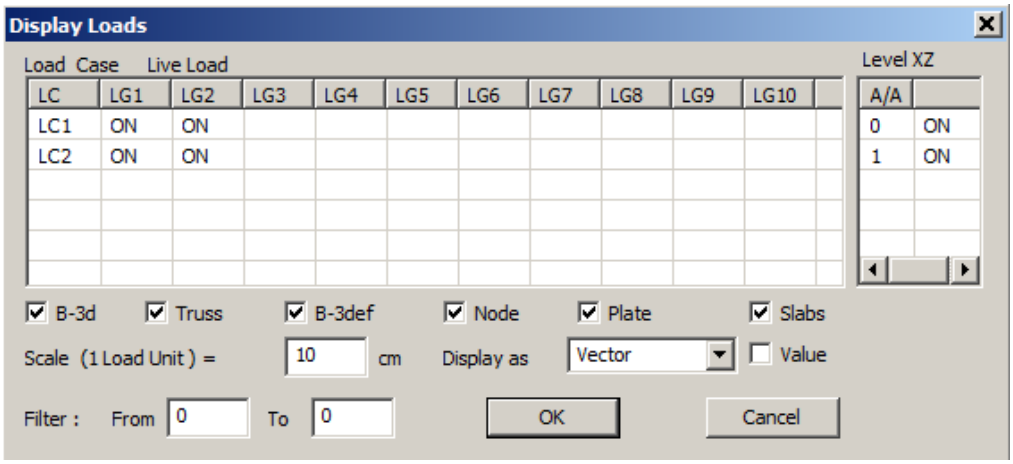
⚠ *Loads are not deleted immediately but first, the "Delete" is displayed on "Status" column, that means “ready to delete”.For the deletion to be applied, press* Apply .

Press Exit to close the dialog box without saving, or Apply to save the changes.

2.2.3 View

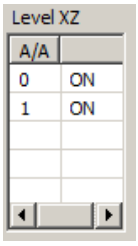
for the display of the loads for all elements, in 3D view as vectors, with or without values, or in 2D view as a number.

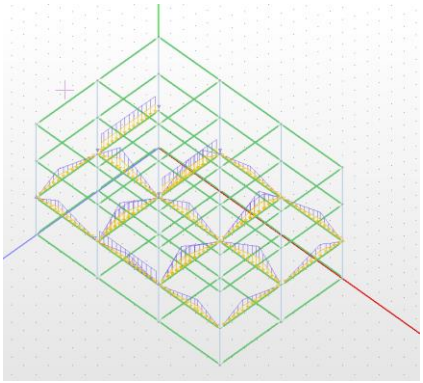
Select the command and in the dialog box:
the defined Load Cases and the Load Groups contained are displayed. Each load group contains a switch ON or OFF (display or not display), that changes by clicking on it.



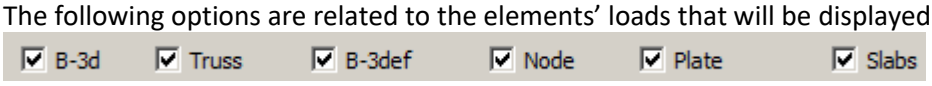
In the picture above there are two loads LC1 (Dead) and LC2 (Live). Each load contains a default group LG1 and a created group LG2, that are all “ON”, which means that all loads will be displayed.

On “Levels XZ” select ON or OFF, to display or not, loads of the corresponding level.



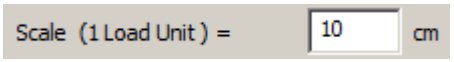


The following options are related to the elements’ loads that will be displayed.

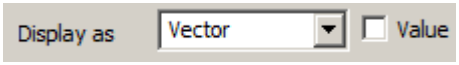


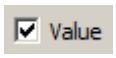
Activate 

to display the value of the loads.

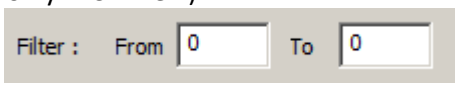


to set the visualization scale of the vectors. Type in the value in cm.



in 3D view select "Vector" and activate 

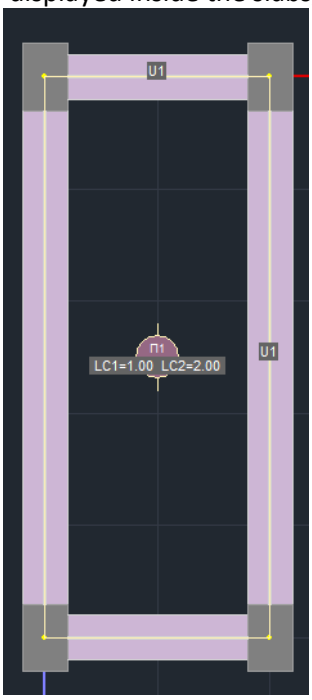
to display the value of the loads. In 2D view select "Number" ("Value" activation doesn't change anything. Load values are visible only in 3D view).

Finally, using the filter 

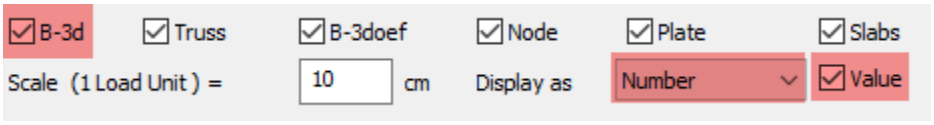
you can specify a range of loads to be displayed.

Also, by choosing 

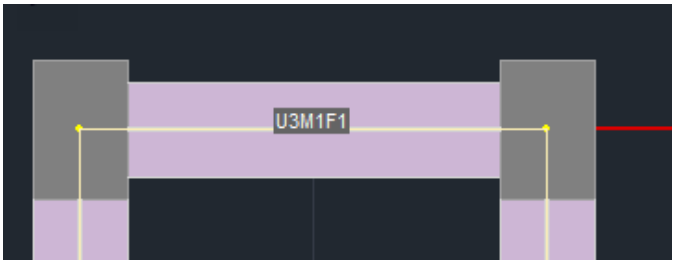
, the values of the slab loads are displayed inside the slabs, in the 2D display.

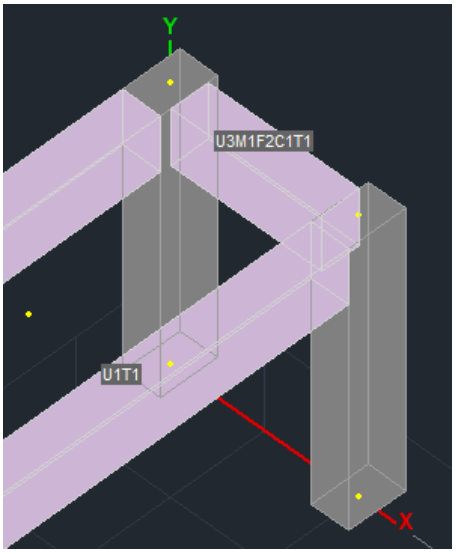


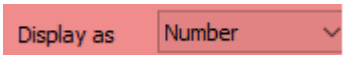
Respectively for members, with B-3d selected,



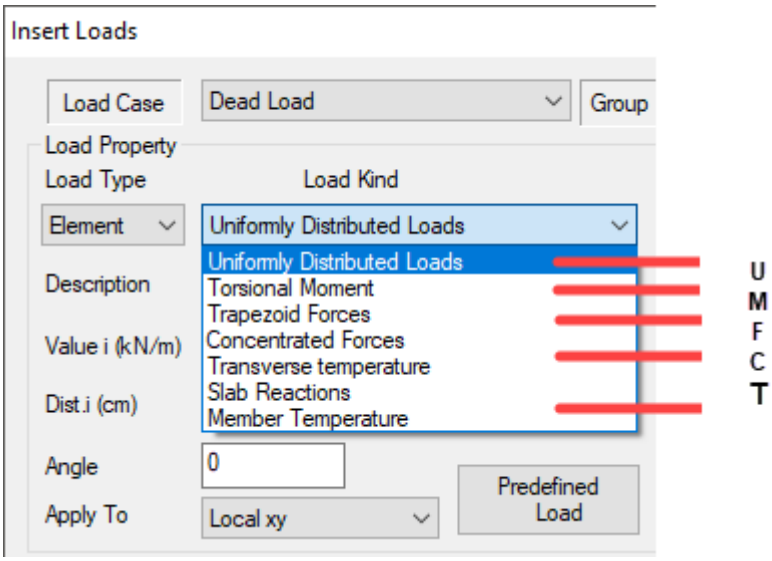
the presence of loads in letters and numbers is displayed on the member.



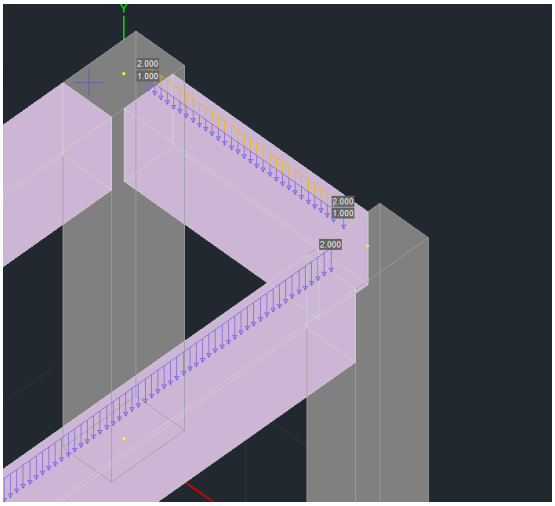


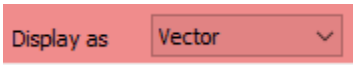


Depending on the type of the load (U,M,F,C,T) :



And the number indicating how many loads of that type exist

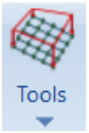




Finally, in the option Filter 

you can define a value range for the loads you wish to appear.

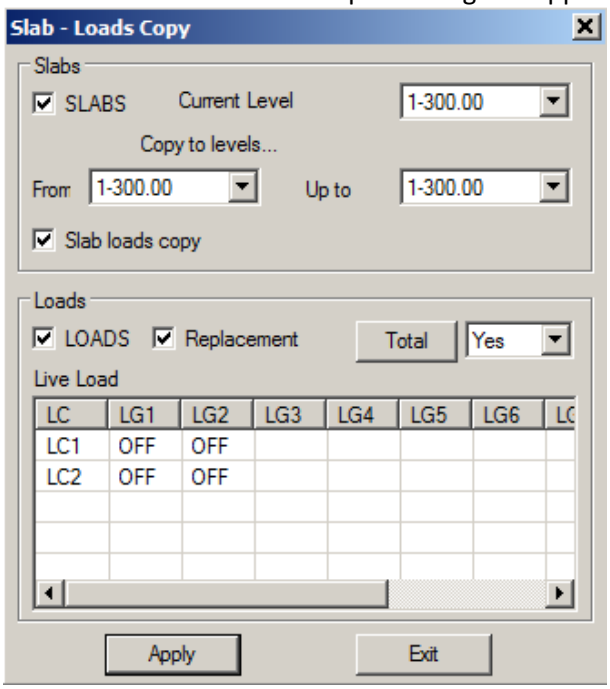
2.2.4 Copy



To copy slabs and loads from one level to another.

⚠ *Use the command <u>only in case you have a typical floor</u>, i.e. the floor is the same.*

Select the command and a 2 part dialog box appears



1. "Slabs" part:

- activate "SLABS", select current level (copy level) and "paste levels" from/to.
- Activate 

if you want to copy the slab loads as well.

2. "Loads" part:

- activate "LOADS", press 

to switch ON all load groups, or

-activate "LOADS", press 

to switch OFF all load groups and then select by clicking on individually.

and 

→ *to replace loads of the selected load groups*

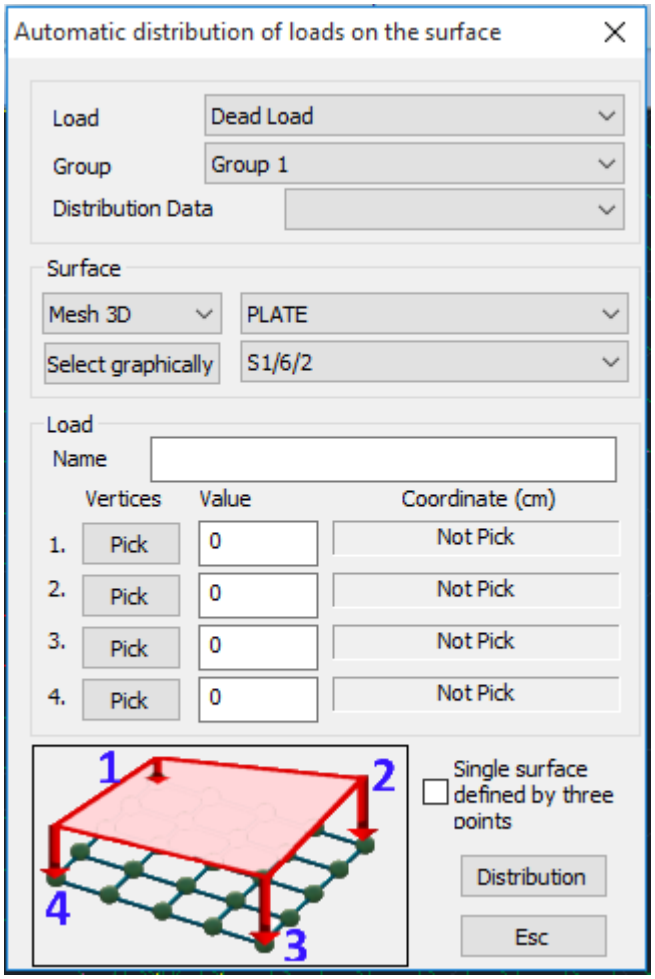
and 

→ *to apply loads of the selected load groups on the existing loads.*

2.2.5 Tools

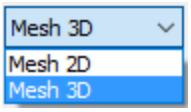
Command used for the automatic distribution of loads on a mesh area

The new version of SCADA Pro includes a new tool for the automatic distribution and application of the loads on the mesh area.
Select the command, and on the dialog box that opens, define:



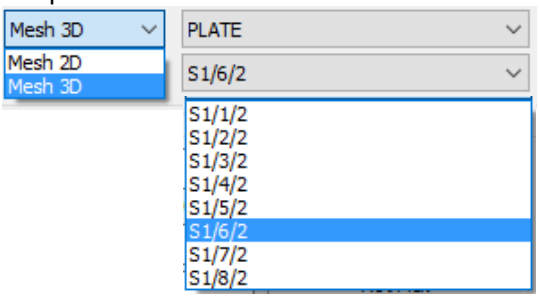
The type of load from the "Load" field and the group from the "Group" field.

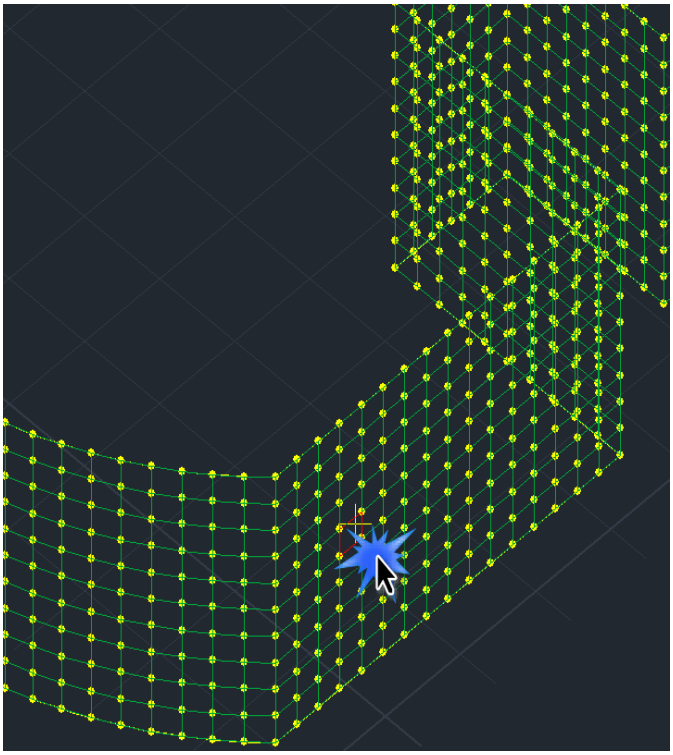
In the "Surface" field,



Select the type of the surface and the surface group that you wish to load.

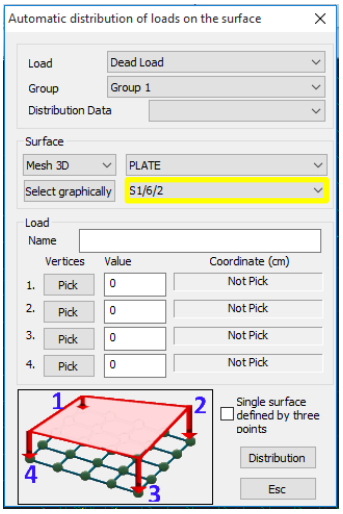
In case that the selected mesh group has more than one surfaces you must select from the list, the preferred surface as well.





The load application can be performed graphically as well, by clicking the button Select graphically.
The dialog box closes automatically so that you can identify the surface by clicking on one of its shell element.

Then the dialog box reopens with the pointed out surface identified.



On the "Load" field give a name for the load. Afterwards, define the way of the load distribution on the selected surface.
The definition can be performed graphically:

- By pointing the four corners of the surface and setting the load value.
- By pointing 3 points the first two of which define a straight line that the first load value will be applied, and the third which defines the height that the second load value will be applied.

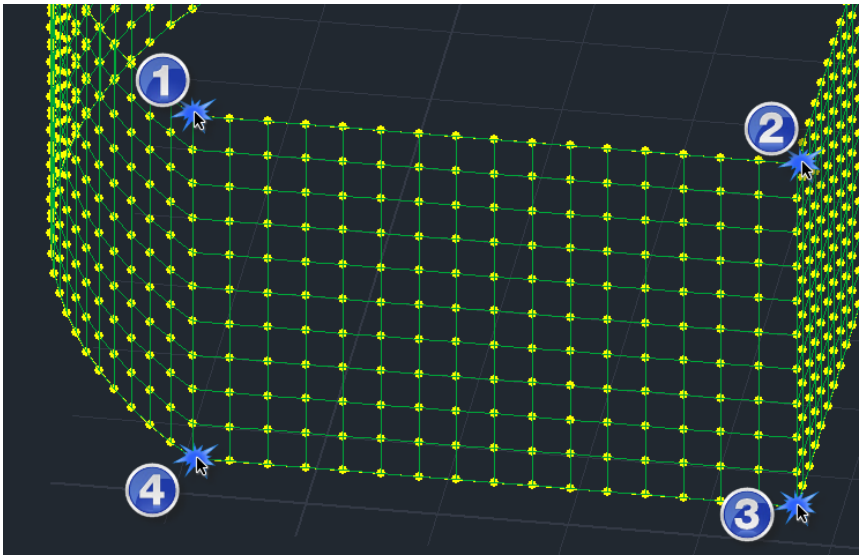
It is not necessary for these points to belong to the same level, while the outline of the surface can contain lines and arcs.

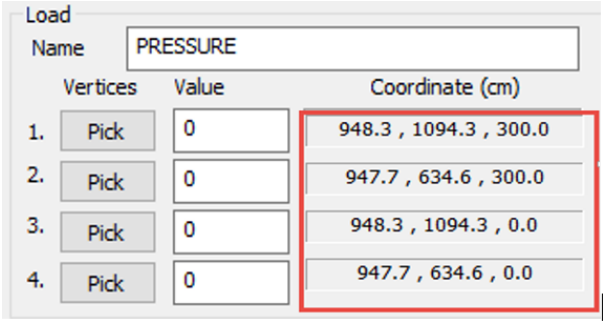
More specifically:

1. FLAT SURFACES:

1st EXAMPLE:

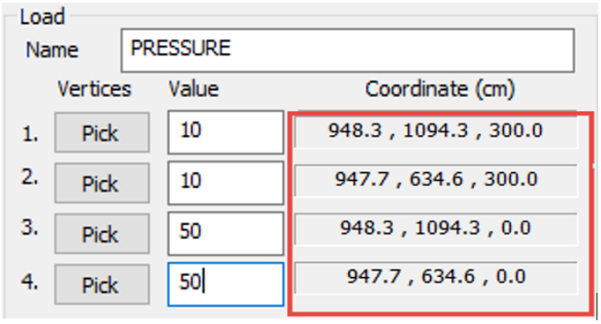
Point the four corners that define the surface by clicking successively the buttons Pick for each corner as shown in the image below.

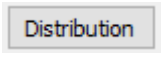


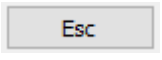


In this way, the coordinates for the four corners are automatically recognized and filled in.

Next, you set the pressure values (in kN/m^2) for each corner point

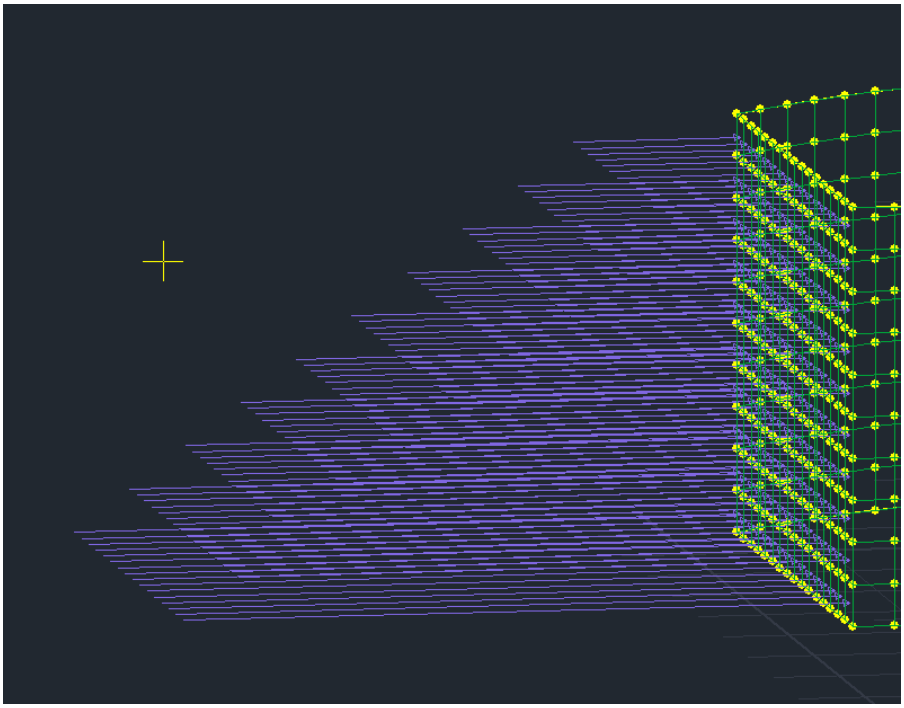


Finally, click the buttons 

and 

.

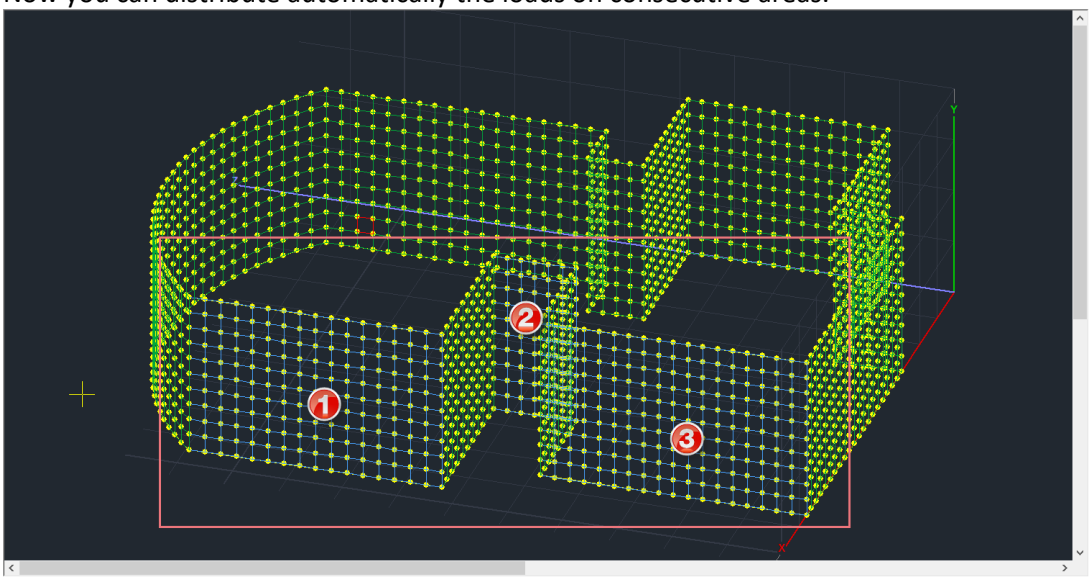
The load distribution on the selected surface is completed and is graphically represented by the elements of the current surface.



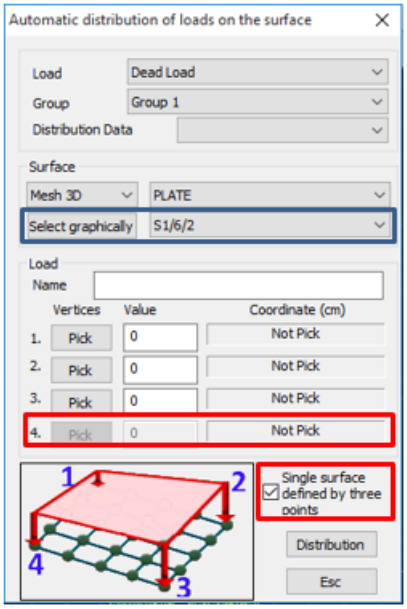
2. CONSECUTIVE SURFACES:

2nd EXAMPLE:

Now you can distribute automatically the loads on consecutive areas.



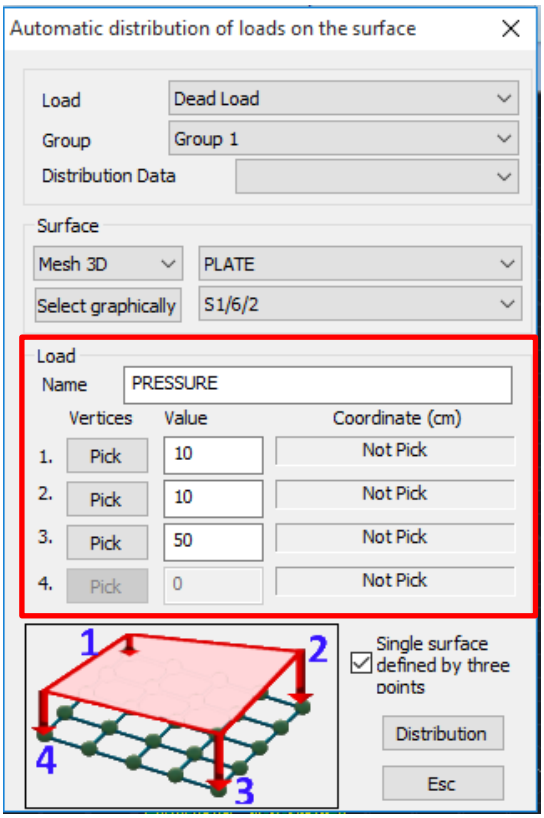
A similar process is followed by the below differences being the only ones:

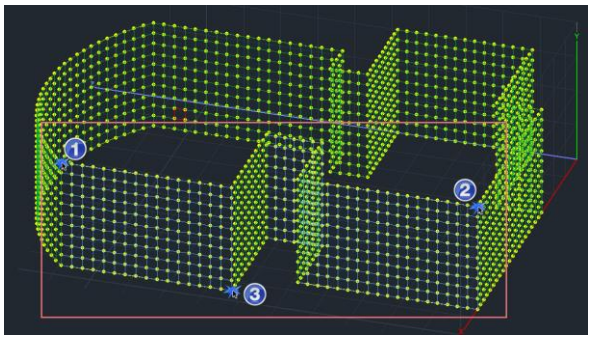


- Using the “Select graphically” command you only select one element form one of the surfaces that are going to be loaded.

- Check the command “Single surface defined by three points” and the 4th option is automatically disabled.

As previously described, using the Pick button, you point the 3 points that define the combined area.
Then specify the pressure values in kN / m2 for 3 points.

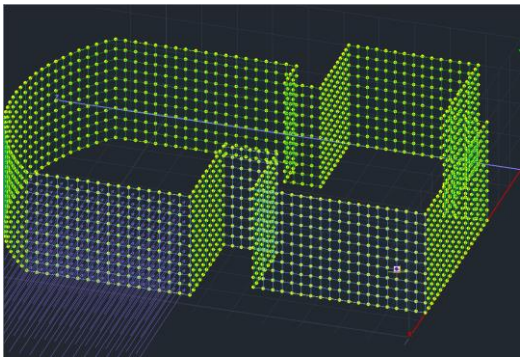


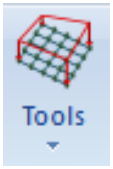


Finally, click the buttons Distribution and Esc.

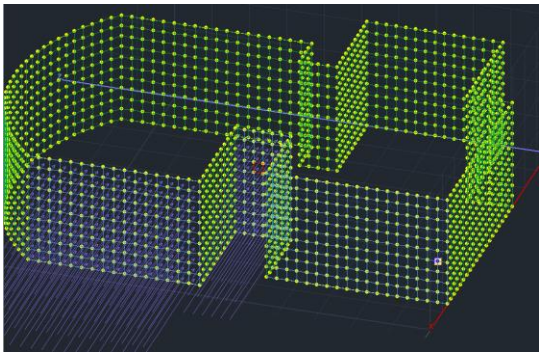
- The load distribution on the selected surface is completed and is graphically represented by the elements of the current combined surface.

Select graphically S1/6/2

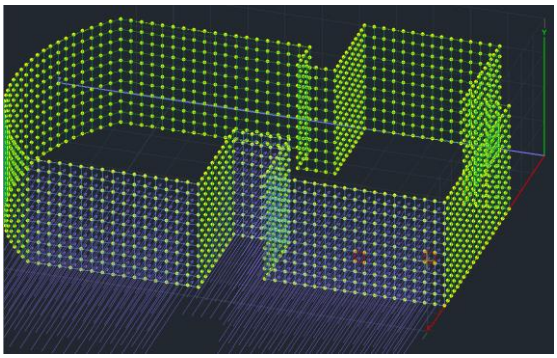


- To distribute the loads to the rest of the surfaces, select again the command 

and at the dialog box, click Select graphically and select an element from the next surface that is automatically recognized and selected from the list S1/7/2. Click the buttons Distribution and Esc.

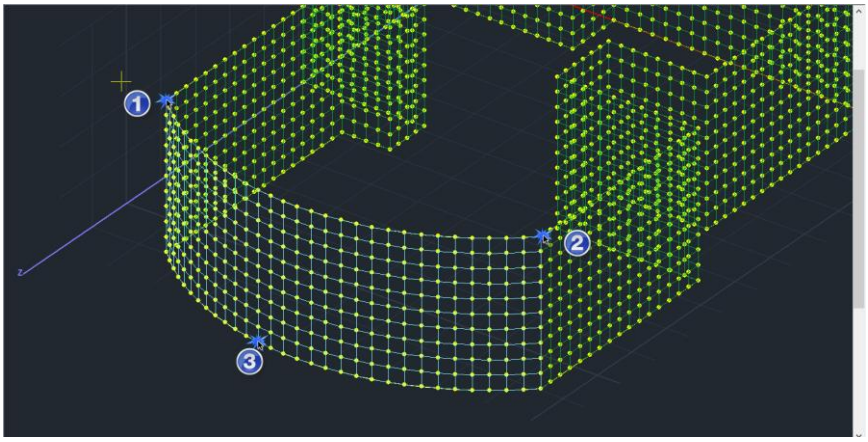


- Follow the same procedure for the third surface.

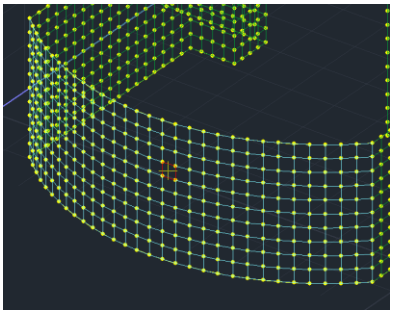


3. CURVED SURFACES:

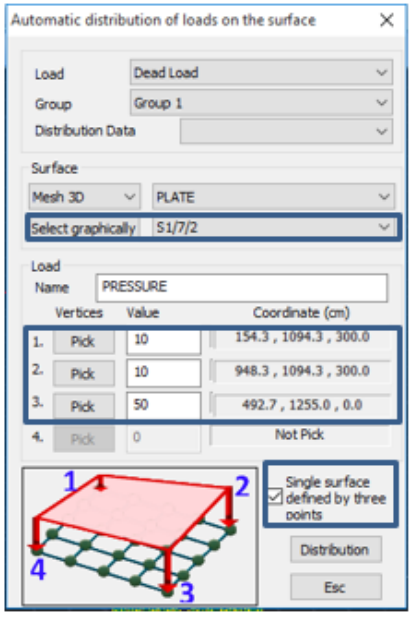
3rd EXAMPLE:



Follow the same procedure:

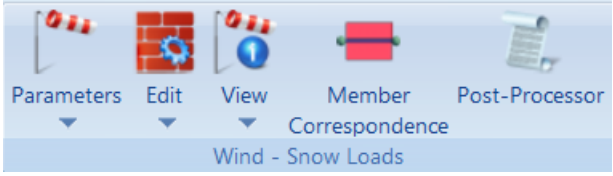


Graphical selection with one click.



Check the option "Single surface defined by three points" and the 4th option is automatically disabled. Define the surface by pointing to the three points that define the surface using the Pick buttons. Fill in the pressure values (in kN/m^2) and click Distribution and Esc.

4. Wind and Snow Loads



"Wind-Snow Loads" commands group contains tools for the automatic calculation of wind and snow loads and the distribution to members by Eurocode 1.

Also Greek, Italian, Germany, and Poland Eurocode 1 appendices, as well as the Italian NTC08 Regulation, are included.

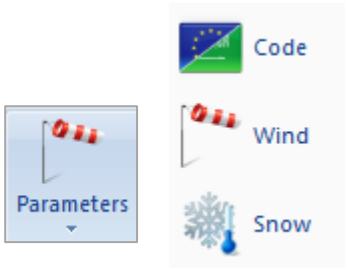
It is an extraordinary tool that includes:

- Automatic calculation of characteristic values of snow load on the ground and the roofs determined by EN 1990 for all types of the roof: flat, single, double, quadruple, vaulted, with proximity roof tallest building drift in protrusions and obstacles of the Roof shape coefficients automatic calculation.
- 2D and 3D display of snow load distribution.
- Basic wind velocity automatic calculation.
- Automatic calculation of average wind speed VM (z) at height z (according to soil roughness and orography)
- Categories and soil parameters
- Wind turbulence
- Max velocity
- Wind pressure distribution on surfaces
- Wind forces
- Pressure coefficients for buildings (vertical walls or roofs)

The procedure for calculating wind and snow loads and their distribution to members includes five groups of commands:

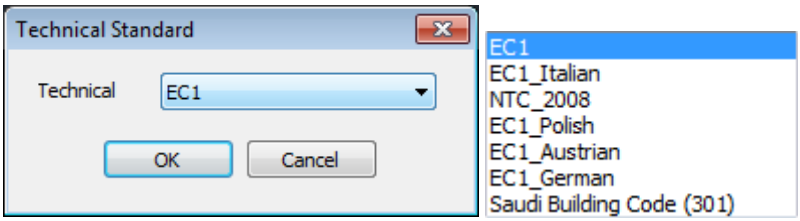
1. **Parameters**: Code selection, Wind-snow general parameters
2. **Edit**: wall-roof
3. **View**: wind-snow
4. **Member correspondence**
5. **Post-Processor**

4.1 Parameters



4.1.1 Code

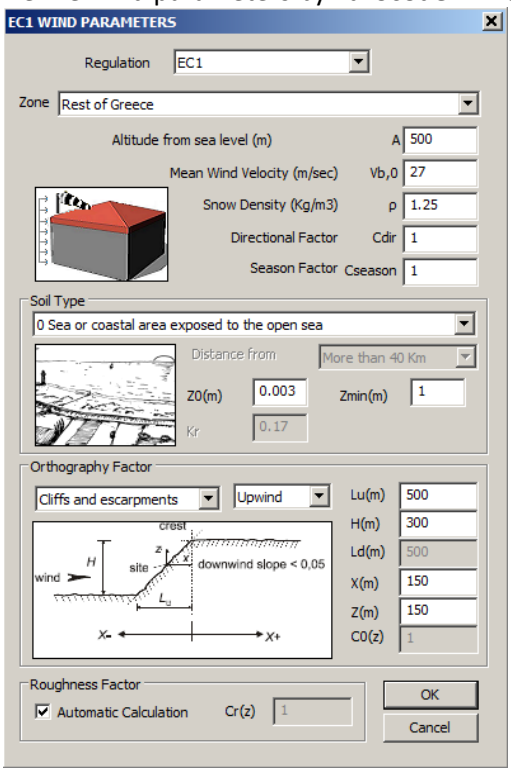
In the dialog box that appears

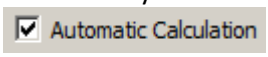


Select the code for wind and snow loads' calculation.

4.1.2 Wind

Define wind parameters by **Eurocode 1** in the dialog box:



Select from the list: “Regulation” and “Zone” and the respect fields are automatically updated.
In “Soil Type”: select type from the list, category, and distance from the coast.
In “Orthography Factor”: define topography and wind direction.
The other fields are updated automatically based on the previous selections.
In “Roughness Factor”: when 

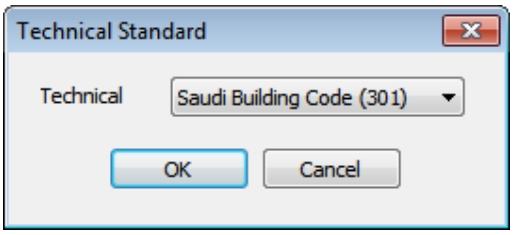
is activated, the program automatically calculates the Cr(z) value, otherwise type a value 

manually.

Press “OK” to save the parameters.

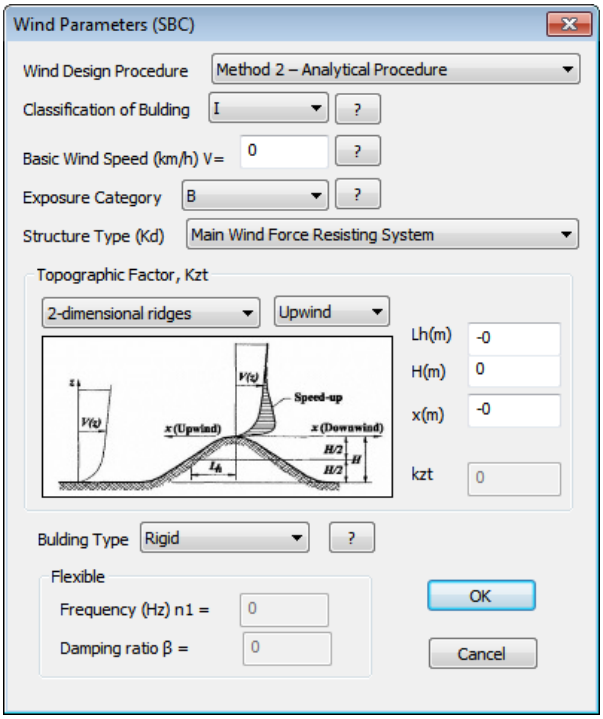
⚠ *The user can modify the calculated values. By typing different values in the fields, data is updated automatically.*


The latest version of SCADA Pro integrates the **Saudi Arabia code (SBC 301)** for wind loads as well.
In the following, a detailed description of the parameters choosing SBC 301 is described:



Wind :

By selecting “Wind” parameters the following dialog box appears:



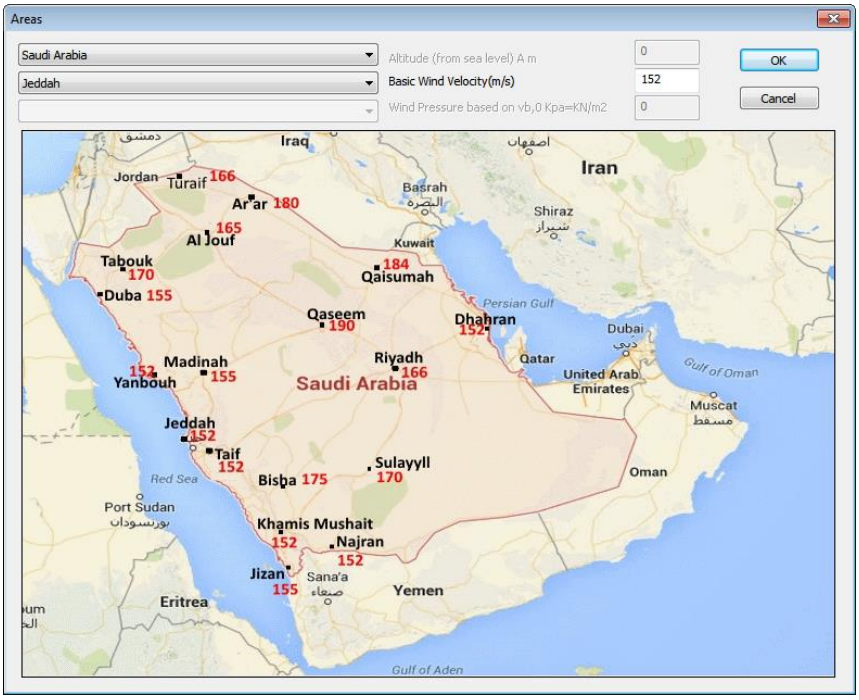
SBC 301 provides three methods for calculating wind loads (par. 6.1.2)

1. Simplified Procedure (Section 7.1)
2. Analytical Procedure (Section 7.2)
3. Wind Tunnel Procedure (Section 7.3)

SCADA Pro incorporates the first two methods (The third method is based on experimental measurements).

First, choose one of the two methods for the calculation of the wind loads Wind Design Procedure. The first method is applied only to buildings that meet specific criteria (par. 7.1.1).

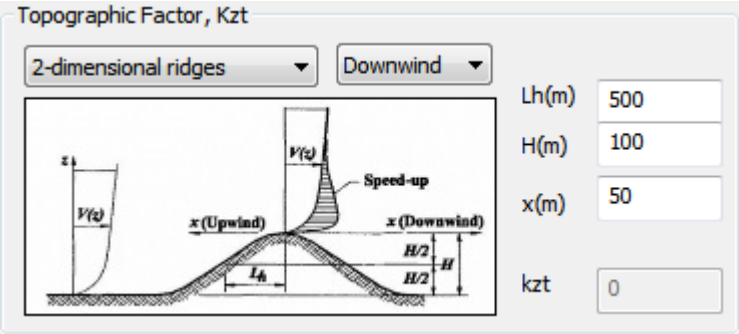
The second parameter Classification of Bulding regards the choice of the class of the building based on the Table 1.6-1. Press ? next to the parameter to open the corresponding table. Then define the Base Wind Speed/parameter based on the values of the map (FIGURE 6.4-1) that appears when pressing ?.



The parameter Exposure Category regards the choice of the class exposure of the building by the paragraph 6.4.2.2 & 6.4.2.3.

Structural Type selection Structure Type (Kd) (TABLE 6.4-1) regards the choice of the Kd (Directionality Factor).

The next parameter section



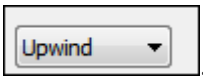
identifies the influence of the topographic configuration of the region where the structure is situated (FIGURE 6.4-2). According to these parameters, calculate the topography coefficient (Topographic Factor) Kzt.

The first selection regards the soil topology 2-dimensional ridges. There are five choices:

- 2-Dimensional Ridges
- 2-Dimensional Escarpments

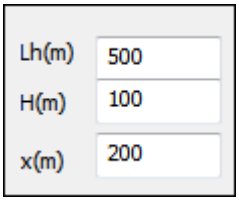
- 3-Dimensional Axisymmetric Hill
- Flat, unobstructed areas and water surfaces
- User

The 3rd choice sets Kzt=1.
On the 4th choice, the user defines a value to Kzt.

Select the position of the structure relative to the wind direction (Windward, Upwind or Leeward, Downwind) 

.

The next three fields

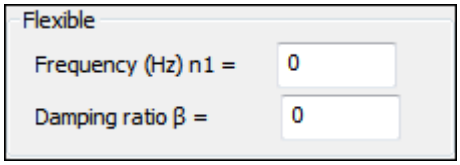


Are related to the topography of the area
Lh: Upwind distance of crest to where the difference in ground elevation is half the height of the hill or escarpment, in meters.
H: Height of hill or escarpment relative to the upwind terrain, in meters.
x: distance upwind or downwind of the crest as in Figure 6.4-2, in meters.

On the next field choose the type of the building based on the stiffness of the diaphragms. There are three choices:

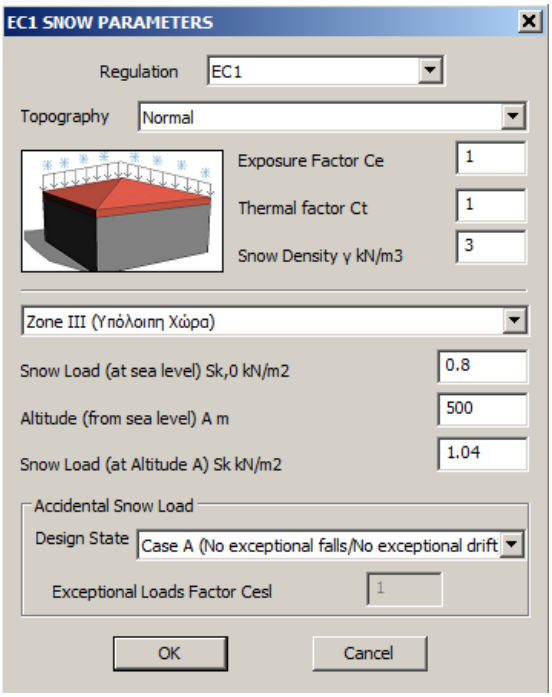
- Rigid
- Flexible
- Parapets

If the type of the building is Flexible you must also set the following two parameters:



4.1.3 Snow :

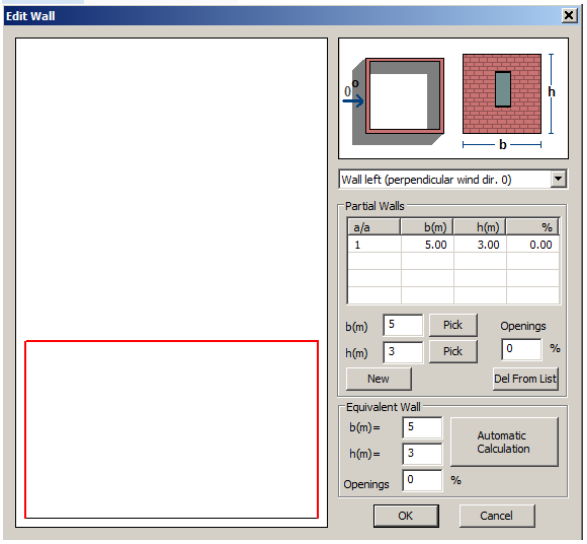
Define snow parameters according to **Eurocode 1** in the dialog box:



Select from the list the parameters: “Regulation”. “Topography” and “Zone” and the related fields are automatically updated.
In the “Accidental Snow Load”: select a condition.
Press “OK” to save the parameters.

4.2 Edit

4.2.1 Walls :



The take advantage of the "Templates" command, in which all the geometric characteristics of the walls are filled in automatically by the program, and save a lot of time and work!

Without using “TEMPLATES”

Select from the list the wall according to the wind direction (clockwise direction).
Press Pick next to b(m). To define the length of the selected wall, left click on start and end points.
Press Pick next to h(m). To define the width of the selected wall, left click on start and end points.

“Partial Walls” list is filled in automatically.

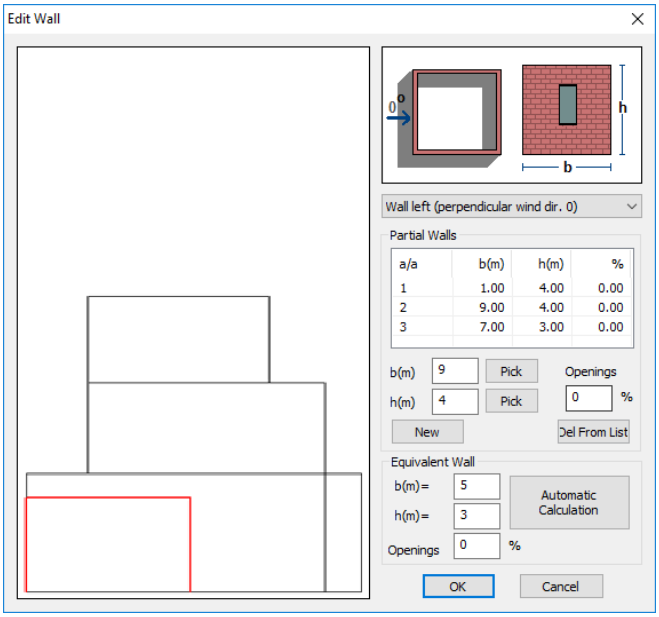
Then type/openings percentage (for doors and windows) and press Automatic Calculation.

The program calculates automatically the "Equivalent Wall."
Press “OK” command to save the parameters.
Repeat the procedure for all four directions of the walls.

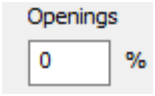
NOTE:

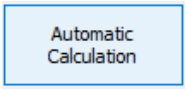
⚠ *The height of the lower wall is always defined starting from level 0 even if the steel structure begins at a higher level.*

⚠ *If the front view consists of several walls at one or more levels, press the button "New" and repeat the above procedure to set the whole face.*



In above table fill in the geometrical characteristics of the "Sub-walls".

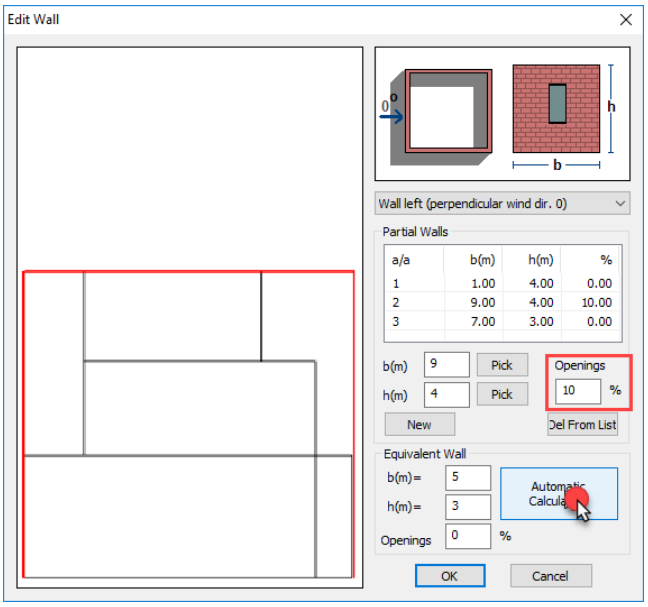
Finally type the percentage of openings 

for each direction and press, every time, the button 

.

The program calculates automatically the “Equivalent Wall”.

⚠ *The red rectangle should circumscribe all the front view.*

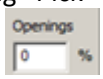


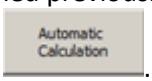
Press “OK” to save the parameters.
Repeat for all four directions of the walls.

Using “TEMPLATES”

By using "TEMPLATES" tool, the user saves a lot of time and work because the geometric characteristics of the walls are updated automatically by the program.

Select from the list the wall according to the wind direction.
“Partial Walls” list is filled in automatically, without using “Pick” as mentioned previously.

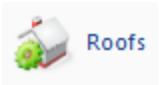
The user needs only to type in the openings percentage 

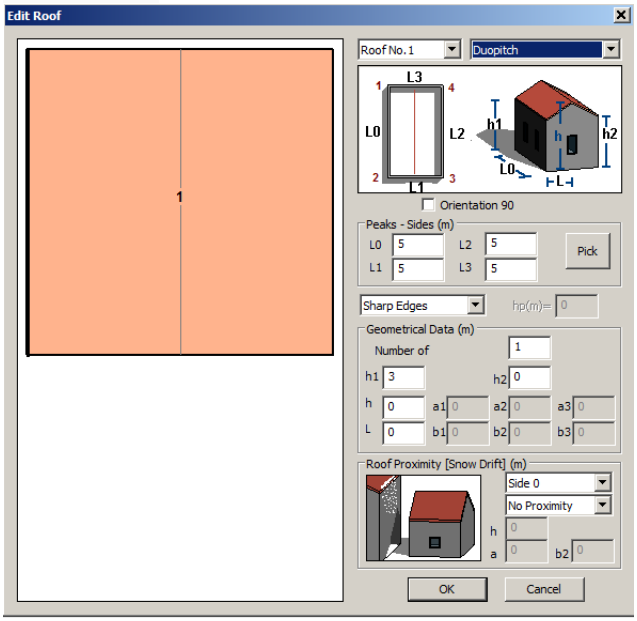
and press 

.
The program calculates automatically the "Equivalent Wall."
Press “OK” to save the parameters.
Repeat the procedure for all four directions of the walls.

Edit

4.2.2 Roofs



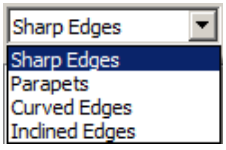


Without using "TEMPLATES"

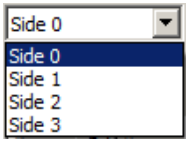
Select from the lists the roof number and the form.

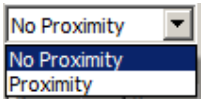
Press 

next to "Peaks-Sides". To define the geometry of the roof, left click on the four peaks of the floor plan of the roof and the cells will be filled in automatically.

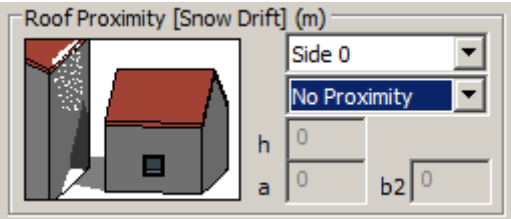
Select from the lists 

and type in the height of the barrier in m.
In "Geometrical Data" type in the number of frames and the other geometrical data in m.

Roof Proximity
If the structure is close to another taller building, in the "Roof Proximity" select the side which shares the boundary 

and from the list 

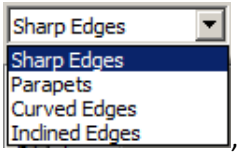
the "Proximity".
The field changes according to the proximity and the side. Type in the geometry data and



Press “OK” to save the parameters.
Repeat the procedure for all four directions of the roof (clockwise direction).

Using “TEMPLATES”

Select from the lists the roof number and the form.
“Geometrical Data” fields are filled in automatically.

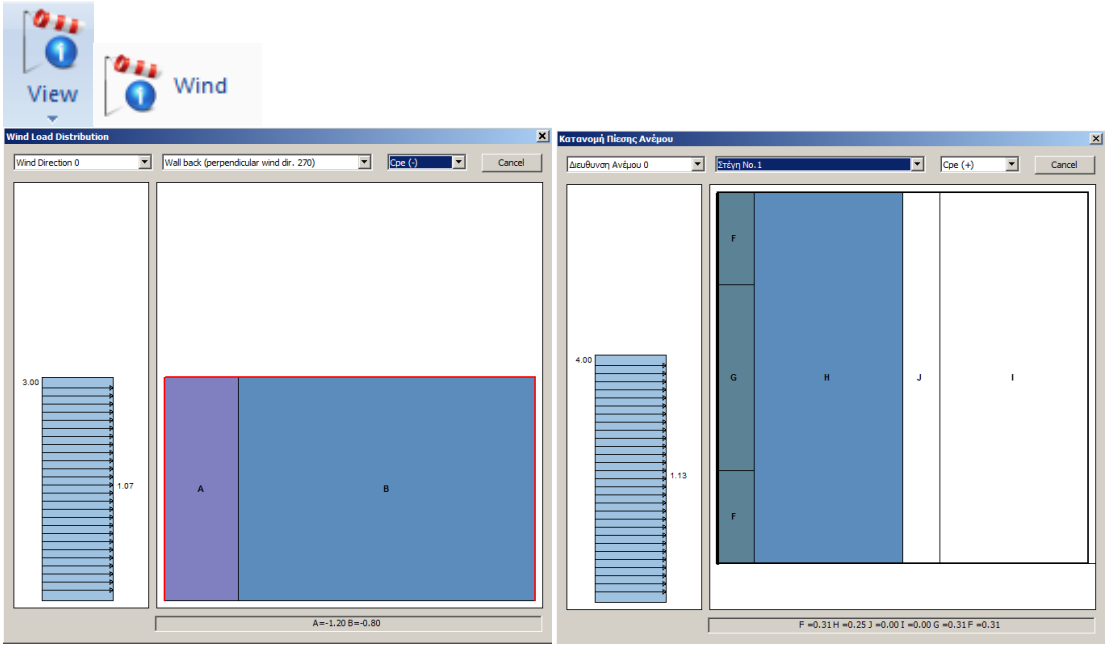


The user only needs to select , and type in the height of the barrier in m and define the "Roof Proximity" as mentioned previously.
Press “OK” to save the parameters.
Repeat the procedure for all four directions of the roof (clockwise direction).

4.3 View

4.3.1 Wind

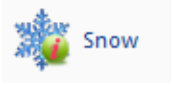
Select the command to see the wind pressure distribution on the walls and the roofs of the building. In the dialog box, select the wind direction, the wall or the roof and the type of pressure. The distribution is automatically displayed with colors. The zones with different pressure have a different color.

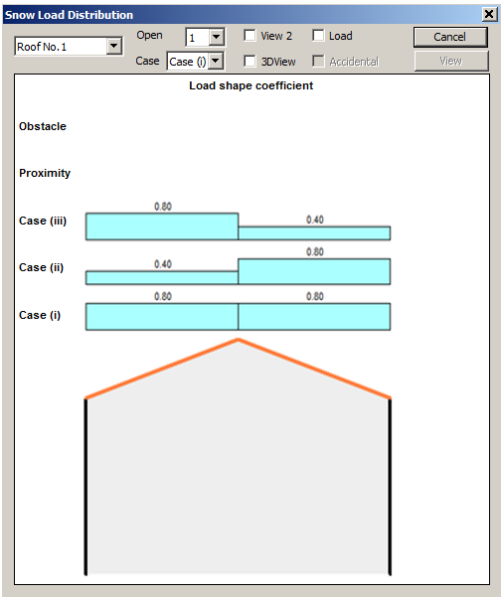


View

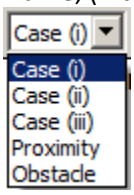
4.3.2 Snow

Select the command to see the snow distribution on the roofs.



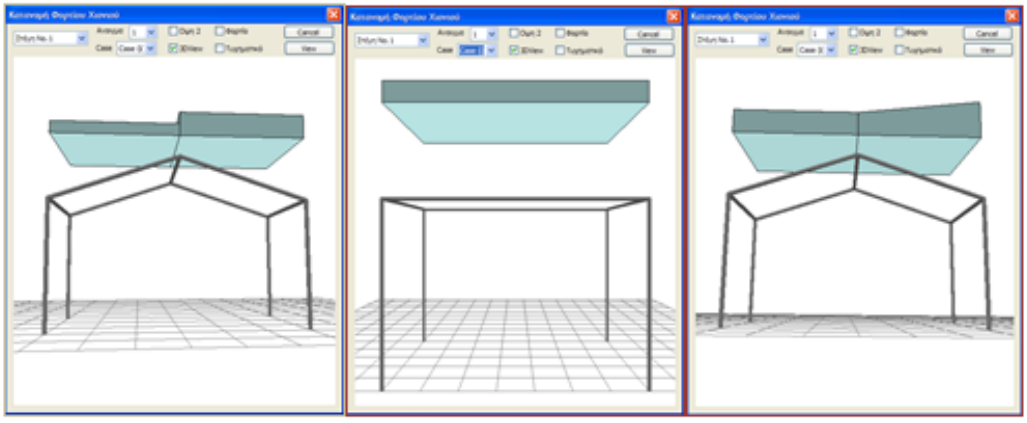


In the dialog box, select from the list the number of "roof" of the "opening" i.e. the number of the frame, (if there are more than one), and "Case"

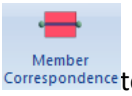


for the load distribution of snow.

Activate “Load” checkbox to display the values and “3DView” checkbox to receive snow distribution as is displayed in the following pictures.



4.4 Member Correspondence



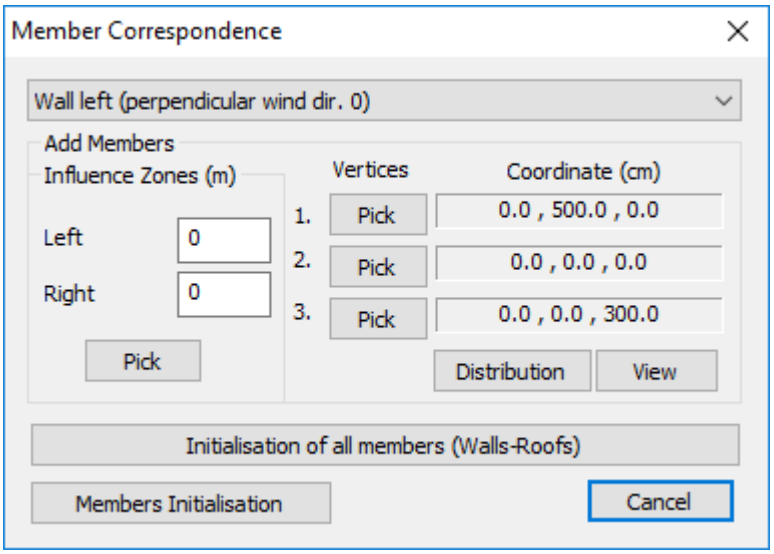
to assign the calculated loads to the members, through the influence zones.
Select the command and in the dialog box: select a wall, or a roof and define the dimension of the influence zones.

In the new version of SCADA Pro, the automatic calculation of influence zones for linear members to make the distribution of wind and snow loads is completed and integrated as well.

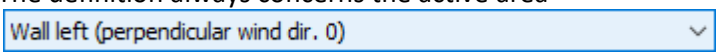
⚠ *We remind you that until now the automatic distribution was possible only for structures derived from Templates. Now you have the opportunity to make this distribution on any surface.*

Let's see in detail the **manual**, **semiautomatic** and **automatic** distribution:

By selecting the command the following dialog box now opens



Concerning the old definition of the influence zones nothing changed, while a new part to define the area with three points was added to the right.
The definition always concerns the active area

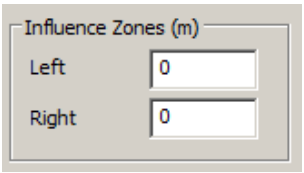


*It is better to start either the **manual** or **semi-automatic** procedure by pressing the "Members Initialization" button.*

Attention:

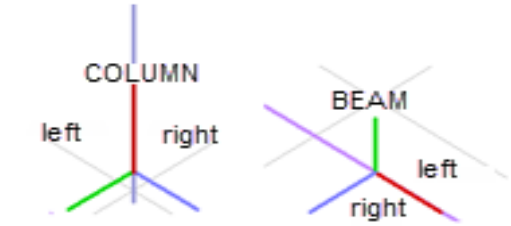
⚠ *In the **automatic** procedure coming from the TEMPLATES, DO NOT press "Members Initialization" button, because it will delete the automatic load distribution to members!!!*

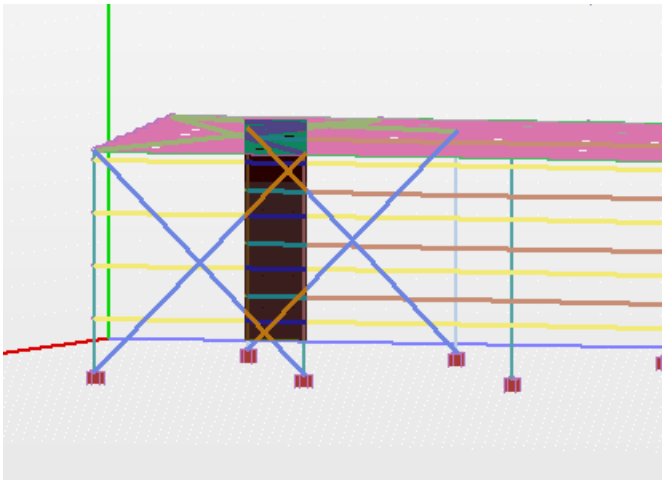
- **Manual Procedure - Without using "TEMPLATES"**



In define the influence zones of a member by typing the corresponding widths in m, on the left and right of that, "Pick" and left click on the member (or parts of the member). The "Influence Zone" is displayed as in the figure below.

⚠ *"Left" and "Right" are determined based on the local axis x (red).*





- **Semi-automatic Procedure - <u>Without</u> using “TEMPLATES”**

A new part to define the area with three points is added to the right. The definition always concerns the active area:

Wall left (perpendicular wind dir. 0)

It is better to start the procedure by pressing the "Members Initialization” button.

Indicate the point graphically along with the following particularity:

- The first two points define the direction by which the automatic calculation of influence surfaces is made for items which are parallel to this direction.
  Note also that the distribution will be for all linear members belonging to this level and are parallel to the first direction.
- Since you define the three points, press the "Distribution" button and the program automatically makes the distribution and displays it.

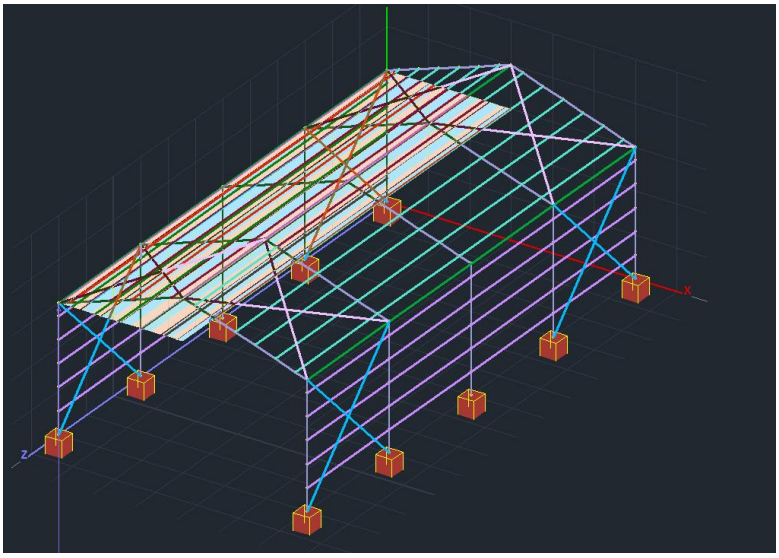
Respectively the definition is made for the other walls.

Concerning the roofs, the definition can be made sequentially.
First, select the roof

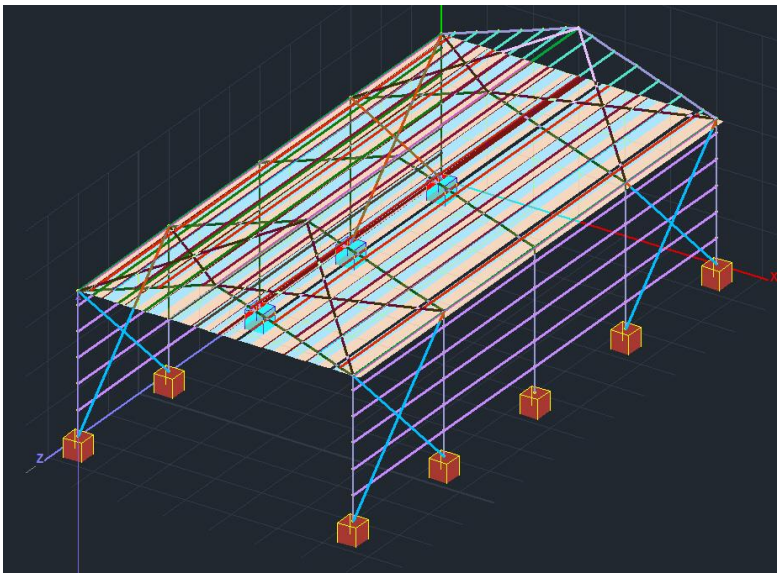
Roof No. 1

, then you must define the individual areas.

EXAMPLE:
For example, first, define the left slope indicating graphically the three points



and then, the right ones. The overall result is the following:

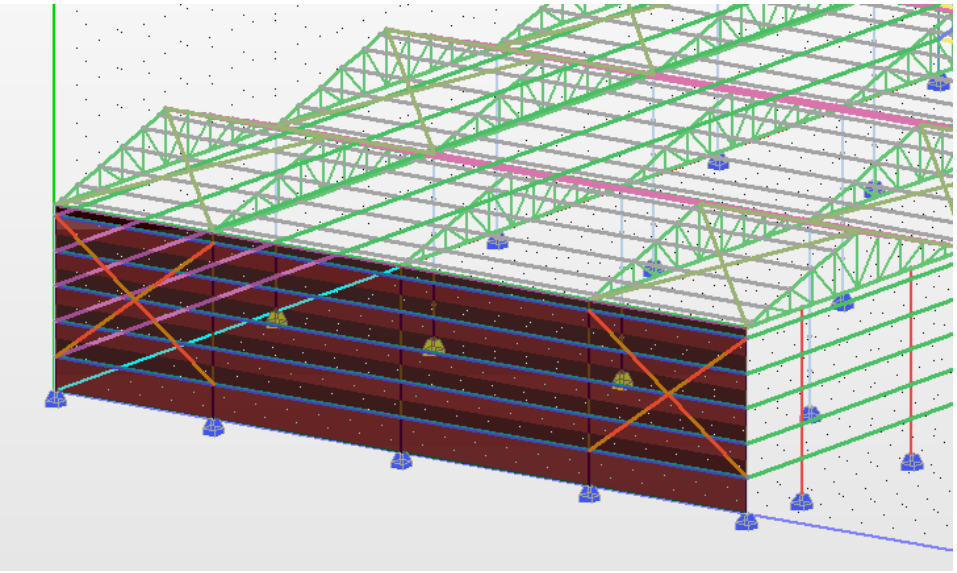


⚠ Finally, it is worth noting that if the walls are properly defined there is NO need for more definition. Just select each wall and press «Distribution». The distribution is made and is displayed simultaneously on the linear members belonging to this wall.

⚠ Same for the flat roofs only.

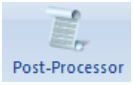
- **Automatic Procedure - Using “TEMPLATES”**

By activating “Purlins” and “Girders” in “Load Attribution” of “Templates”/, just select "Pick" and the program automatically calculates the influence zones distributing the pressure in all purlins and girders.

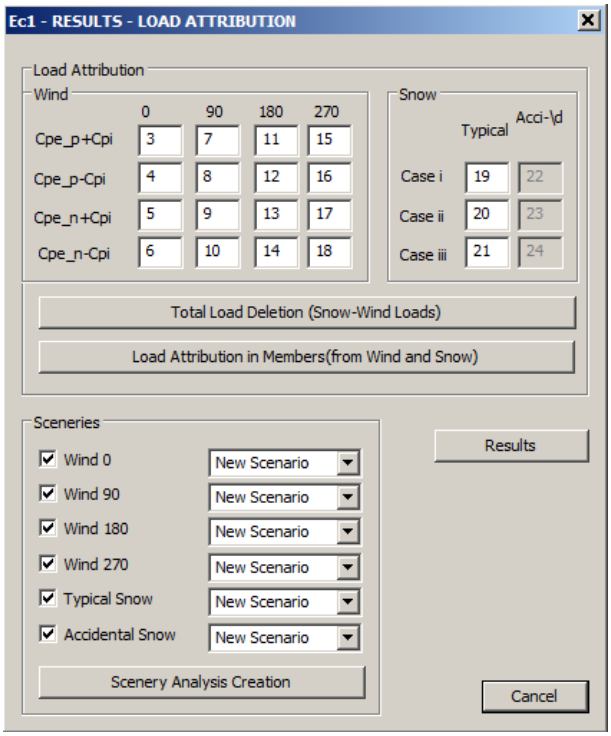


4.5 Post-Processor

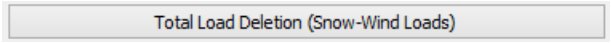
The last command is “Post-Processor”.



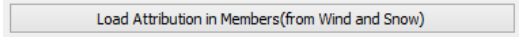
In the dialog box, in “Load Attribution” there are two labels:
one with the wind loads, 4 cases for four directions, i.e. 12 cases to each load and one with snow loads, 3 cases for typical snow.



The numbers that appear on labels correspond to the Load Cases serial numbers.
Remember:
Load Case1: Dead
Load Case2: Live
and now there are 16 new Load Cases for the Wind (from 3 to 18) and 3 for the Snow (19, 20 and 21).
Select

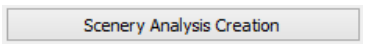


to attribute wind and snow loads on the structure members, or



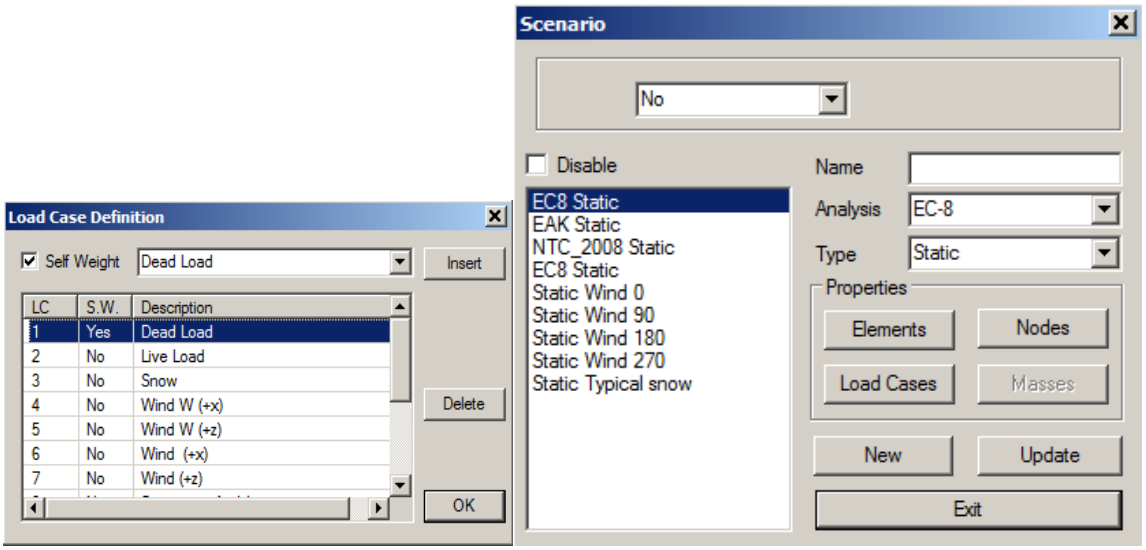
to delete them all.

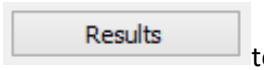
In “Scenarios” there is a list with the analysis scenarios created automatically by selecting the



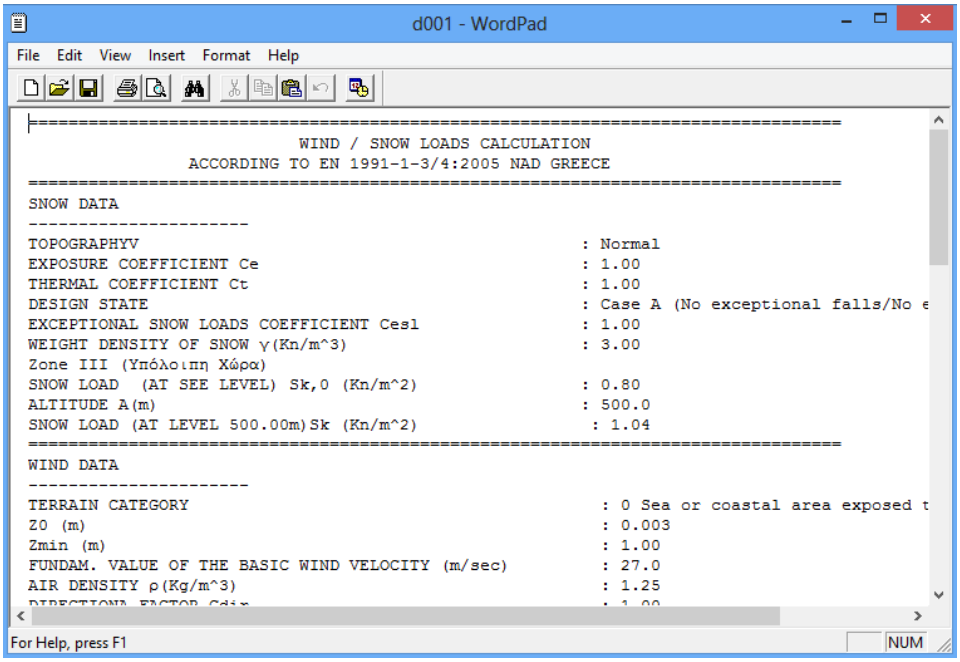
command.

SCADA Pro not only calculates automatically the load distribution of wind and snow, but it also creates automatically all the analysis scenarios too, saving the user from hard work and much time.



Press 

to open the txt results file, containing in detail all data and calculations derived from all Eurocode 1 procedures1.



Choosing the SBC 301 Regulation, the printing is as follows:

WIND LOADS CALCULATION ACCORDING TO SBC 301 CHAPTER 7

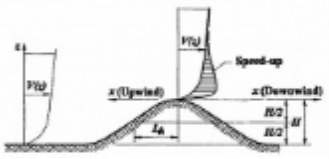
Design Method : Method 2 – Analytical Procedure

Basic Wind Parameters

DESCRIPTION	SYMBOL	UNITS	VALUE	REFERENCE
Classification of Building			I	Table 1.6-1
Importance Factor	I		0.77	Table 6.5-1
Country				Figure 6.4-1
City				Figure 6.4-1
Basic Wind Speed	V	(km/h)	165.00	Figure 6.4-1
Exposure Category			B	6.4.2.3
Structure Type			Main Wind Force Resisting System	Table 6.4-1
Wind Directionality Factor	Kd		0.85	Table 6.4-1

Topographic Factor (Kzt) Calculation (Figure 6.4-2)

DESCRIPTION	SYMBOL	UNITS	VALUE
Topography			2-dimensional ridges
Hill Height	H	(m)	-500.00
Half Hill Length	Lh	(m)	100.00
Distance from top of crest	x	(m)	-100.00
Building up/down wind			Upwind
Topographic Factor	Kzt		1.49



Building Type : Rigid | **Frequency (Hz) n1 =** | **Damping Ratio β =**

Gust Effect Factor (G) Calculation (7.2.7) Rigid Structure (7.2.7.1)

DESCRIPTION	SYMBOL	UNITS	WALL LEFT (w0)	WALL FRONT (w90)	WALL RIGHT (w180)	WALL BACK (w270)	REFERENCE
Mean height	h	(m)	4.00	4.00	4.00	4.00	
Width	B	(m)	6.00	6.00	6.00	6.00	
Equivalent height	/z	(m)	10.00	10.00	10.00	10.00	7.2.7.1
Constant	/ε		0.33	0.33	0.33	0.33	Table 7.2-1
Constant	l	(m)	100.00	100.00	100.00	100.00	Table 7.2-1
Minimum height	zmin	(m)	10.00	10.00	10.00	10.00	Table 7.2-1
Constant	c		0.30	0.30	0.30	0.30	Table 7.2-1
Constants	gq, gv		3.40/3.40	3.40/3.40	3.40/3.40	3.40/3.40	
Integral length scale	Lz	(m)	100.00	100.00	100.00	100.00	Table 6.4-1
Intensity of turbulence	Iz		0.30	0.30	0.30	0.30	Eq. 7.2-2
Background response	Q		0.93	0.93	0.93	0.93	Eq. 7.2-3
Gust Effect Factor	G		1.00	1.00	1.00	1.00	Eq. 7.2-1

WIND LOADS CALCULATION
ACCORDING TO SBC 301 CHAPTER 7

WALLS DATA

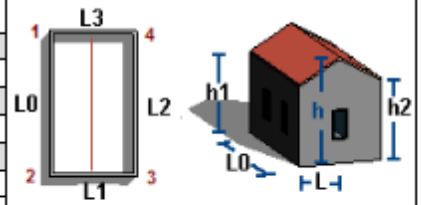
		Individual Walls						Equivalent Walls					
	Num	Section	Length b (m)	Height h (m)	Ag (m2)	Open (%)	Ao (m2)	Length b (m)	Height h (m)	Ag (m2)	Open (%)	Ao (m2)	Ao>0.8*Ag
Wall Left	W1	1	6.00	3.00	18.00	0.00	0.00	6.00	3.00	18.00	0.00	0.00	No
Wall Front	W2	1	6.00	4.00	24.00	0.00	0.00	6.00	4.00	24.00	0.00	0.00	No
Wall Right	W3	1	6.00	3.00	18.00	0.00	0.00	6.00	3.00	18.00	0.00	0.00	No
Wall Back	W4	1	6.00	4.00	24.00	0.00	0.00	6.00	4.00	24.00	0.00	0.00	No
Totals					84.00		0.00			84.00		0.00	

BUILDING IS QUALIFIED AS OPEN — No

ROOFS DATA

Number	Roof Type	L0 (m)	L1 (m)	L2 (m)	L3 (m)	Type of Roof Edge	Repetitions	h1 (m)	h2 (m)	h (m)	L (m)
1	Duopitch	6.00	6.00	6.00	6.00	Sharp Edges	1	3.00	3.00	4.00	3.00

Number	a0 (m)	a1 (m)	a2 (m)	a3 (m)	b0 (m)	b1 (m)	b2 (m)	b3 (m)	Ag (m2)
1									36.00
Total									36.00



Enclosure Classification (Section 6.2)

Number	Wall	Reference Area 0.4 (m2)	Reference Area 1%*Ag (m2)	Reference Area Value (m2)	Aoi (m2)	Agi (m2)	Aoi/Agi <=0.20	Ag (m2)	Ao (m2)	Ao> Ref. Area	Ao>1.1*Aoi
1	Wall Left	0.4	0.180	0.180	0.000	102.000	Yes	18.000	0.000	No	No
2	Wall Front	0.4	0.240	0.240	0.000	96.000	Yes	24.000	0.000	No	No
3	Wall Right	0.4	0.180	0.180	0.000	102.000	Yes	18.000	0.000	No	No
4	Wall Back	0.4	0.240	0.240	0.000	96.000	Yes	24.000	0.000	No	No

BUILDING IS QUALIFIED AS PARTIALLY ENCLOSED | BUILDING IS QUALIFIED AS ENCLOSED ✓

LOW-RISE BUILDING ✓