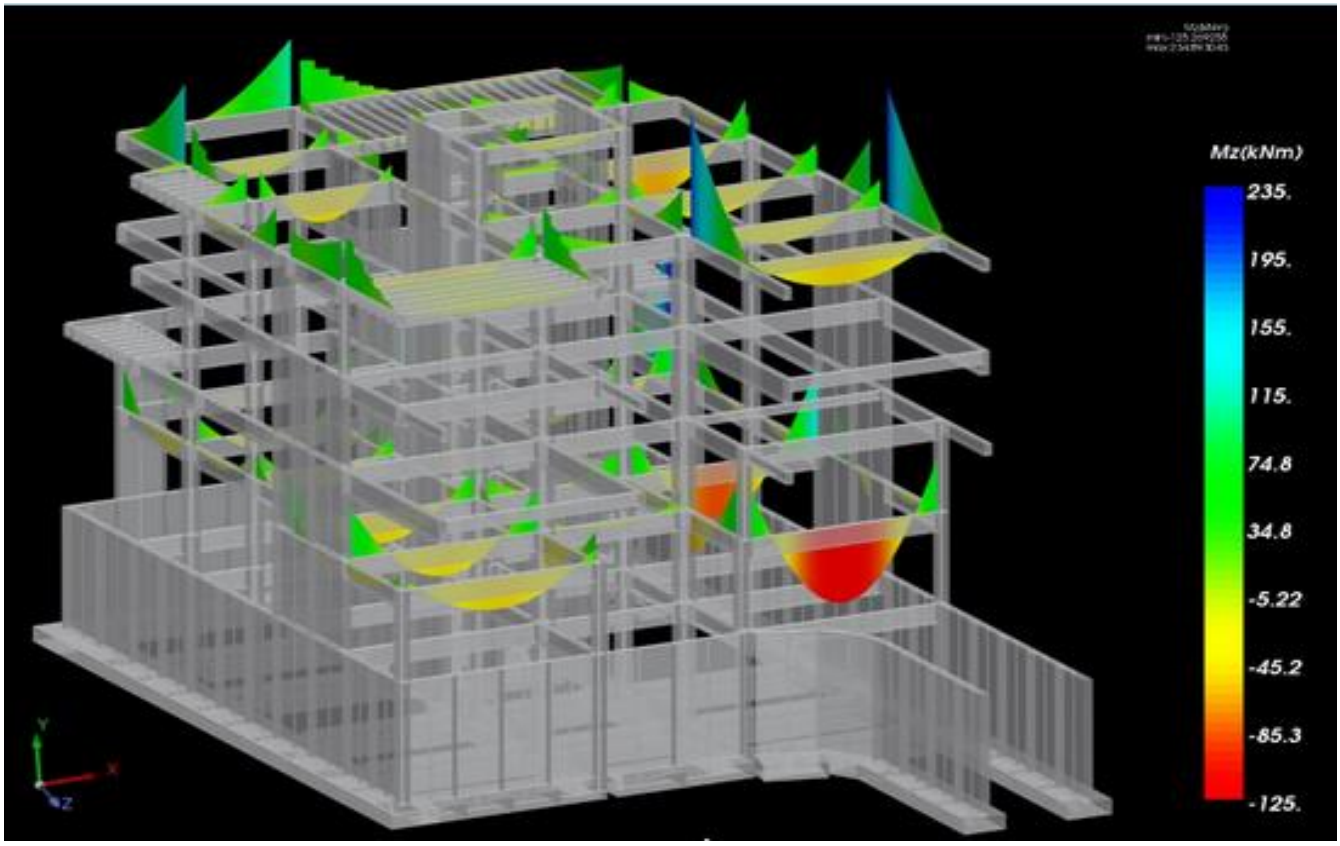
SCADA Pro™
Structural Analysis & Design

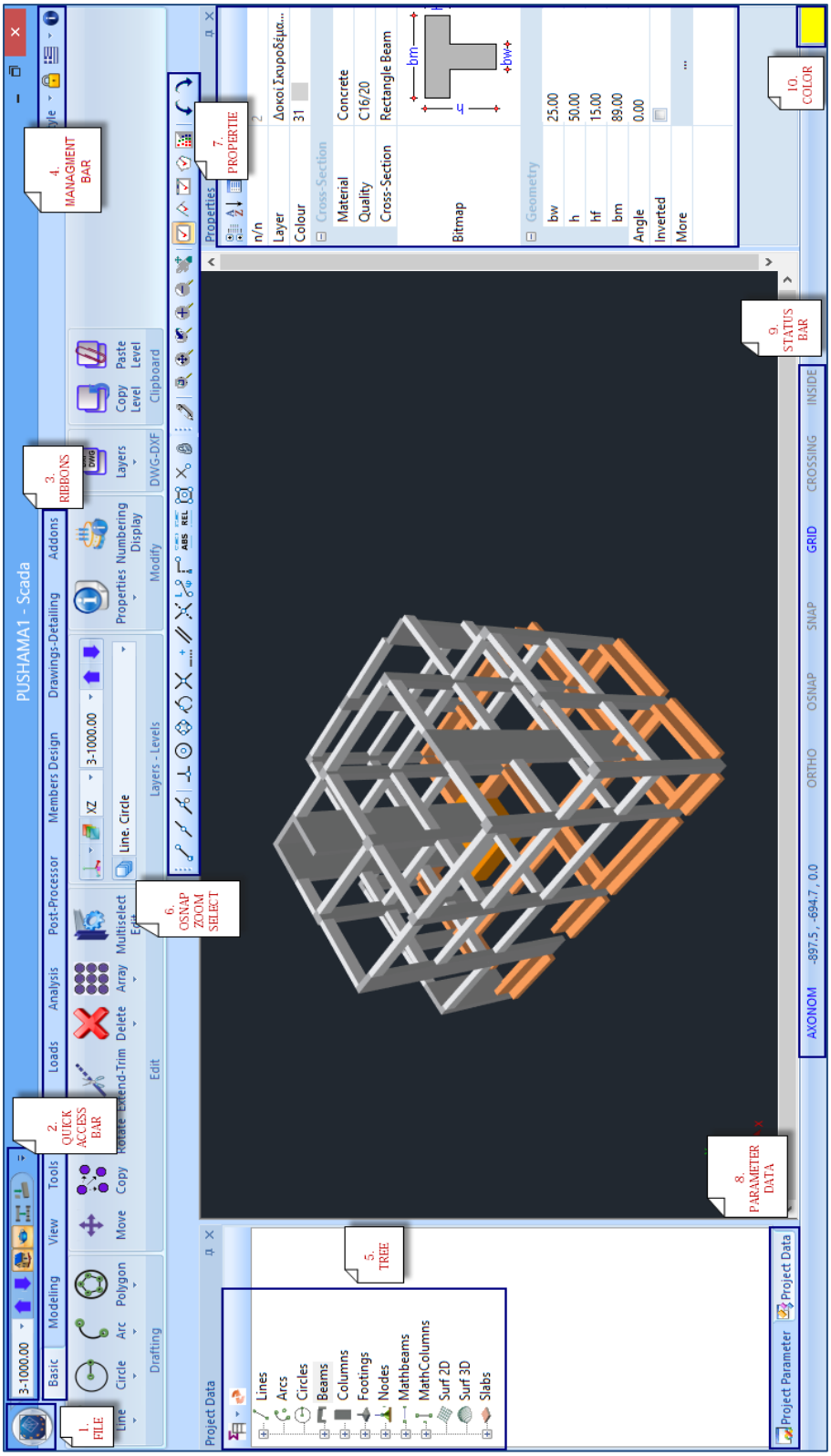
User's Manual
8.POST-PROCESSOR



ACE HELLAS
INTEGRATED SOLUTIONS

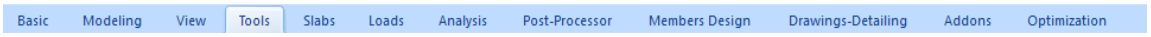
CONTENTS

I. THE NEW UPGRADED INTERFACE of SCADA Pro

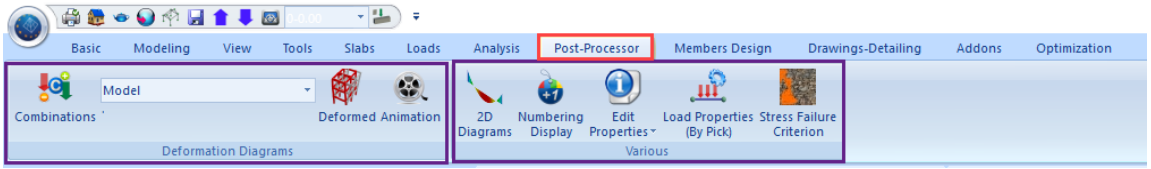


II. DETAILED DESCRIPTION OF THE NEW INTERFACE

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



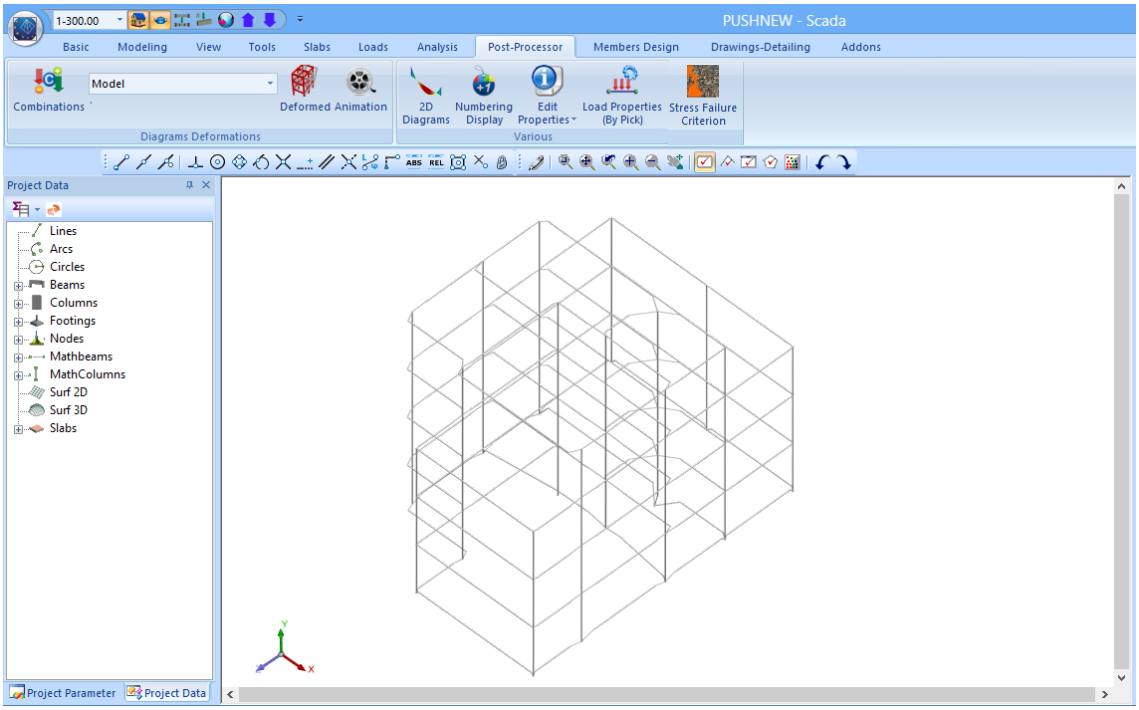
Post-Processor



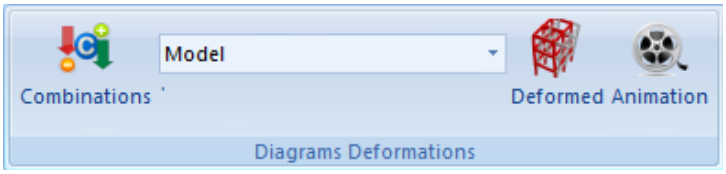
The 8th Unit called "Post-Processor" includes the following two groups of commands:

1. **Diagrams Deformations**
2. **Various**

“Post-Processor” offers the user a detailed observation of the internal forces, the diagrams (M, V, N) and the deformed shape of the model as a result of an individual load or load combination.

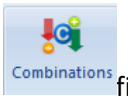


1. Diagrams / Deformations

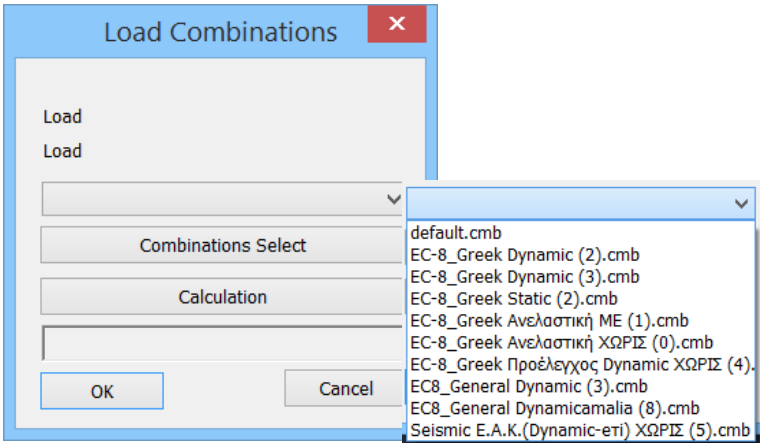


“Diagram Deformations” command group allows you to observe structure’s deformation for each load or load combination and receive the intensive forces M, V, N diagrams on each element.

1.1 Combinations



first, select “Combinations” and load a combination’s file, depending on the results you want to see. In the dialog box:



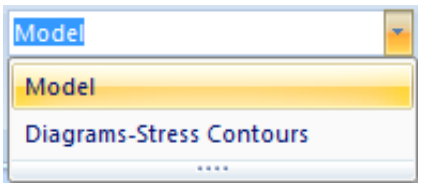
- Choose a combination from the list that includes the combinations of all the analyses that have been performed, and wait so as the calculation is completed automatically, or

- press “Combinations Select”, select the combinations file from the correspondent folder and press "Calculation".

⚠ *To see the deformed shape of the corresponding eigenvalues, choose a dynamic scenario .cmb file.*

1.2 List

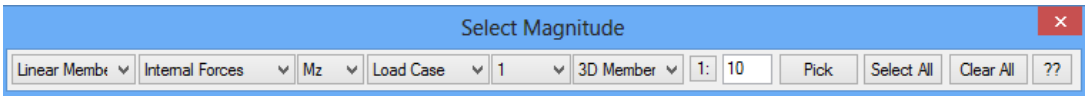
It contains the "Model" and



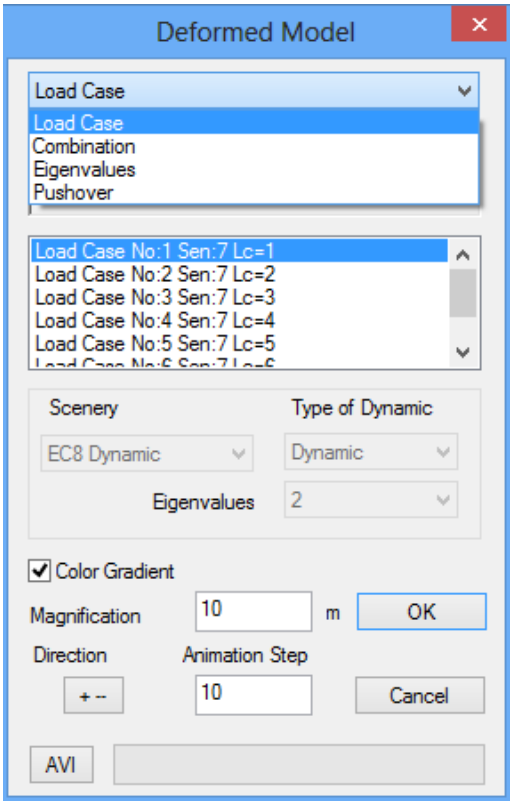
Select Model or Diagrams-Stress Contours

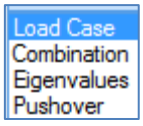
The commands are combined according to the choice:

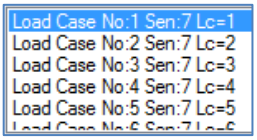
Model + Diagrams – Stress Contours +

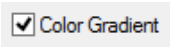


1.3 Model + "Deformed"



Choose from the list 

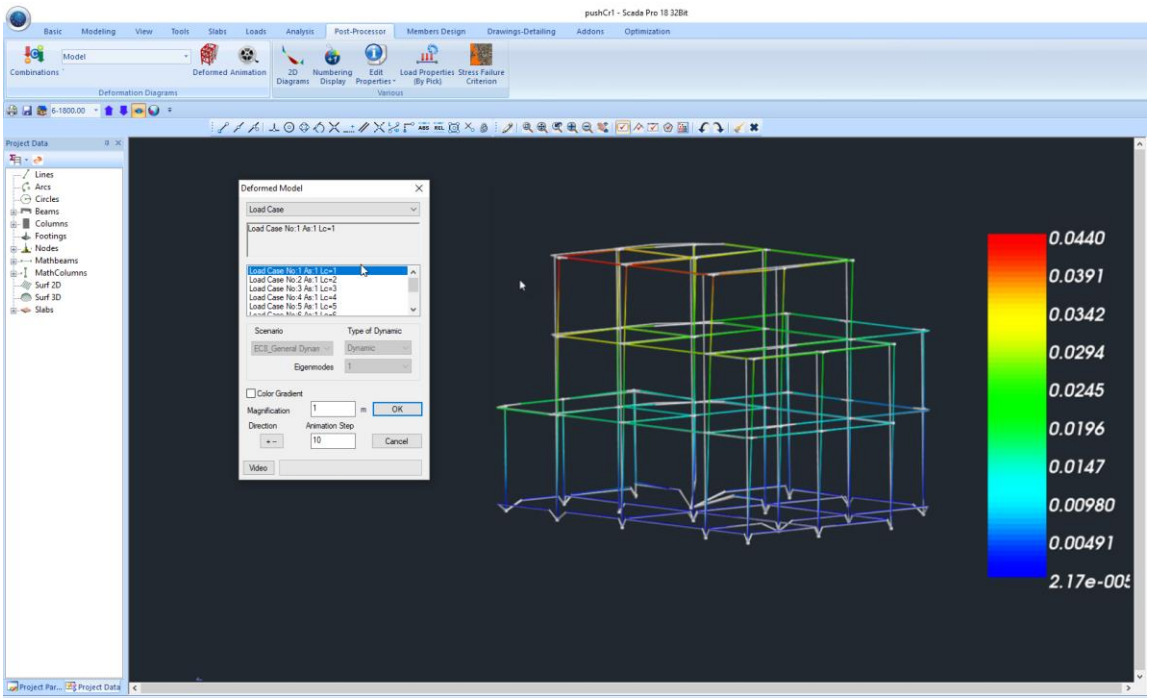
the general deformation cause and the next list, a general cause subcase. 

Activate 

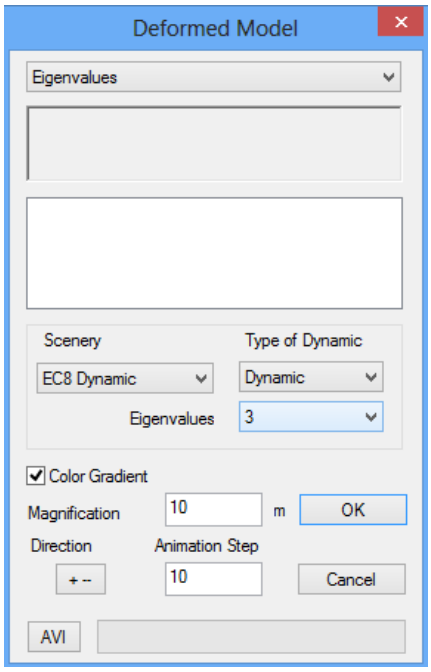
, modify "Magnification" and type in the value of the "Animation Step" to receive a better visualization.

"AVI" button gives the possibility to register a short video with the deformed shape of the structure.

It is now possible to display the deformation values based on the color gradation.



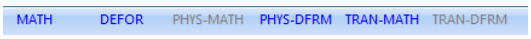
- “Deformed Model” window remains on the screen where you can select the next deformation origin. Press Cancel to close the window.
- According to the selected .cmb file, you can see the corresponding deformation
- Checking the model’s deformation helps you to understand the structural behavior of the building and sometimes to assess if any errors in the structural model leading to an unjustified structural behavior are located.
- By loading *Static combination* you can’t see Eigenvalues’ deformed shape.
- Static analysis scenarios produce deformation for each Load case or Combination.



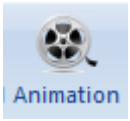
To receive Eigenvalues deformed shape (you must have first operated a dynamic analysis which creates a dynamic scenario), select the Dynamic combination file.

Select “Eigenvalues”, then the corresponding “Scenarios”, the Type and the number of the Eigenvalues.

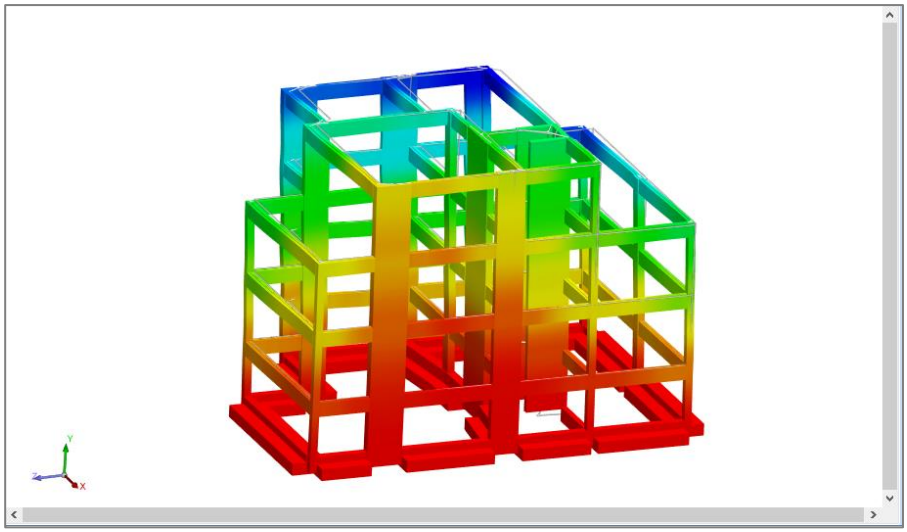
On the “Status Bar” check (double click, blue=active, grey=inactive) the type of the visualization of the deformed model.



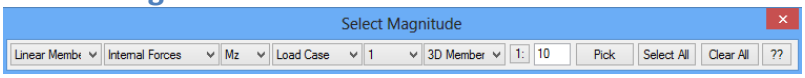
1.4 Model + "Animation"



"Animation" command is a button that activates and deactivates the deformed structure animation, according to the selections made in the "Deformed Model" dialog box.

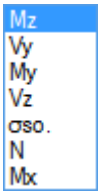


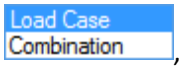
1.5 Diagrams – Stress Contours +

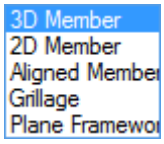


§ Linear Members

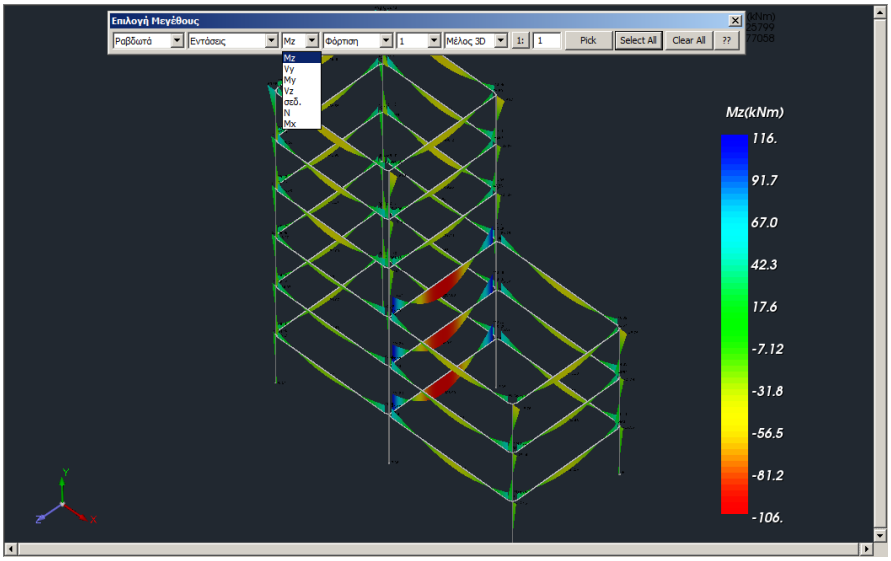
For Linear Members you can see:

tensions 

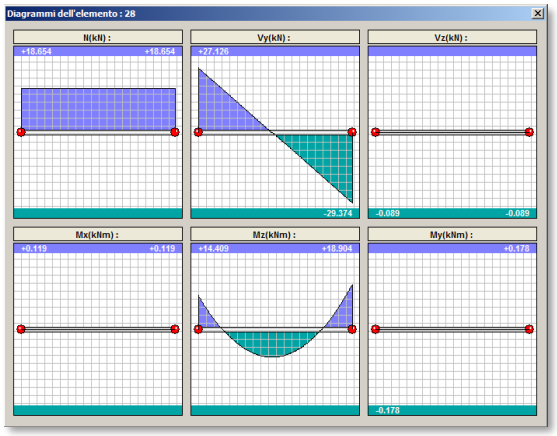
, for each 

, on a 

, in scale 



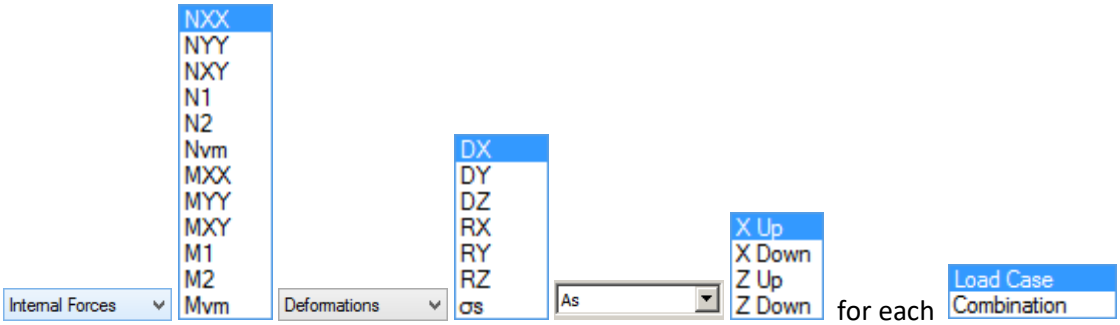
Select the “2D Member” view to see the six internal forces of a linear member concentrated in one window. Also, while moving the mouse you can see the values for each stress along the member.

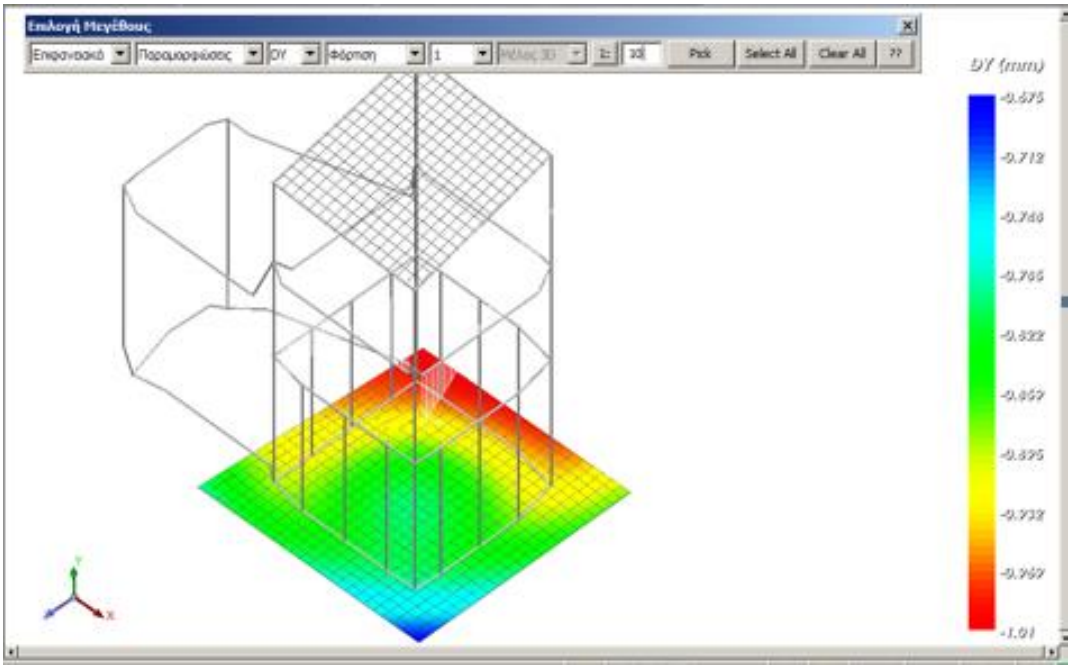


⚠ *Sign convention is done is according to the member’s local axes.*

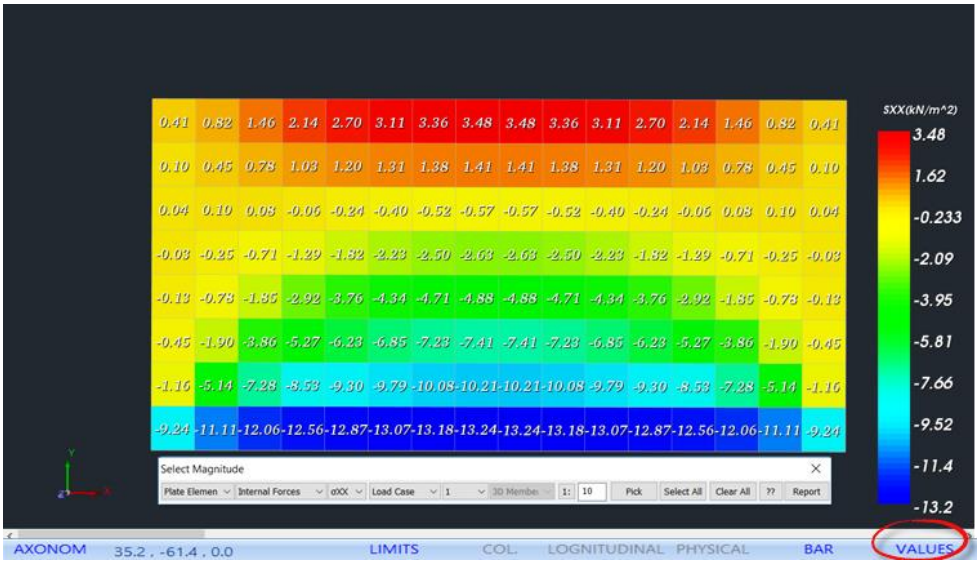
§ Plate Elements

For Plate Elements you can see:

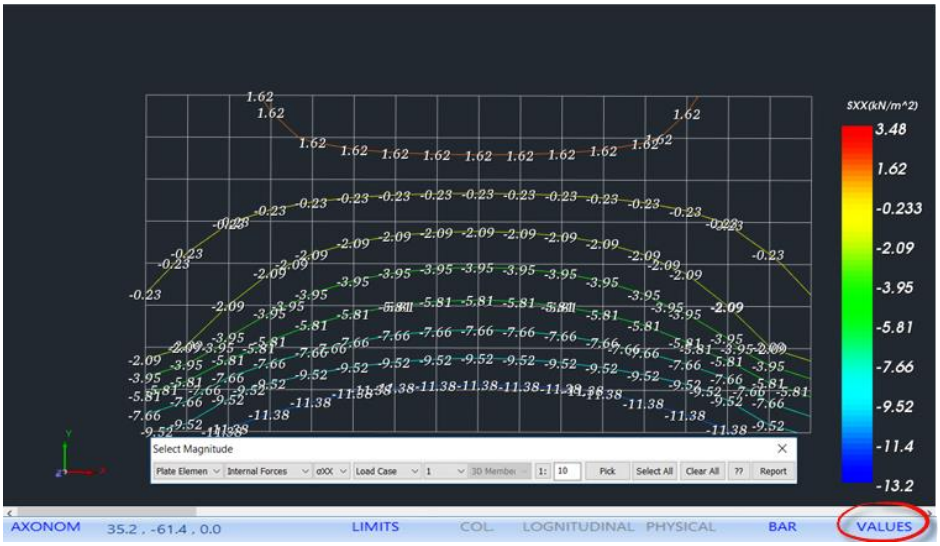




By activating VALUES in the lower horizontal bar, you can see the values of the selected size in the surface of the surface element,



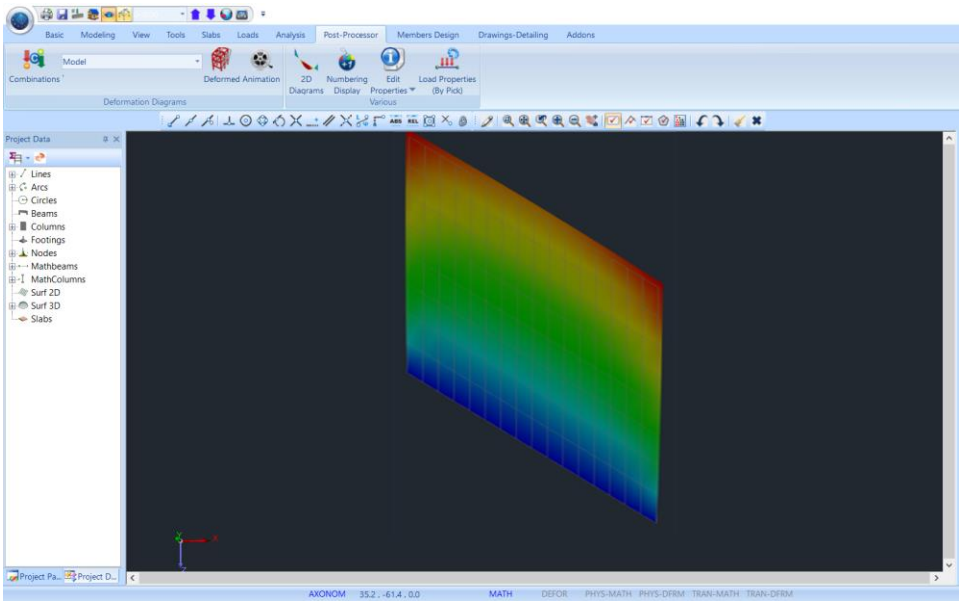
as well as the value of the Stress Contours over them



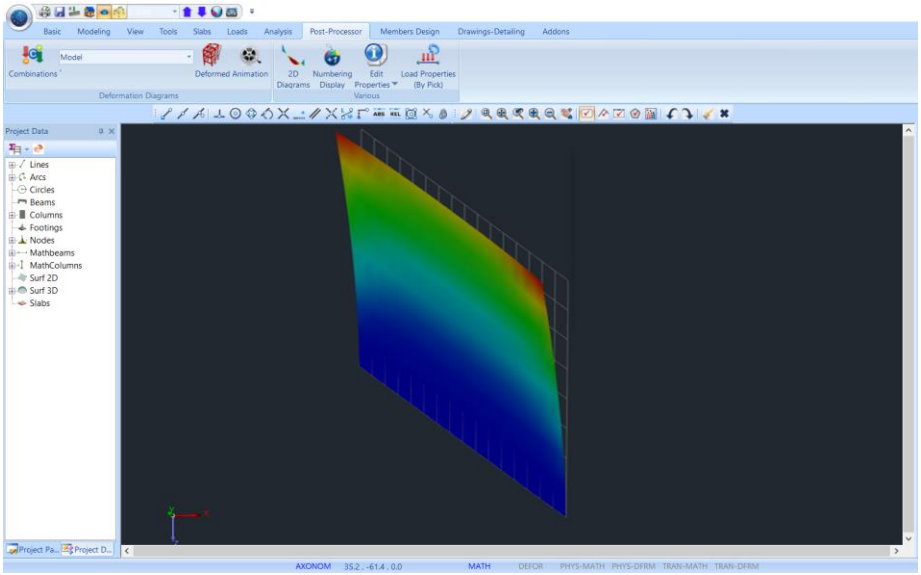
§ TEMPERATURE VARIATION

Furthermore,
for Plate (shell) elements imposed with Uniform Temperature Variations load and/or Linear Temperature Variations load.

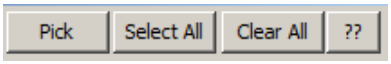
- The **Uniform Temperature Variations** causes membrane deformation in the plane of the element, while



- The **Linear Temperature Variations** causes deflection.

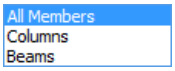


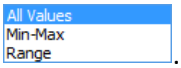
§ Select items



Select the elements by using:

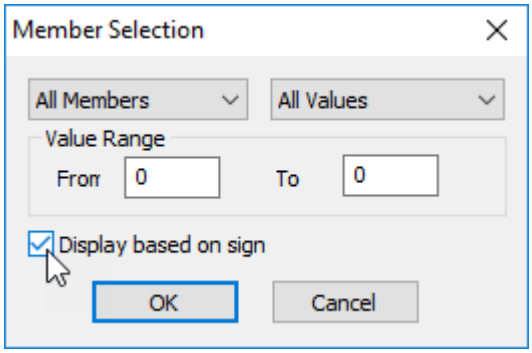
- “Pick” and right click on the elements
- “Select All”
- filters (?? button) allows you to filter the values in the diagrams.

For Linear Members, you can select to see member’s diagrams according to their type 

(Beam, Column or All Members) using a value filter 

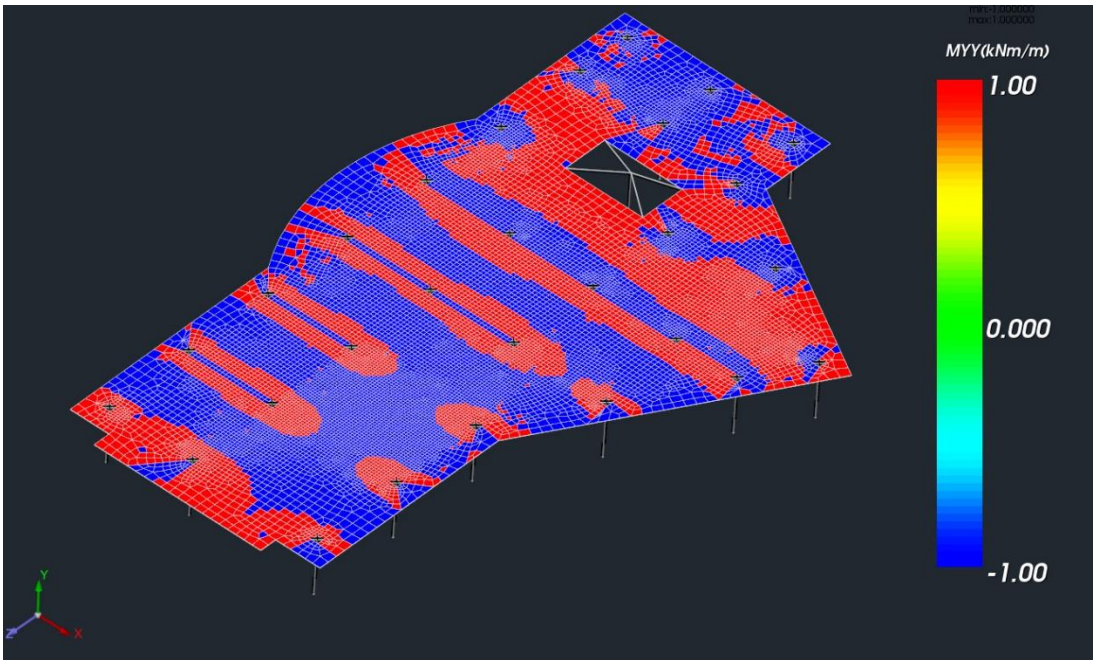
.

By pressing the button, the following dialog box appears, in which you can select from the first list, the type of the element for the display of the corresponding diagrams and the second list:



- “All values”: to display diagrams without value limits
- “Min-Max”: to display diagrams only on the items with the maximum and minimum values.
- “Value Range”: to display diagrams only on the items where the corresponding value is between the defined values’ range.

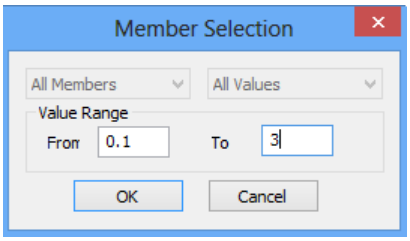
Additionally, there is the "Display based on sign" option. By activating the checkbox, the selected size values are displayed with two different colors, one for the positive values and one for the negative.



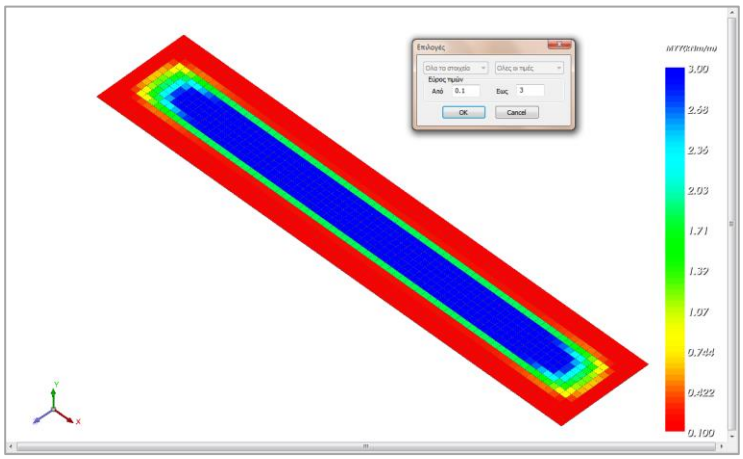
This option works for all sizes and elements (linear and surface)

EXAMPLE:

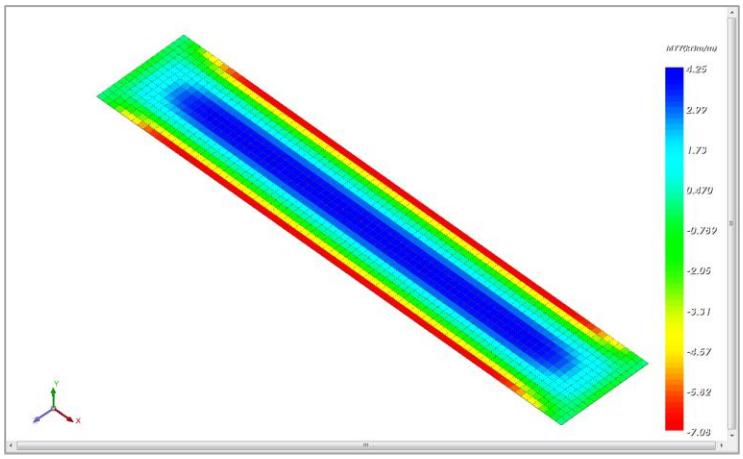
For Plate Elements/works only with “Value Range”



For example, for a value of Myy bending moment defined in a range between 0.1 and 3, the following moment contour is displayed:

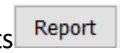


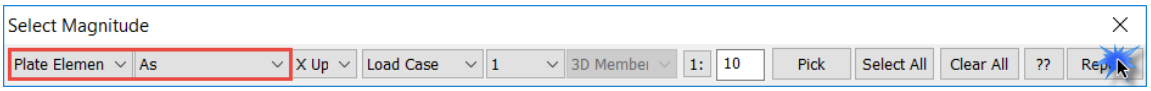
The part of the plate where the values are less or equal to 0.1KNm / m are colored red, while the values close to 3KNm / m are depicted with blue color.
The corresponding image without using the filter:



⚠ Notice that the color bar always has the same color gradation ranging from red to blue (red, green, blue (RGB).

§ REPORT

It concerns the reinforcement of the surface elements 



By selecting it, it is displayed for each surface element:

- the worst As,
- the combination is coming from and
- the corresponding intensive forces.

Calc_As.txt - Notepad

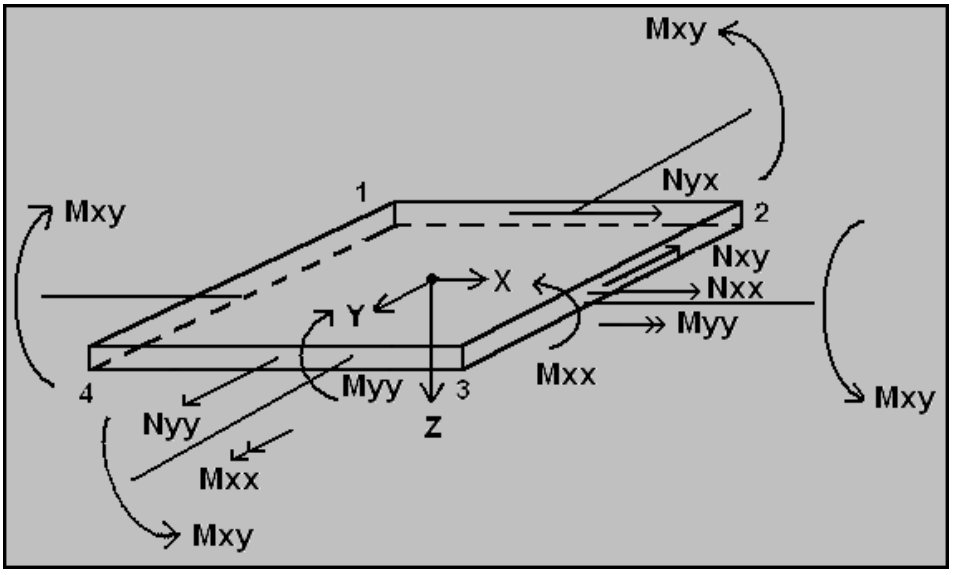
File Edit Format View Help

Name	Comb.	As(cm2/m)	Mx(kNm/m)	My(kNm/m)	Mxy(kNm/m)	Nx(kN)	Ny(kN)
1	1	0.002	-10.786	-4.891	-8.110	0.206	-2.309
2	-1	0.000	0.000	0.000	0.000	0.000	0.000
3	-1	0.000	0.000	0.000	0.000	0.000	0.000
4	-1	0.000	0.000	0.000	0.000	0.000	0.000
5	-1	0.000	0.000	0.000	0.000	0.000	0.000
6	-1	0.000	0.000	0.000	0.000	0.000	0.000
7	-1	0.000	0.000	0.000	0.000	0.000	0.000
8	-1	0.000	0.000	0.000	0.000	0.000	0.000
9	1	0.004	-30.956	-1.791	-10.160	0.408	-2.291
10	1	0.002	-31.269	-4.649	-17.166	0.226	-6.865
11	1	0.001	-33.164	-5.198	-24.971	0.050	-11.454
12	1	0.001	-37.101	-3.219	-33.757	-0.123	-16.095
13	1	0.002	-44.058	2.105	-42.549	-0.392	-20.869
14	1	0.005	-56.334	12.425	-49.352	-0.952	-25.856
15	1	0.014	-78.067	29.631	-48.256	-2.570	-30.990
16	1	0.788	-101.377	42.971	-24.980	-5.555	-35.047
17	1	0.008	-49.163	-0.141	-10.513	0.732	-2.336
18	1	0.004	-50.587	-0.714	-15.894	0.390	-7.018
19	1	0.000	-53.798	-0.373	-21.803	0.040	-11.740
20	1	0.002	-59.536	1.347	-28.024	-0.357	-16.522
21	1	0.005	-68.643	4.768	-33.109	-0.927	-21.354
22	1	0.010	-82.176	9.588	-34.696	-1.930	-26.161
23	1	0.082	-100.042	12.975	-28.847	-3.642	-30.662
24	1	0.033	-116.152	0.647	-11.913	-6.029	-34.782
25	1	0.011	-65.003	0.798	-9.646	1.069	-2.359
26	1	0.005	-67.274	1.692	-13.621	0.513	-7.103
27	1	0.000	-71.291	2.419	-17.671	-0.029	-11.886
28	1	0.004	-77.645	3.245	-21.482	-0.645	-16.688
29	1	0.008	-86.696	3.967	-23.682	-1.462	-21.461
30	1	0.014	-98.419	3.311	-22.557	-2.635	-26.108
31	1	0.023	-111.503	-2.161	-16.344	-4.264	-30.537
32	1	0.611	-122.655	-19.988	-6.006	-6.280	-34.746
33	1	0.018	-78.002	1.318	-7.955	1.351	-2.365
34	1	0.037	-80.723	2.971	-10.732	0.599	-7.129

The local axes of the plate and the corresponding intensive forces are shown in the figure below.

§ CLARIFICATIONS:

Clarifications on forces and reinforcement in finite surface elements:



⚠ Concerning the **intensive forces**:

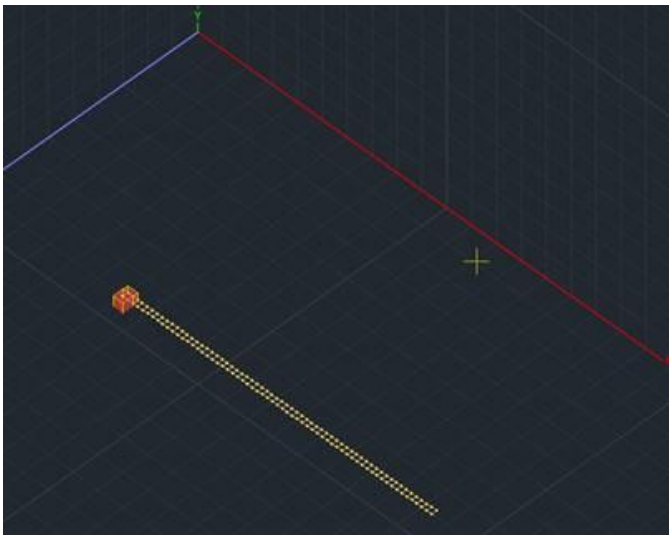
The forces refer to the **local system** of the element which is also shown, and listed in the file “out”.

Attention to the moments:

- **Mxx** is the moment about **y** local axis and
- **Myy** is the moment about **x** local axis.
-

EXAMPLE:

For better understanding, a cantilever has been introduced in the example below.

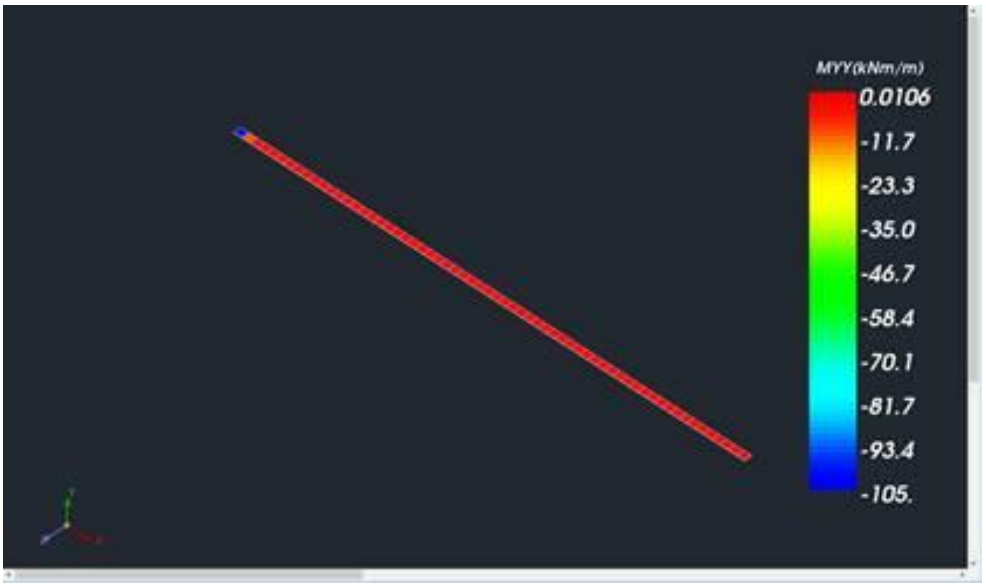


- The local axis **x** coincides with the Catholic **X**.

Therefore, in the out file, we aspect to see the large moments about the local axis y means the Mxx moments.

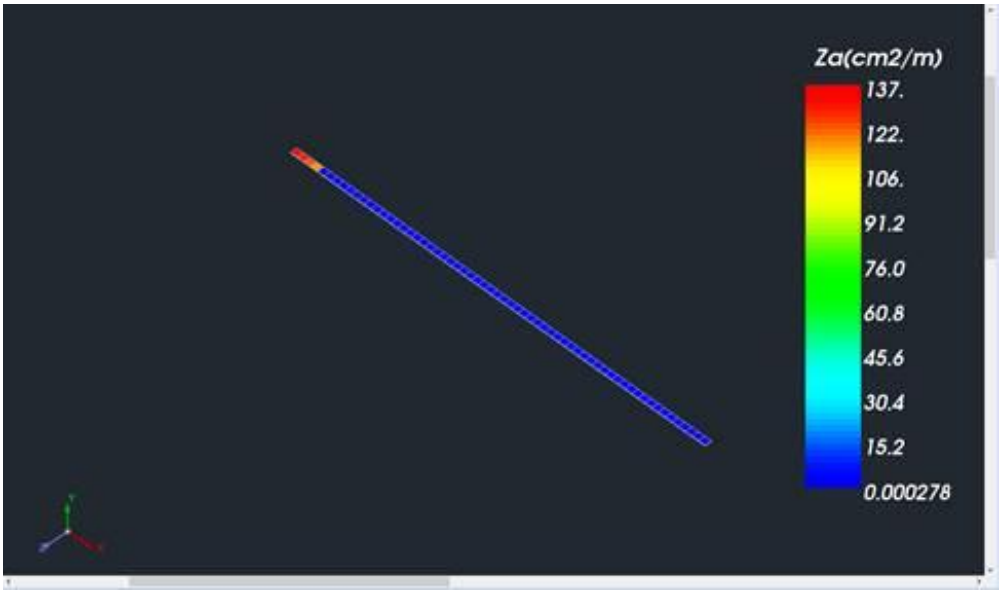
Go in the results and show the Mxx moments:

While the respective Myy moments are clearly much smaller:



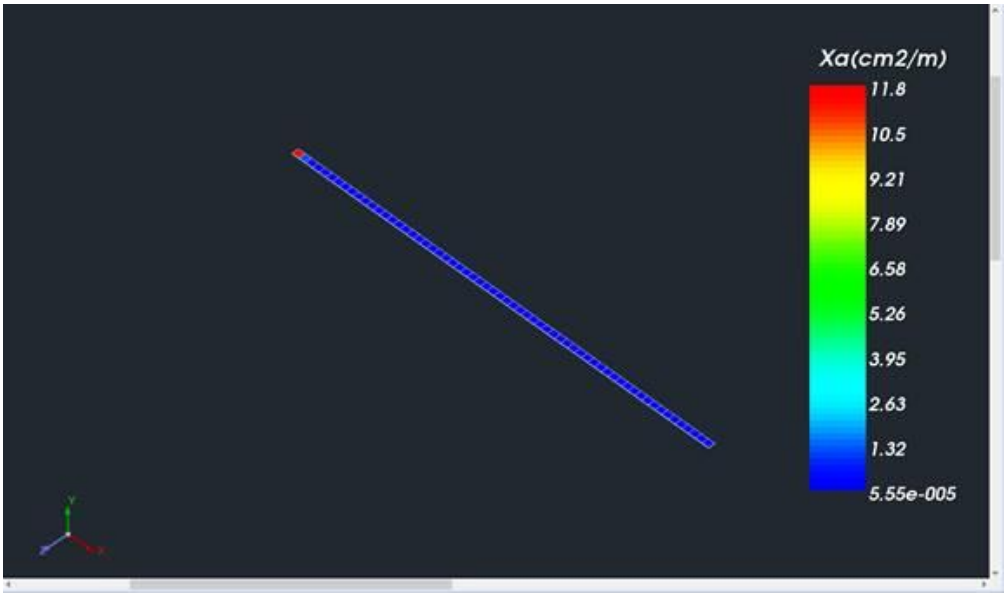
⚠ Concerning the **reinforcements**:

- **Up** for the plate is the **START** point of the local **z**.
  Naturally, for the cantilever, we expect to see more upper reinforcement - ATTENTION - in **z up** the option and **not** in **x** up.

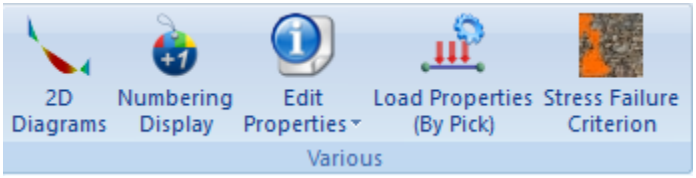


It is clarified that the **reinforcement:**

- now refers to **Catholic's** axes and
- the position direction is **perpendicular** to the respective axis

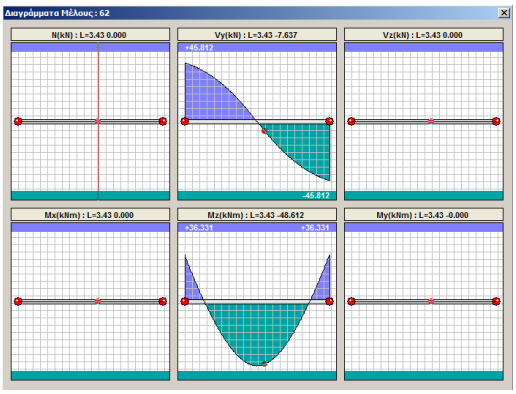


2. Various



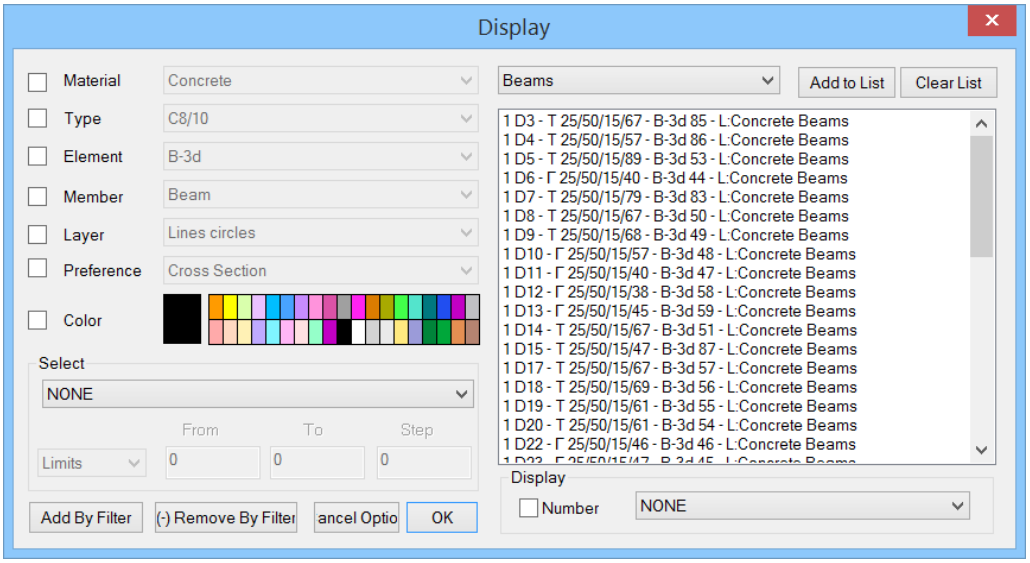
2.1 2D Diagrams

 : is the short way to see all six internal forces of a member (selected with left click) in a single window. By moving the mouse, you can also see the corresponding values along the member.

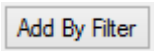


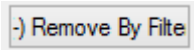
2.2 Numbering Display


: to display any information like numbering, degrees of freedom, members releases, rigid link constraint, etc.
Press the command and in the dialog box:



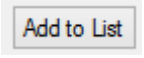
Select the elements by using the filters.
Select as appropriate:

- A filter like “Material”, “Type”, “Element Type” etc., and press 

to add the selected elements in the list, or 

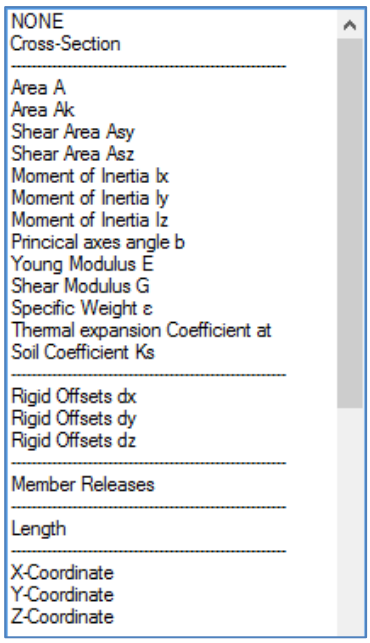
to remove them from the existing list.

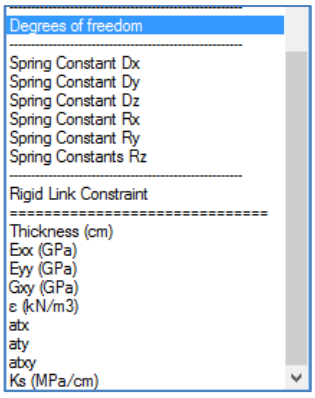
- One of the groups of the list 

and press 

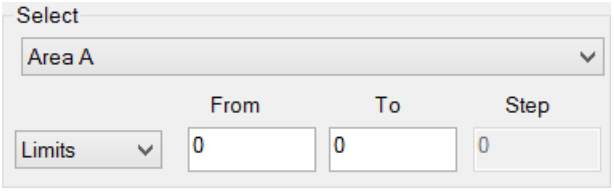
Then, choose the information you want to display from the “Display” list:

Check “Number” to display the numbering of the selected items.

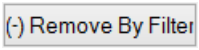




In “Select” field you can set additional filters as a function of the maximum and minimum values, or the limits that you specify. For example, you can display the max and min values of beams’ cross-sections, or those included within the limits "From", "To" with a certain step, etc.



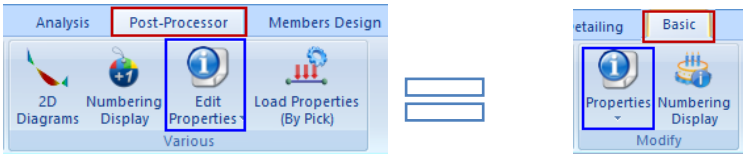
- To make all the displaying values disappear, select "NONE", disable the check box "Number"

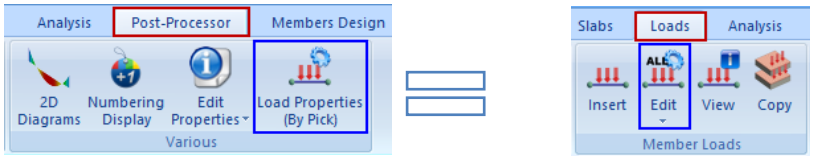
and press 

.

2.3 & 2.4 Edit Properties & Load Properties

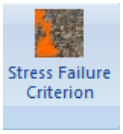
“Edit Properties” and “Load Properties” have already been explained in BASIC and LOADS Ribbon, respectively. (see Chapter 1 & 6)





⚠ *For reasons of easy use and quick search you can also find them in "Various"*

2.5 Load bearing masonry check based on Stress Failure Criterion



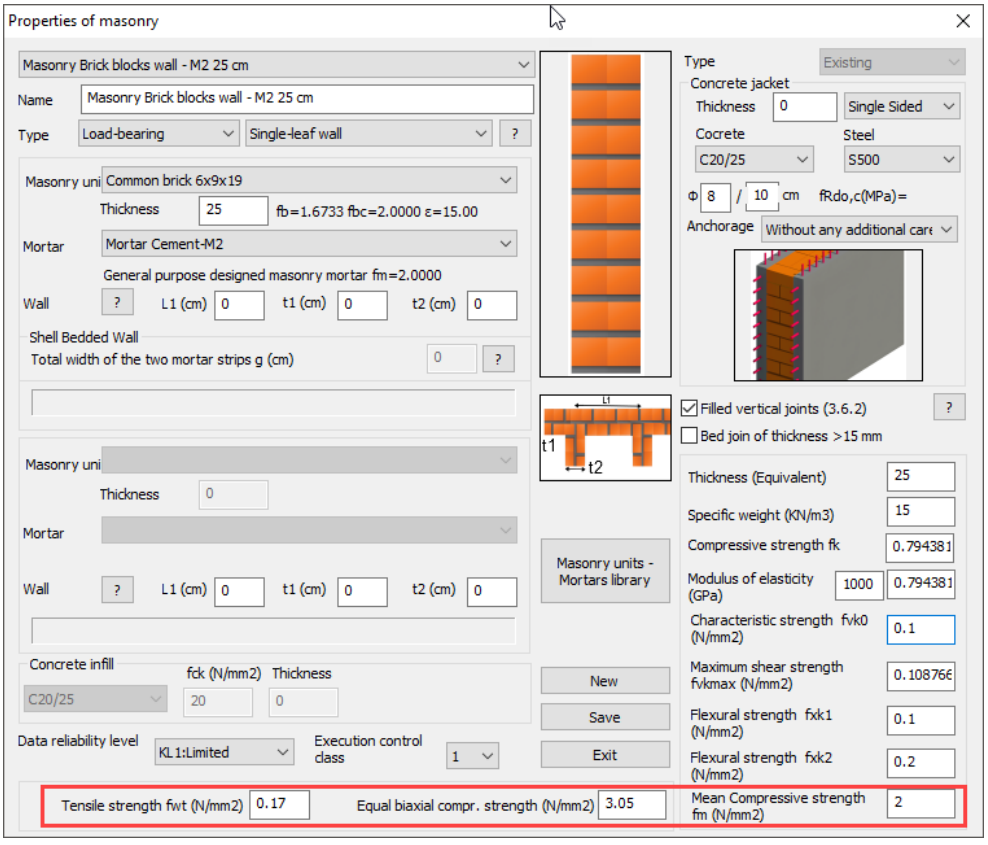
In the new version of SCADA Pro, the check of load bearing masonry regarding stresses based on the **Karantoni et al (1993)** stress failure criterion is added.
The check regarding stresses is performed in both curved and planar walls for existing or new masonry.

The check command is located in the **Post-Processor** tab where:

The requires steps are the following:

1. **We define the material and the stengths in the masonry library.**

We define the values of the strengths in the masonry library:



- Axial compressive strength f_{wc}
- Axial tensile strength f_{wt}

- Equal biaxial compressive strength f_{wcb}

Indicatively, the values suggested by the authors can be used:

$$\frac{f_{wt}}{f_{wc}} = 0.085 \quad \& \quad \frac{f_{wdc}}{f_{wc}} = 1.65 \qquad (4)$$

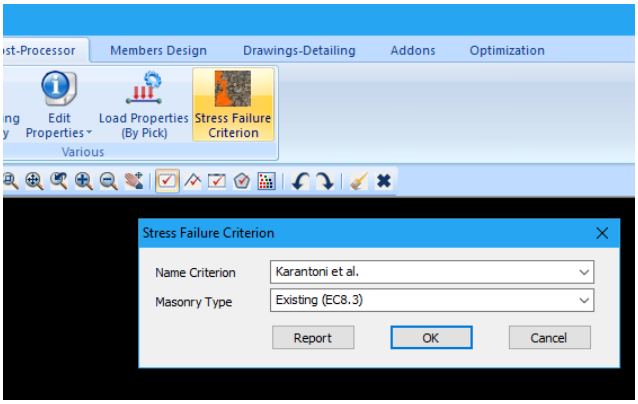
⚠ In cases that the values are **NOT** filled in, something that is not recommended, the suggested ones will be used automatically.

2. **We run the analysis and we create the combinations.**

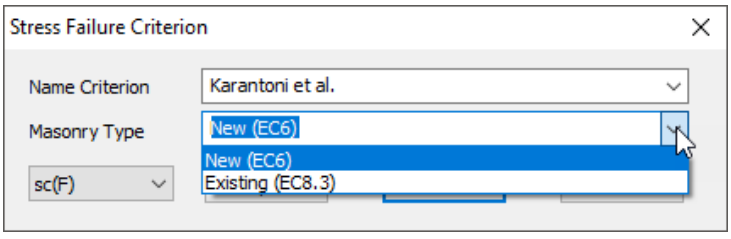
3. **We move to the ribbon POST-PROCESSOR.**

4. **We pick the combinations.**

5. **We move to the command STRESS FAILURE CRITERION and we select the TYPE of the masonry.**



We must choose whether the masonry is NEW or EXISTING so that appropriate safety coefficients divide the strengths f.



More specifically:

- For a **NEW** one --> coefficient γ_s based on EC6
- For an **EXISTING** one --> coefficient CF_m based on EC8-3

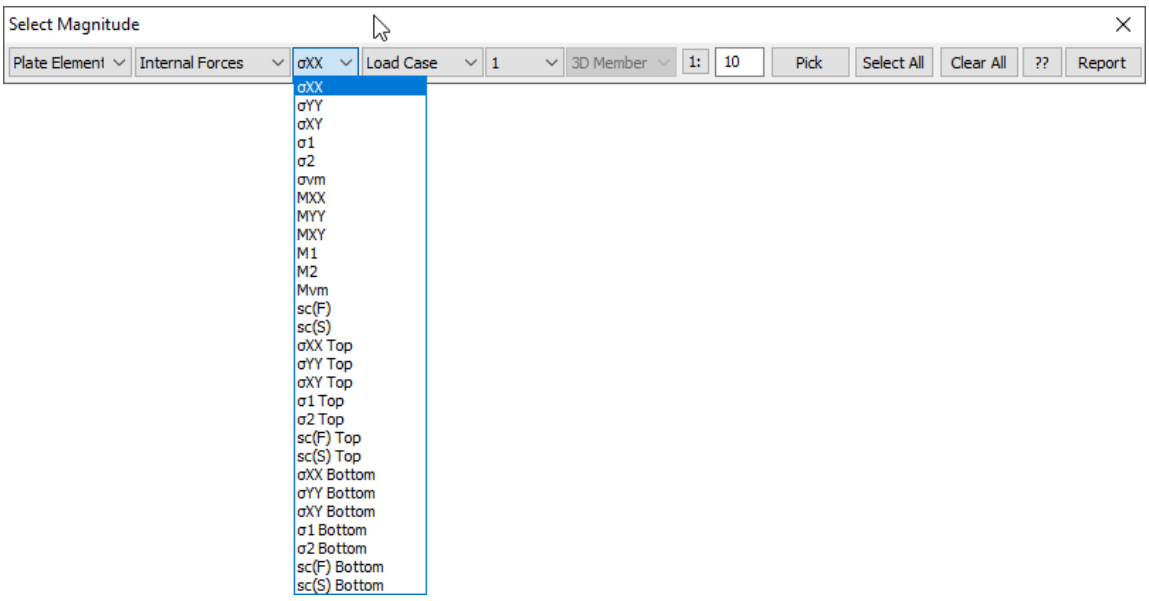
6. **We press OK and we go out.**

7. **In the select magnitude bar, we select the criterion display.**

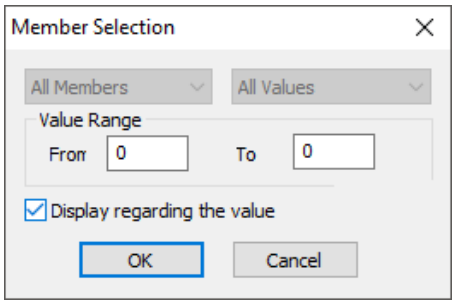
⚠ We remind you that according to recent changes surface tensions are calculated on both the mean plane and the two alignments of the element.

So we can choose to see the criterion:

- either with **MODE 1 (scF)**
- or with **MODE 2 (scS)**
  in each of these three positions.

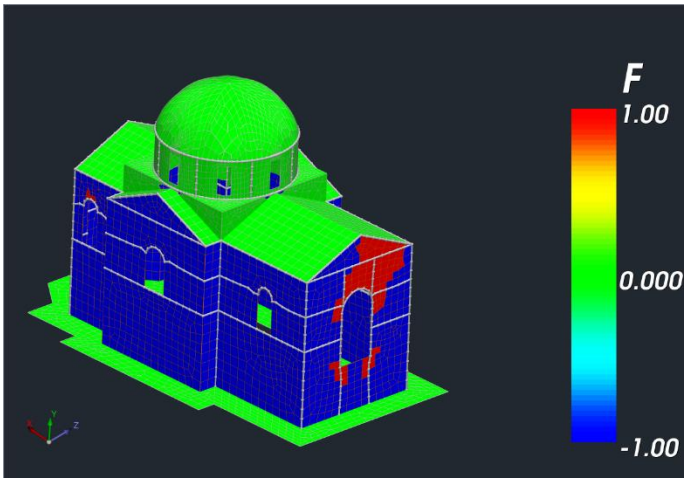


By choosing display based on sign 



then the structure is colored according to the criterion:

- BLUE for **SUFFICIENCY**
- RED for **INSUFFIENCY**
- GREEN for a material different than the masonry's one(i.e concrete)



For a better evaluation of the checks results to be achieved, there are the two following options

8. **If we wish so, in the select magnitude bar we select the command REPORT.**

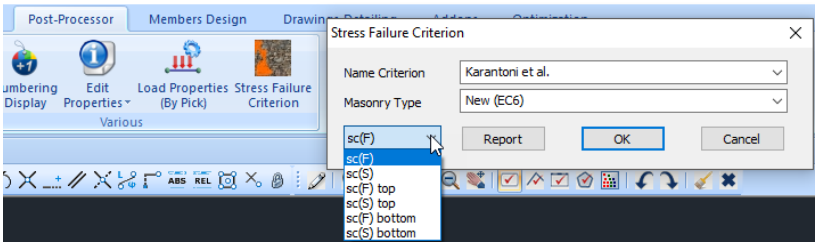
By selecting this order, the values of the examined element are printed per surface element.

```
Name    Comb.         F
-----------------------

****** Plegma - S30 ******
  696       4       -0.549
  697       4       -0.573
  698       4       -0.625
  699       4       -0.731
  700       4       -0.798
  701       4       -0.807
  702       4       -0.761
  703       4       -0.748
  704       4       -0.679
  705       4       -0.333
  706       4       -0.519
  707       4       -0.338
```

9. **By selecting the command STRESS FAILURE CRITERION we see a summary printout which contains all the details about the sufficiency or insufficiency of each mesh.**

From the dropdown list, we select the format and the display location of the criterion. By pressing the REPORT command, the printout is printed.



The printout that comes up has the following form:

Stress Failure Criterion

Name of Criterion Karantoni et al.
Masonry Type Existing (EC8.3)
Criterion Description $F = \alpha J_2/f_w^2 + \lambda J_2^{(1/2)}/f_w + \beta I_1/f_w - 1$
SUFFICIENCY : Για $F < 0$
INSUFFICIENCY : Για $F >= 0$

Mesh Check

Mesh Name : Plegma S21 Material : Masonry stone wall - M2 50 cm

Compressive strength f_w = 0.000 (N/mm^2) γ_M = 2.20 / 1.50
Tensile strength f_{wt} = 0.000 (N/mm^2) CF = 1.35
Equal biaxial comp. strength f_{wc_b} = 0.000 (N/mm^2)

Criterion Parameters : α = 0.665 b = 1.650 c_1 = 13.765 λ_1 = 0.581
β = 3.835 f = 0.085 c_2 = 0.959 λ_2 = 0.995

Number of elements	Total Area (m^2)	Number of elements that fail	Total Failure Area (%)	Critical Combination			
				ID.	Number of elements that fail	Total Failure Area (%)	F_{max}
128	8.64	0	0.00	37	0	0.00	-0.31

##########

Mesh Name : Plegma S22 Material : Masonry stone wall - M2 50 cm

Compressive strength f_w = 0.000 (N/mm^2) γ_M = 2.20 / 1.50
Tensile strength f_{wt} = 0.000 (N/mm^2) CF = 1.35
Equal biaxial comp. strength f_{wc_b} = 0.000 (N/mm^2)

Criterion Parameters : α = 0.665 b = 1.650 c_1 = 13.765 λ_1 = 0.581
β = 3.835 f = 0.085 c_2 = 0.959 λ_2 = 0.995

Number of elements	Total Area (m^2)	Number of elements that fail	Total Failure Area (%)	Critical Combination			
				ID.	Number of elements that fail	Total Failure Area (%)	F_{max}
146	7.13	7	7.75	35	4	4.37	0.44

##########