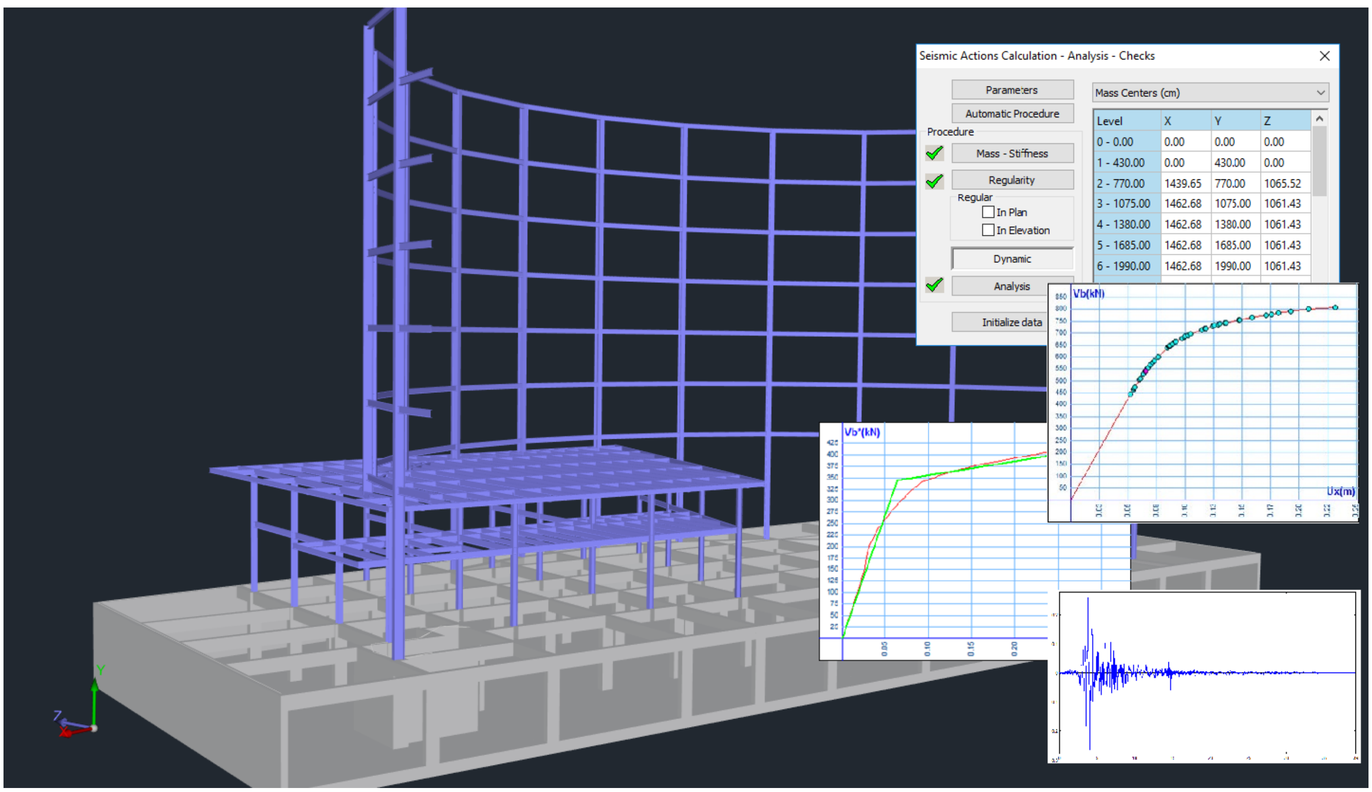
SCADA Pro tm
Structural Analysis & Design

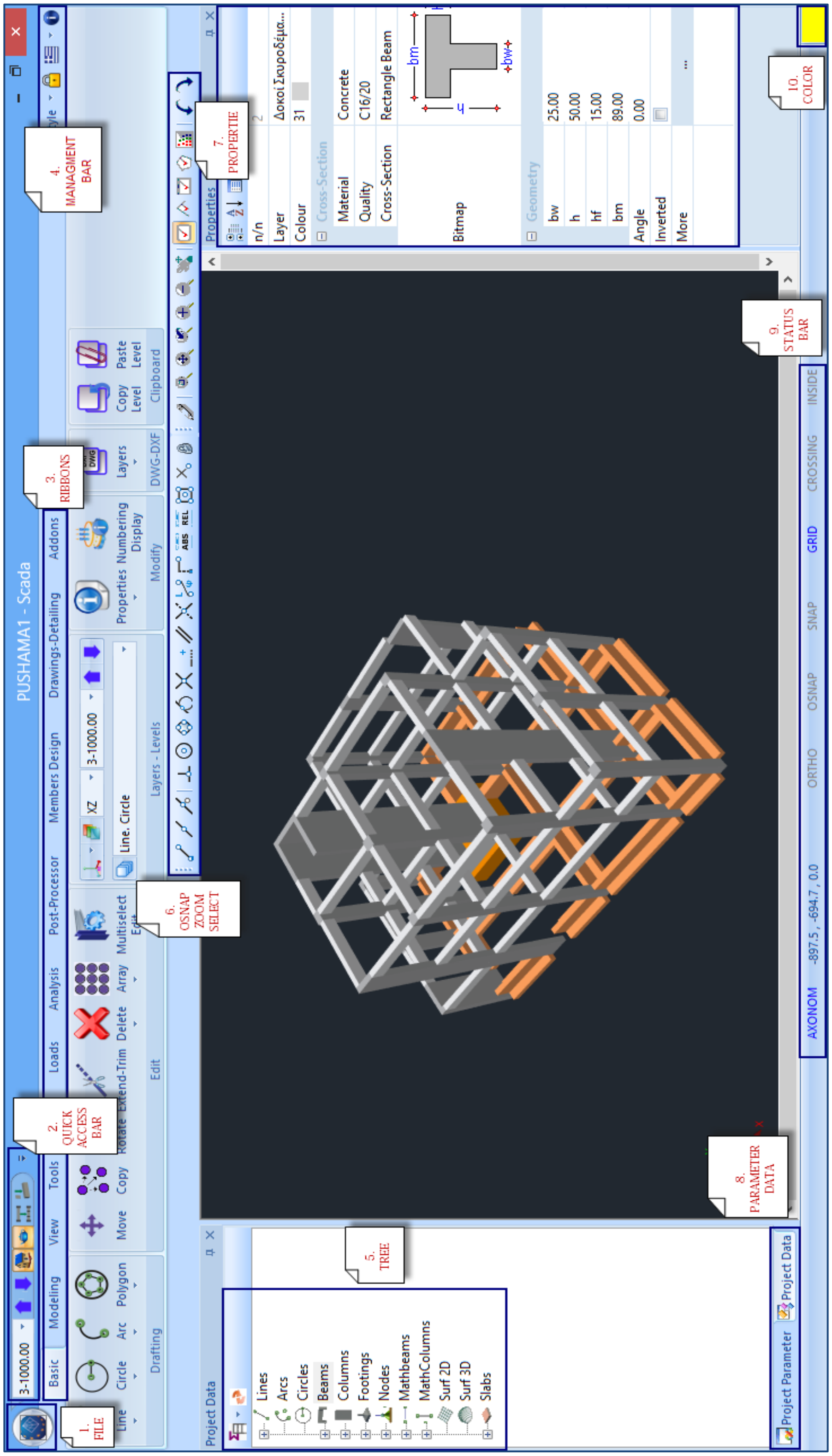
User's Manual
7.ANALYSIS



ACE
HELLAS
INTEGRATED SOLUTIONS

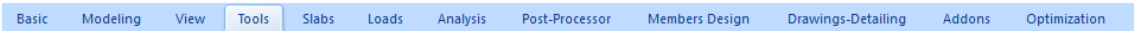
CONTENTS

I. THE NEW UPGRADED INTERFACE of SCADA Pro

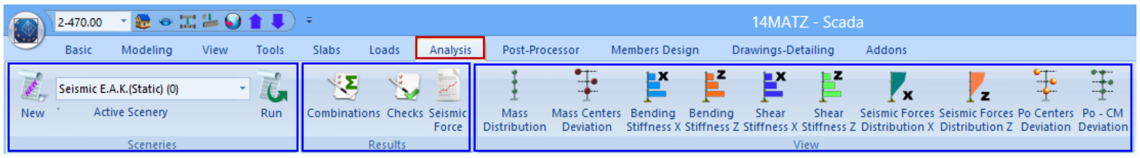


II. DETAILED DESCRIPTION OF THE NEW INTERFACE

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



Analysis

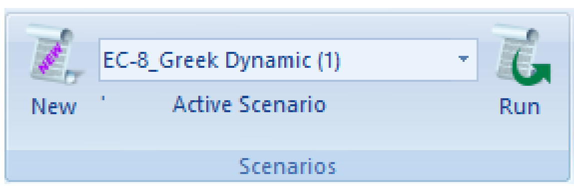


The 7th Unit called “ANALYSIS” includes the following three groups of commands:

Scenarios
Results
View

⚠ Since modeling and loads’ assignment have been completed, the “Analysis” of the structural member, for the design of the structure, follows, based on provisions of the current design codes. SCADA Pro contains European, Greek, and Italian Design Codes, in which the application of linear and nonlinear analysis methods is proposed, as appropriate.

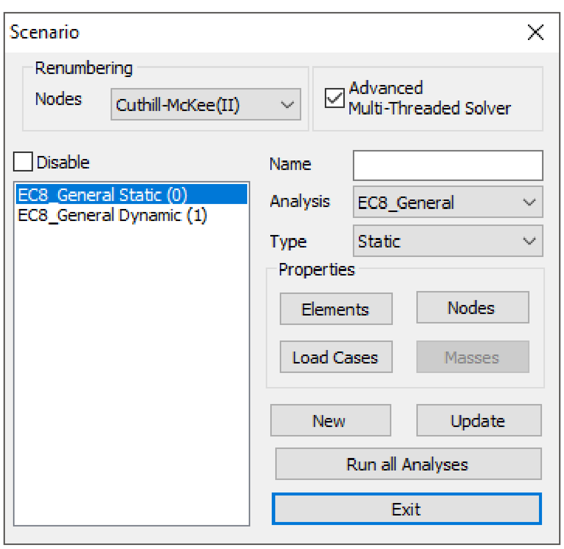
1. Scenarios

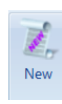


“Scenarios” commands group allows scenarios’ creation (choosing regulation and type of analysis) and implementation.

1.1 New

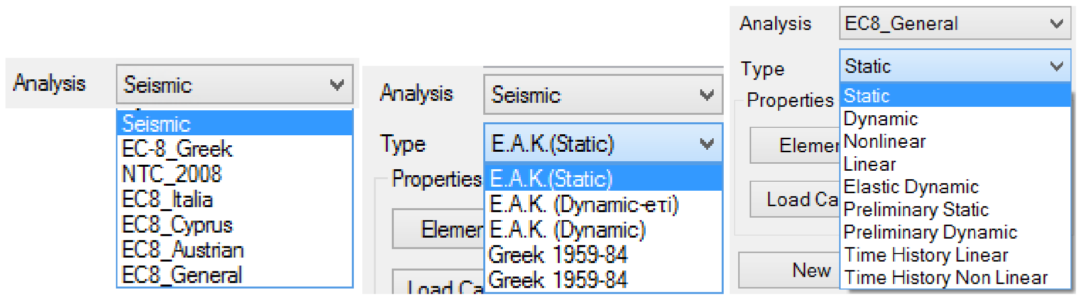
The command for the scenarios’ creation

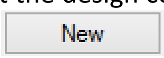




Press “New” and in the dialog box, you can create analysis scenarios by choosing different design regulations and methods of analysis. By default, there are two scenarios based on the selected “language” codes (including local Annex if there are any, or “EC-General” if there are not)

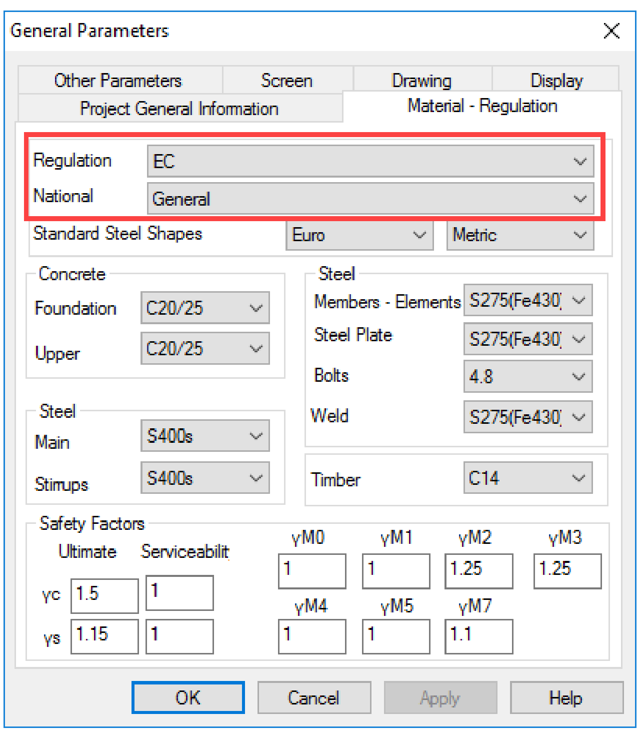
The program is now integrated with new rapid analysis algorithms, using more resources, such as the graphic card, resulting in the more rapid implementation of theirs (Parallel Processing). The activation is achieved through the creation of scenarios.



Select the design code from the “Analysis” list and the analysis method from the “Type” list and click 

to create a new analysis scenario. Optionally, type a name.

⚠ *Predefined scenarios are created according to the Rule and Attachment option you made at the beginning, within the General Parameters window that opens automatically immediately after the file name is defined.



Select among the possible scenarios provided in SCADA Pro:
For Greece:

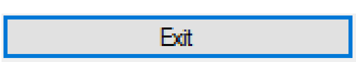
LINEAR – NON LINEAR METHODS

- EAK Static	Simplified spectral analysis according to EAK
- EAK Dynamic-eti	Dynamic spectral analysis according to EAK
- EAK Dynamic	Dynamic spectral analysis (masses displacement) according to EAK
- Old 1959-84	Seismic analysis according to 1959 Regulation
- Old 1984-93	Seismic analysis according to 1984 Regulation
- Static	Static Analysis without seismic actions
- EC 8 Greek static	Static analysis according to Eurocode 8 and the Greek Annex
- EC8 Greek dynamic	Dynamic analysis according to Eurocode 8 and the Greek Annex
- EC 8 Greek Preliminary Static	Static Preliminary analysis according to KANEPE
- EC8 Greek Preliminary Dynamic	Dynamic Preliminary analysis according to KANEPE
- EC 8 Greek Time History Linear	Static analysis according to Eurocode 8
- EC 8 Greek Time History Non Linear	Dynamic analysis according to Eurocode 8
- EC 8 Greek NonLinear	Nonlinear analysis according to Eurocode 8 & KANEPE.

For other countries:

LINEAR – NON LINEAR METHODS

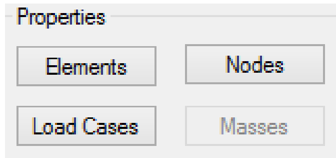
- NTC 2008	Seismic analysis according to the Italian Regulation 2008
- EC8 Italia	Seismic analysis according to Eurocode 8 and the Italian Annex
- EC8 Cyprus	Seismic analysis according to Eurocode 8 and the Cyprus Annex
- EC8 Austrian	Seismic analysis according to Eurocode 8 and the Austrian Annex
- EC8 General	Seismic analysis according to Eurocode 8 with no Annex (enabled typing values and coefficients)
- EC 8 General Non Linear	Nonlinear analysis according to Eurocode 8
- SBC 301	Seismic analysis according to Saudi Arabia code (SBC 301)

Select 

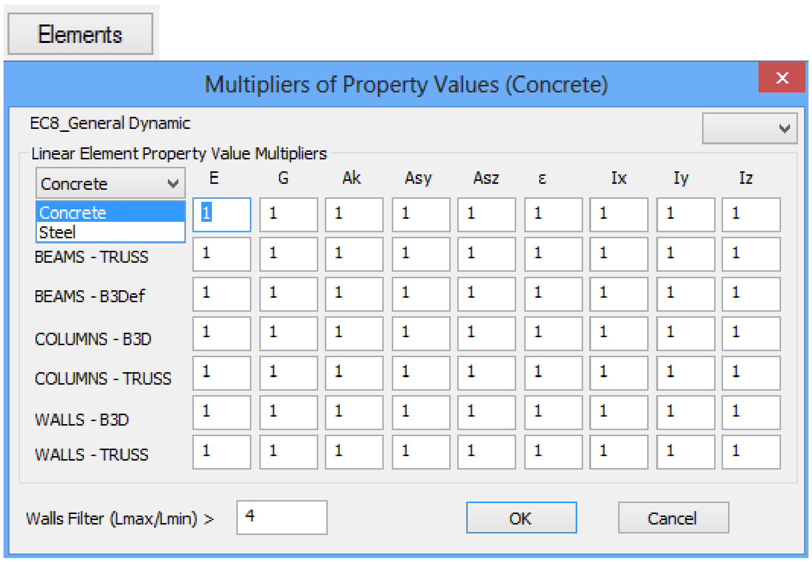
to save the scenarios and move on to the analysis.

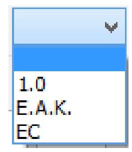
PROPERTIES

“**Properties**” field includes the buttons Elements, Nodes and Loads to define the relevant coefficients.



§ Both predefined and new scenarios have, by default, filled in these Elements, Nodes and Loads. The user can modify them.





Click “Elements” to open the dialog box that contains the multipliers of the characteristic properties of the linear element, considered in the analysis.

By default, the values of the multipliers are defined according to the design code, while any modification is acceptable.

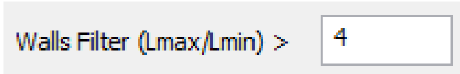
If for example, you select “EC” the values of the multipliers will automatically be updated by the Eurocode provisions.

E	G	Ak	Asy	Asz	ε	Ix	Iy	Iz
1	1	1	1	1	1	0.1	1	0.5
1	1	1	1	1	1	0.1	1	0.5
1	1	1	1	1	1	0.1	1	0.5
1	1	1	1	1	1	0.1	1	1
1	1	1	1	1	1	0.1	1	1
1	1	1	1	1	1	0.1	0.666	0.666
1	1	1	1	1	1	0.1	0.666	0.666

E	G	Ak	Asy	Asz	ε	Ix	Iy	Iz
1	1	1	1	1	1	0.1	0.5	0.5
1	1	1	1	1	1	0.1	0.5	0.5
1	1	1	1	1	1	0.1	0.5	0.5
1	1	1	1	1	1	0.1	0.5	0.5
1	1	1	1	1	1	0.1	0.5	0.5
1	1	1	1	1	1	0.1	0.5	0.5
1	1	1	1	1	1	0.1	0.5	0.5

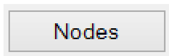
- EC8_General is a scenario without an Annex considered. This means that the user has to update the coefficients according to a state Annex.
- Using EC8_Greek, Italy, Cyprus and Austrian all coefficients are automatically filled in.

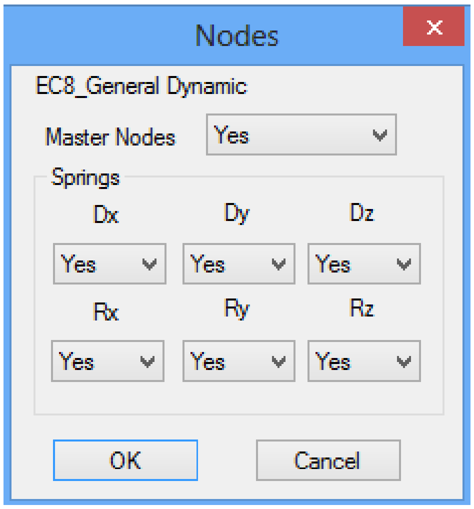
You can also define the dimensions for vertical elements to qualify "Walls".



Press 

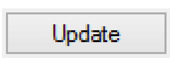
to update the scenario including these changes.



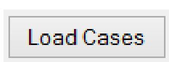


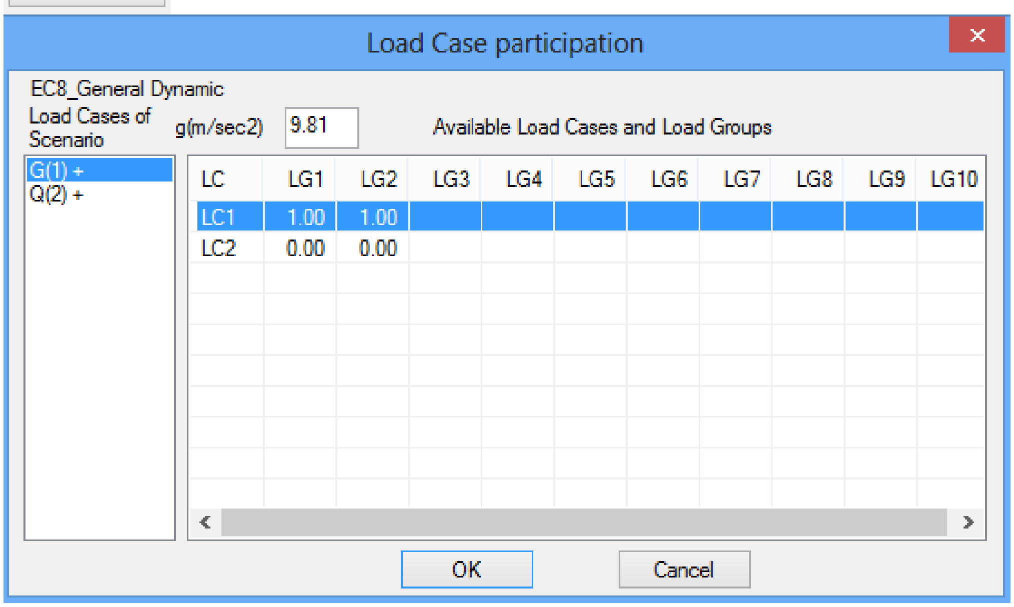
Select whether to consider slab’s Master Node (FSR) by selecting "Yes" (default) or not by selecting "No".

Moreover, choose whether to allow the corresponding displacement or rotation of the foundation’s springs or not (fixed support conditions).

Press 

to update scenarios including the changes.





In this form the loads’ participation factors are defined, i.e. the participation of each “Load Case of Scenario” (LC) including the load groups (see “Loads”>>”Load Groups”).

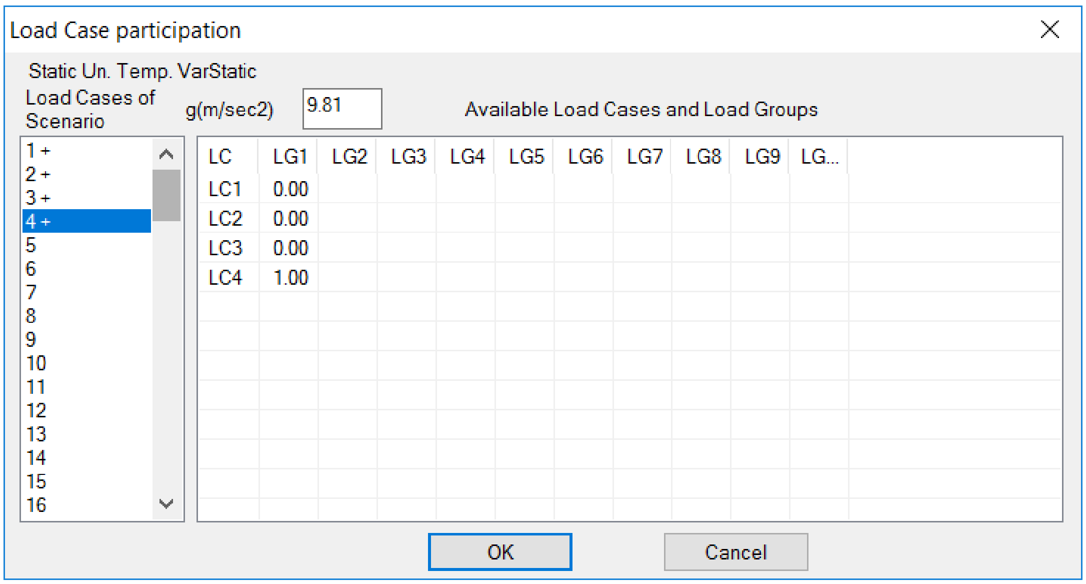
For scenarios **considering the seismic actions**,

- select “Dead Loads” – G(1) and type 1.00 next to LC1, under LG1 or LG2 or both (it depends on your choice to consider all dead loads together or not).
- select “Live Loads” – Q(2), and type 1.00 next to LC2, under LG1 or LG2 or both (it depends on your choice to consider all live loads together or not).

  “+” sign located next to the load category G(1) + Q(2) + indicates that there is an indicative multiplier for the participation of the specific load.

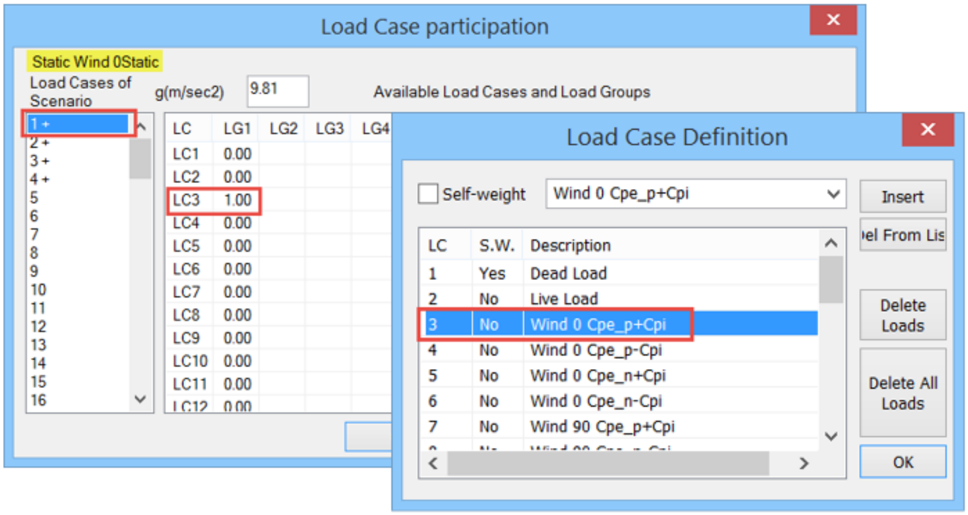
Scenarios **without considering seismic actions** (simple static method),

- Each load case (“Load Cases of Scenario”) is displayed with a number (i.e. LC1) and contains a load and its groups (i.e LG1). The load group is taken into consideration when the value in the corresponding cell is set to a value different than 0.00.
- Each Analysis Scenario can contain up to 4 loads.



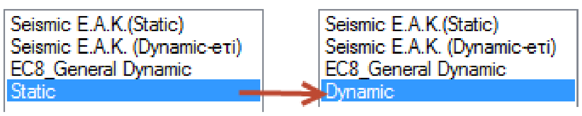
Click Update to apply any performed modifications.

EXAMPLE:
For example, the LC3 is defined as the first load of the Static scenario.

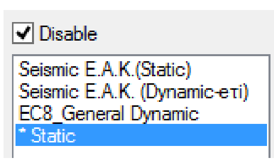


A scenario can be:

adapted: first select the scenario from the list, then change the characteristics and press "Update".



disable: first select the scenario from the list, then activate and press "Update" in order the “ * ” sign to appear. Deactivate “Disable” to restore it. (select > deactivate > “Update”)

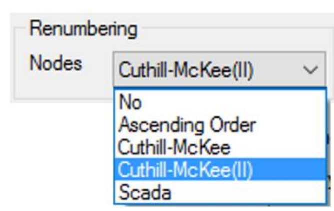


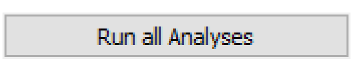
The **“Renumbering”** field includes a drop-down list with multiple options:

The choice of each option affects the computational time.

Default choice: “Cuthill-Mckee(II)”.

“Cuthill-Mckee” and “Ascending Order” take more time to complete the analysis, while choosing "No" is not recommended.

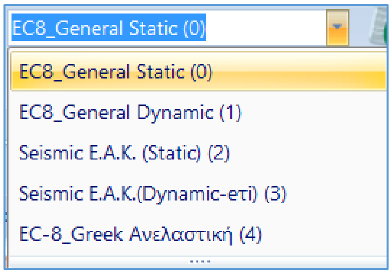




The new command allows you to run all the scenarios in the list with one click.

1.2 Active Scenario

Select a scenario from the list. The selected scenario will be the Active Scenario and it will be used for the analysis.



§ In the scenarios list, apart from the two predetermined ones, all the previously created scenarios are created. Choose one scenario at a time and continue with the definition of the parameters of the corresponding analysis.

1.3 Run

Click the “Run” button to open the parameters of the current analysis window which are classified into:

- EAK scenarios(figure 1)
- Linear scenarios (figure 2)
- Non Linear scenarios (figure 3)
- Time History scenarios (figure 4)

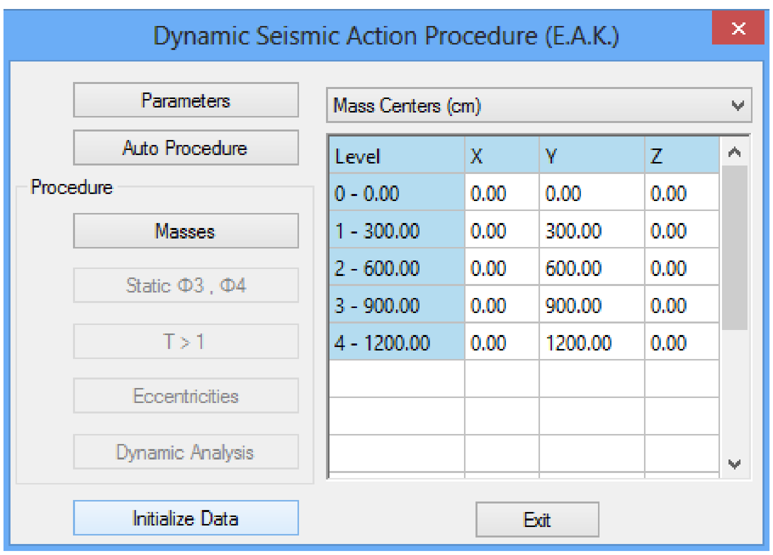


Figure 1

EAK scenarios*** are not described in the English version of the manual.

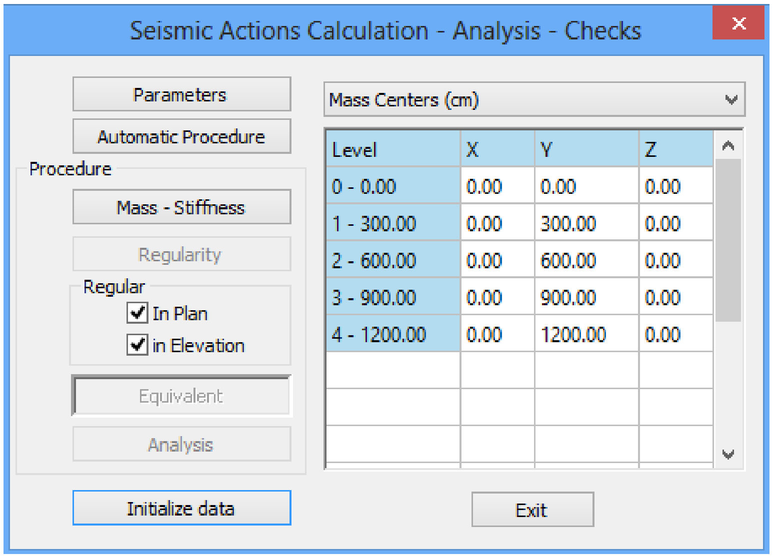


Figure 2

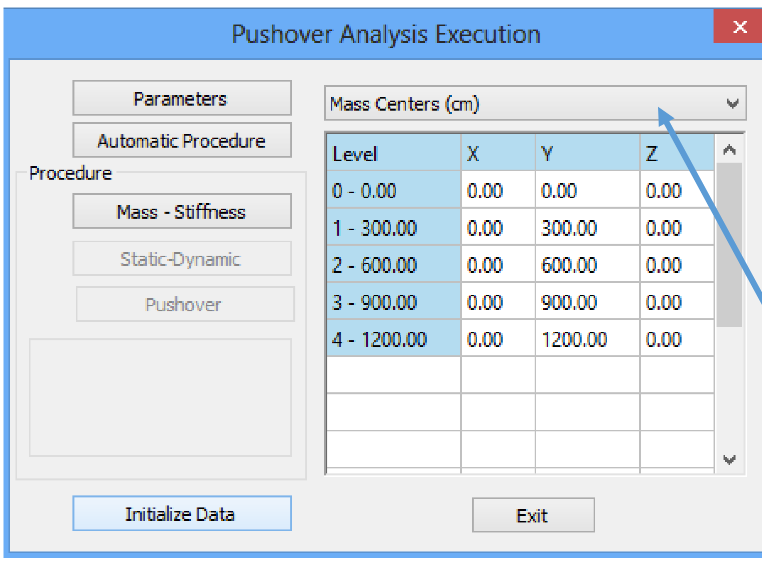
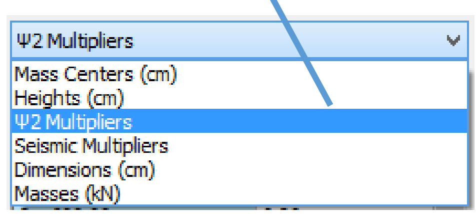
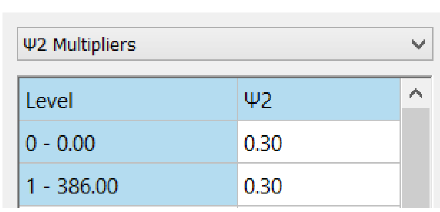


Figure 3



⚠ Always define the live loads’ coefficient ψ2. Default value: ψ2=0.30.



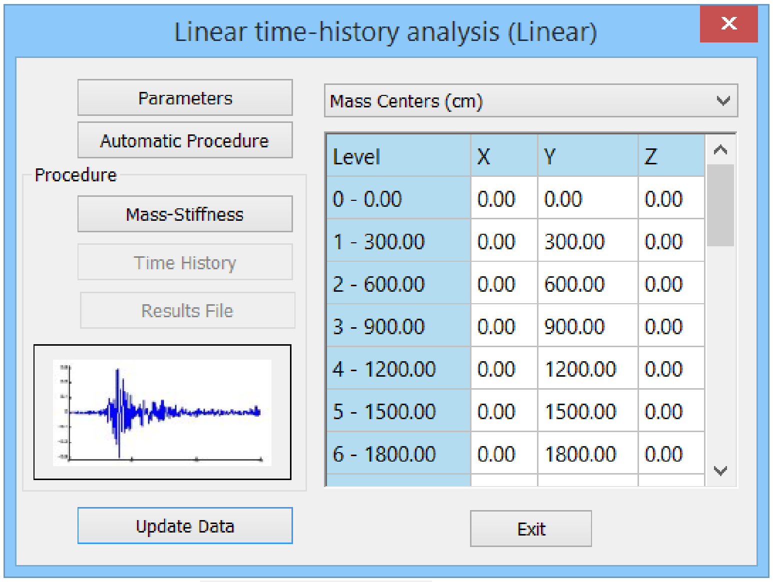


Figure 4

First of all, press Initialize Data to update the parameters of the current scenario.

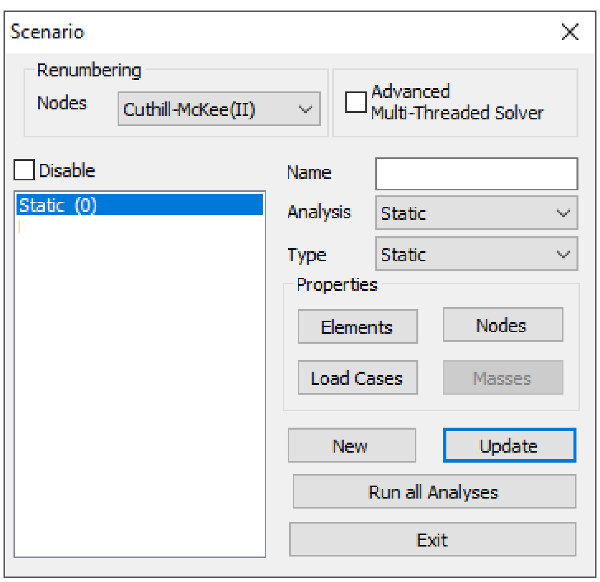
Then press Parameters to define the parameters of the project.

The parameters’ dialog box vary for each selected scenario, and so:

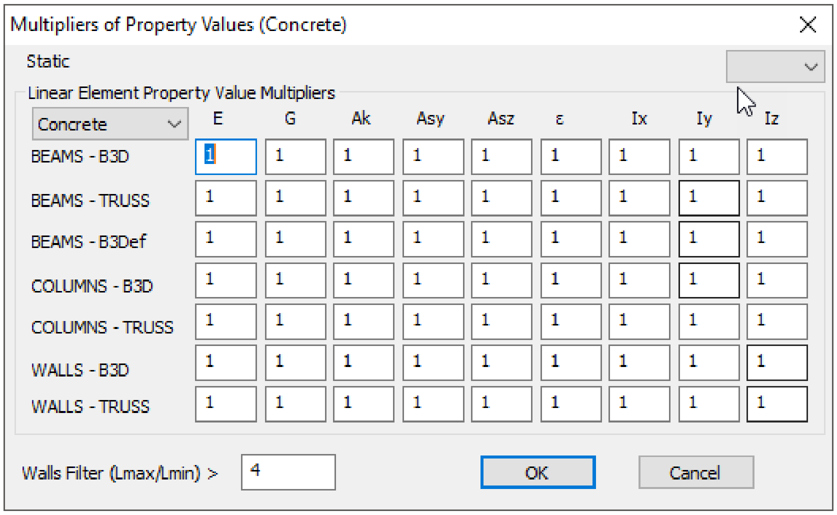
1. Scenario

§ Scenario Static

Select Analysis **Static** and Type **Static** and press New.



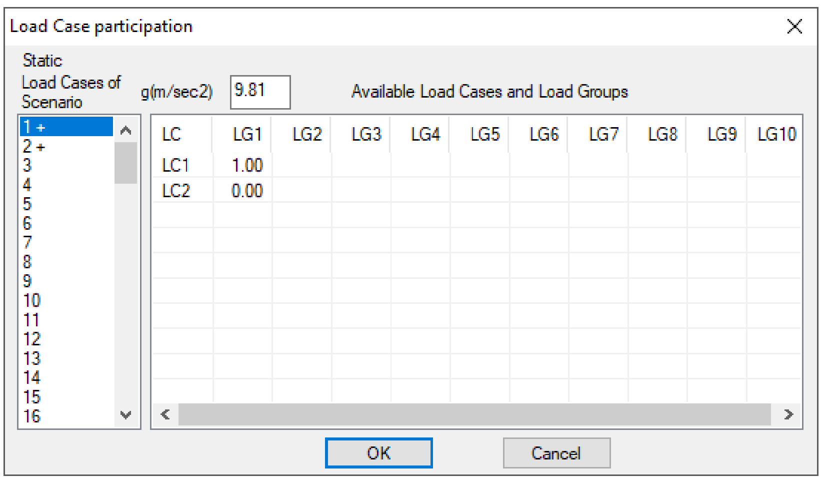
Elements, the values of the multipliers are automatically updated.



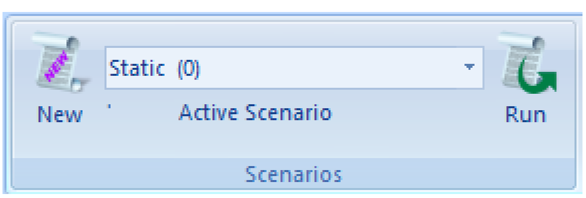
In ***Load Cases***:

1: LC1 →1

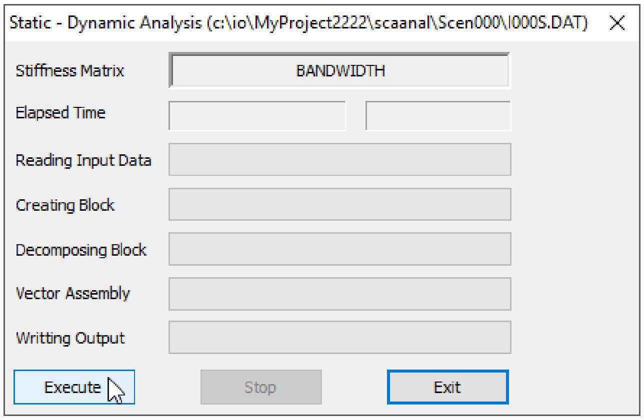
2: LC2 →1



When the active scenarios are **Static**



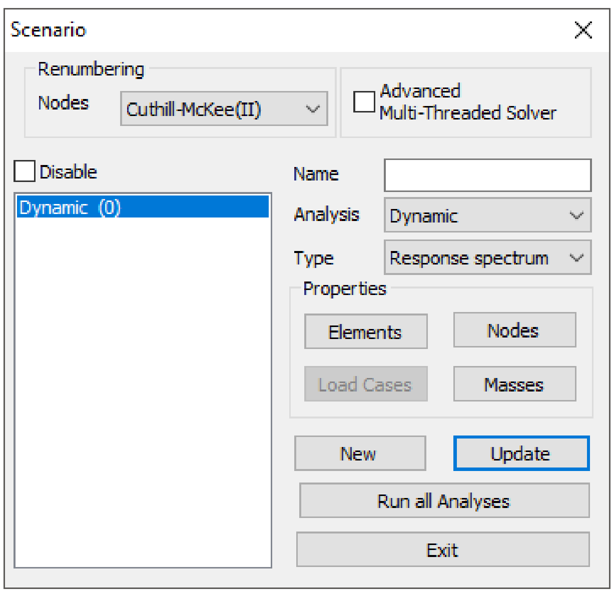
The command ***Run*** opens the following window:



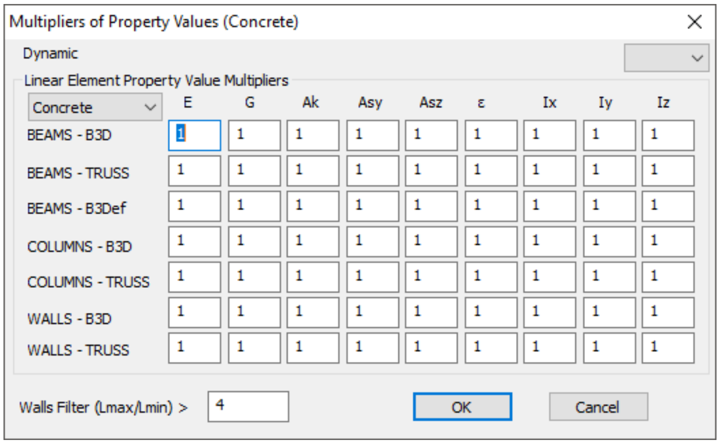
The analysis runs by pressing the Execute button. Then press Exit.

§ Scenario Dynamic

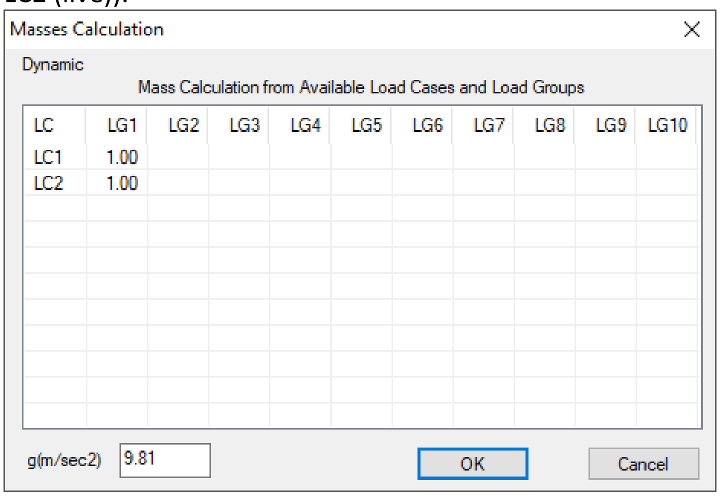
Select Analysis **Dynamic** and Type **Response spectrum** and press New.



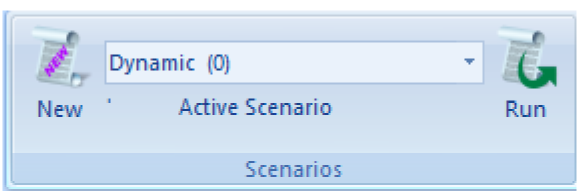
In ***Elements,*** the values of the multipliers are automatically updated.



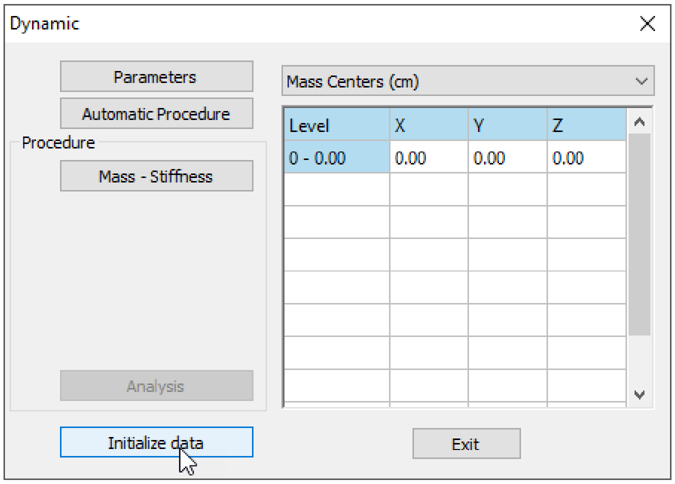
In ***Masses***, set the coefficients for calculating the masses from the available loads (LC1 (dead), LC2 (live)).



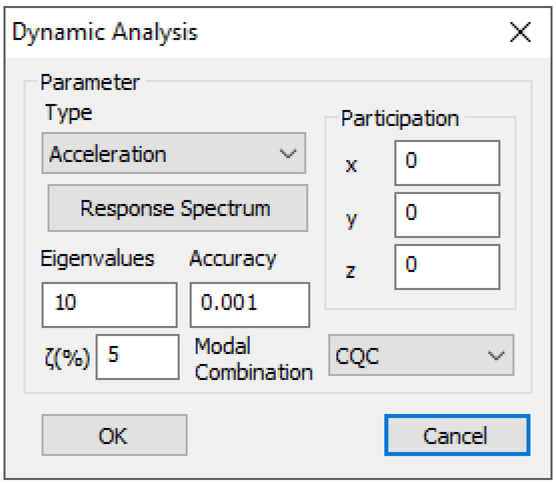
When the active scenario is **Dynamic**



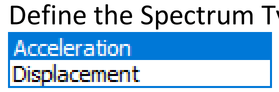
Since you press the command *Run,* the window for running the scenario opens and by pressing ***Initialize data***, the following commands are activated:

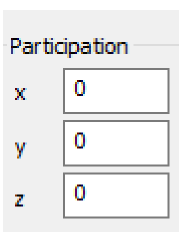


Press the command ***Parameters*** and define:

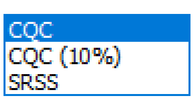


Define the Spectrum Type:

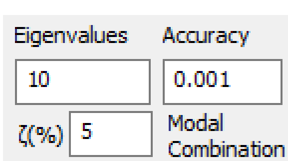




Define the Seismic Force Participation Factor per direction.

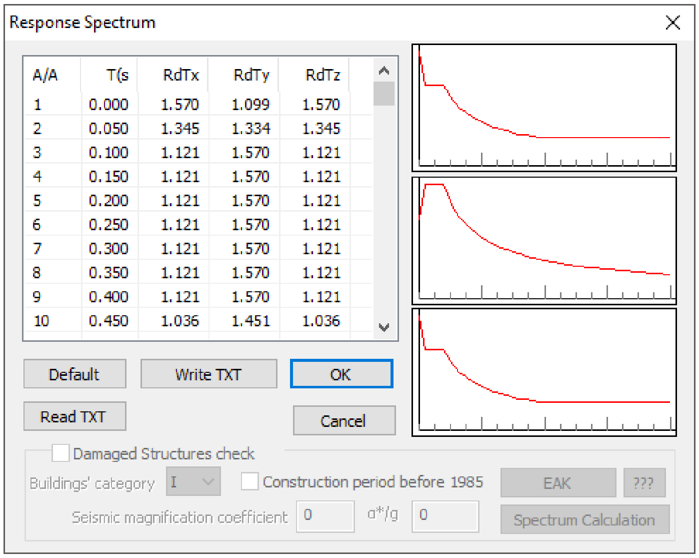


Define the method of combining the modal responses according to either Complete Quadratic Combination CQC and CQC (10%)(3.6 EAK), or the square root of the sum of squared (SRSS) method.

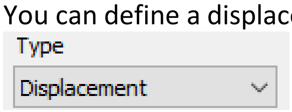


Define the number of the Eigenvalues, the Accuracy and the damping ratio ζ (%).

Click Response Spectrum to see the spectrum or change it by changing the values of the table:

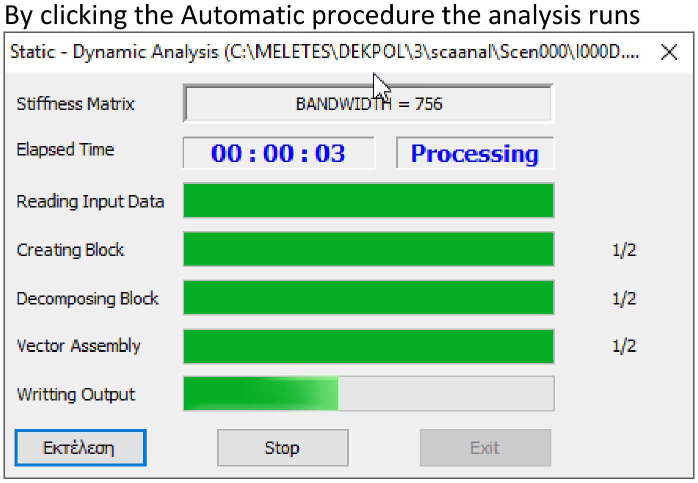


The commands Write TXT και Read TXT allow the recording and the opening respectively of a .txt file which contains all the values of the response spectrum.
You can define a displacement spectrum:

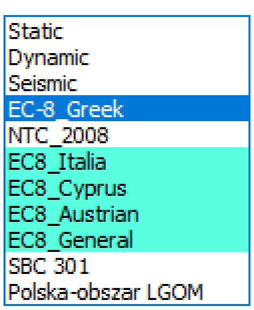


and choose a.txt displacement file to create the Displacement Response Spectrum.

By clicking the Automatic procedure the analysis runs

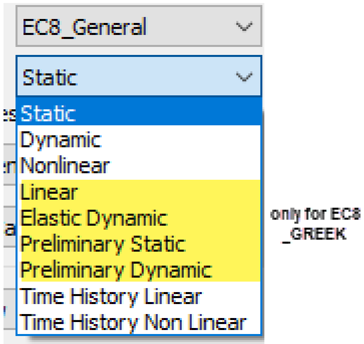


§ Eurocode Scenarios



SCADA Pro contains Eurocode 8 in its general form (EC-8_General), while it also incorporates the national annexes for Greece (EC-8_Greek), Cyprus (EC-8_Cyprus), Italy (EC-8_Italia) and Austria (EC-8_Austrian).

In the option of scenarios’ creation and analysis type “EC8_General”, there are the following types of analysis scenarios:



The types:

- Static
- Dynamic

Are used for the analysis of new structures according to EC8.

The types:

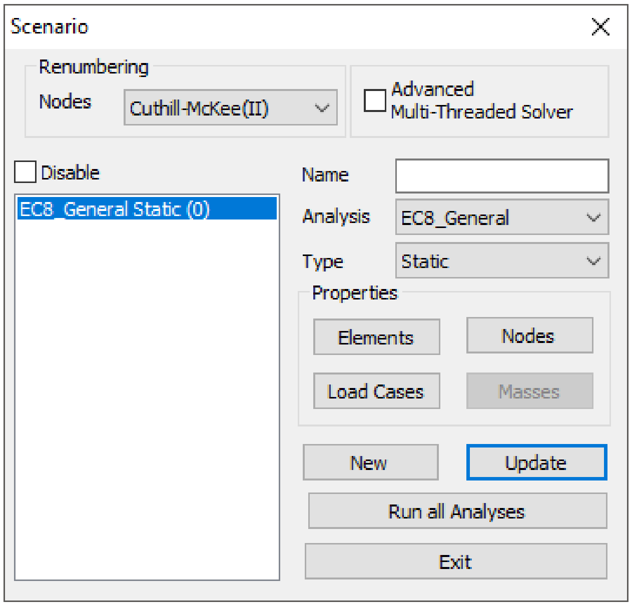
- Elastic Static
- Elastic Dynamic
- Preliminary Static
- Preliminary Dynamic

Are used for the evaluation and the redesign of existing structures ONLY for the Greek Regulation

§ 1. EC-8_General Analysis and Static Type

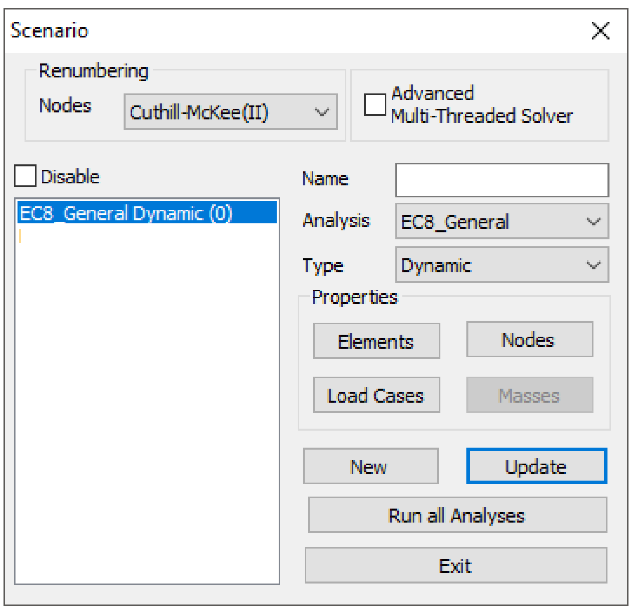
Choose Analysis **EC-8_General** and Type **Static** and press the *New* button.

ATTENTION: *The materials must be related to the selected regulation and the cross sections during the data input must have the right qualities (C for EC8 scenarios)*



§ 2. Ανάλυση EC-8_General και Τύπο Dynamic

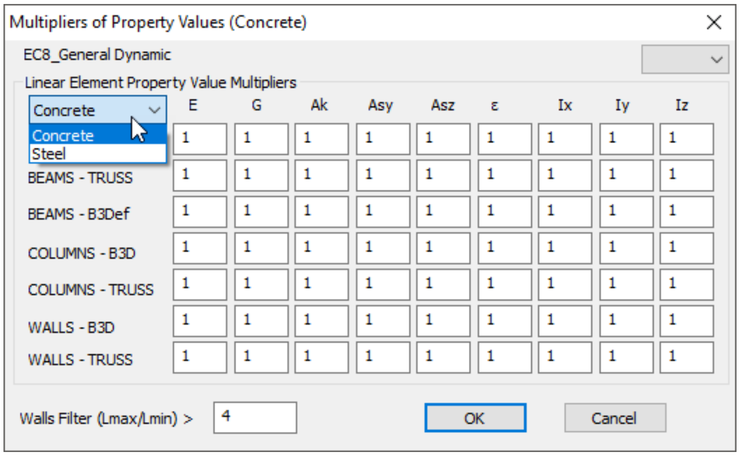
Choose **EC-8_General** Analysis and **Dynamic** Type and press the *New* button.



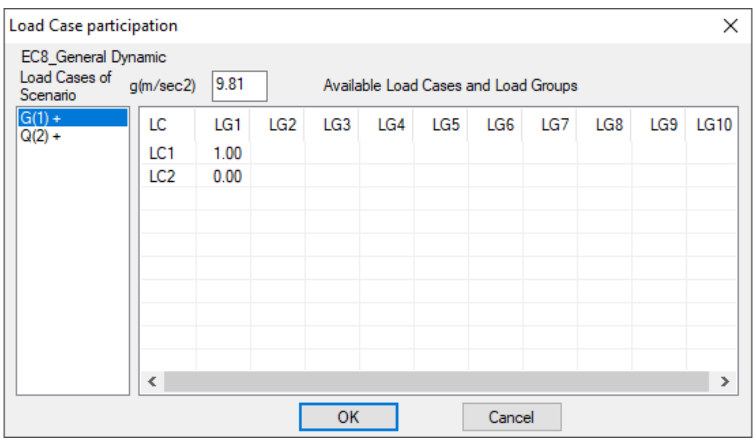
§ 1&2. EC-8_General Static Analysis and EC-8_General Dynamic Analysis

All of the following concern the **EC-8_General** both for **Static** and **Dynamic** type so they are described once for both.

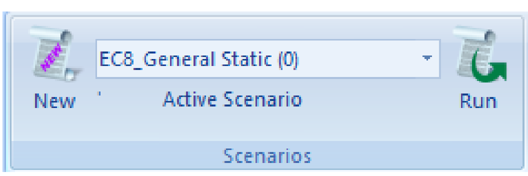
In ***Members*** you have to update the corresponding coefficients according to the annex of your country for Concrete and Steel respectively:

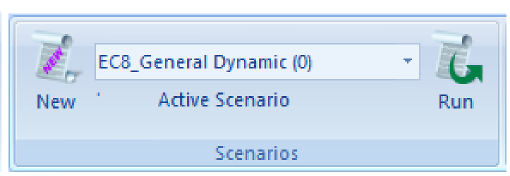


In ***Load Cases***, type 1.00 next to LC1 for "Dead Loads" and 1.00 next to LC2 for "Live Loads" Q and press the button Update.

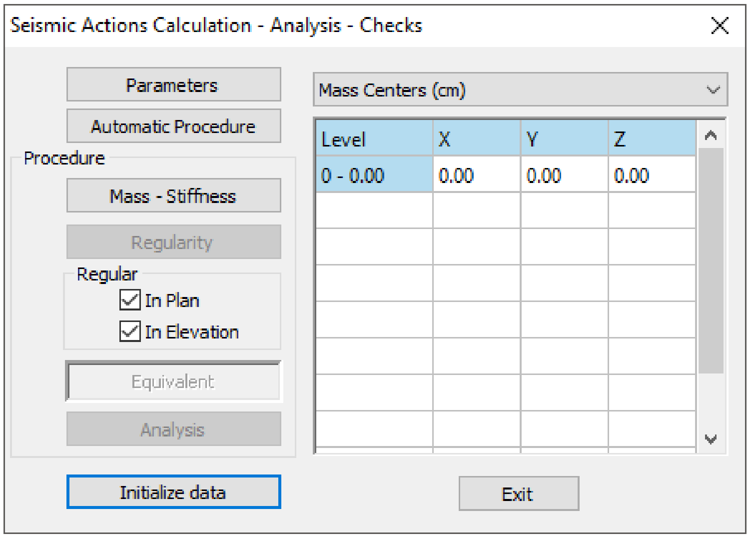


By activating either the **EC-8_General Static** scenario or the **EC-8_General Dynamic** scenario,

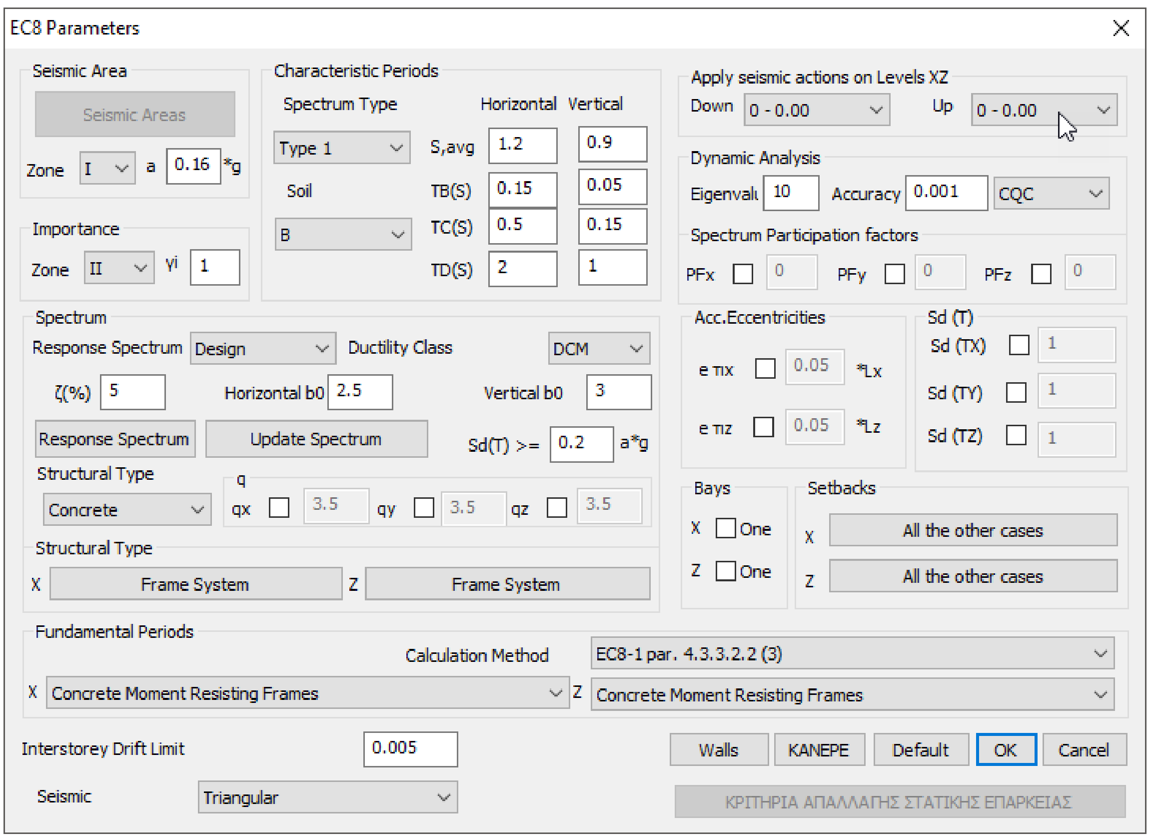


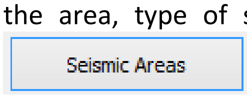


the command *Run* opens the window for the scenario's run and by pressing **Initialize Data**, the following commands are activated:

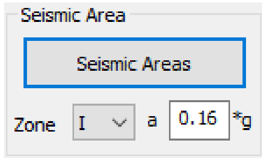


For the “EC8_General (static/dynamic)” scenario, the parameters’ dialog box is the following:

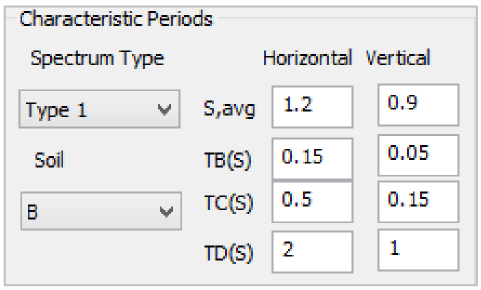


Special parameters for a specific analysis are determined in this dialog box (level of seismicity of the area, type of soil, the importance of the structure etc.). By clicking “**Seismic areas**” 

a file that contains a list taken by the national annex, with the places and their corresponding seismicity zone, pops up.

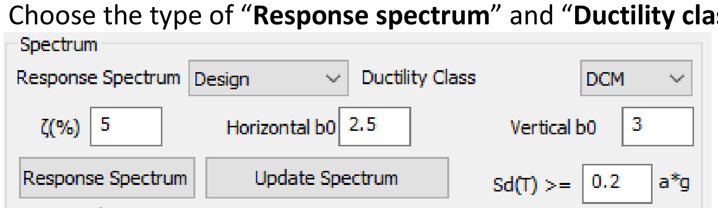


Select the considered seismic zone and the coefficient "a" will be filled in automatically.

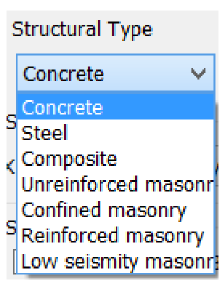


Define the Spectrum Type (for Greece Type 1) and the Soil Type so that all the coefficients for both horizontal and vertical spectrums are filled in

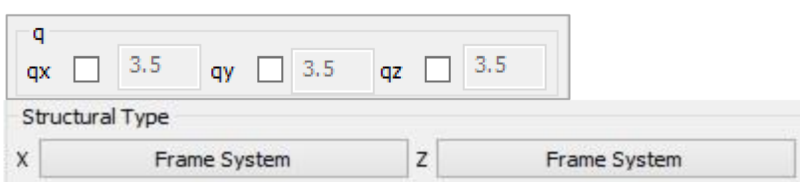
Choose the type of "**Response spectrum**" and "**Ductility class**" to suit your analysis



Choose the "**Structural Type**



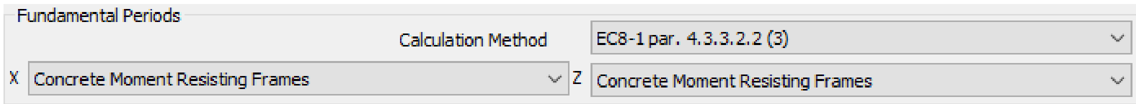
*The "**Behavior factor q**" of the structure is a result of a computation procedure. Additionally, the "**Structure type**" follows certain criteria*



SCADAPro gives the engineer the opportunity to get rid of them and follow the procedure described in the next chapter: "How to calculate the behavior factor q"

In the field ***Structure periods:***

In previous versions, there was the **Structure Type** X and Z field to calculate the fundamental period. Now it is replaced by the section:

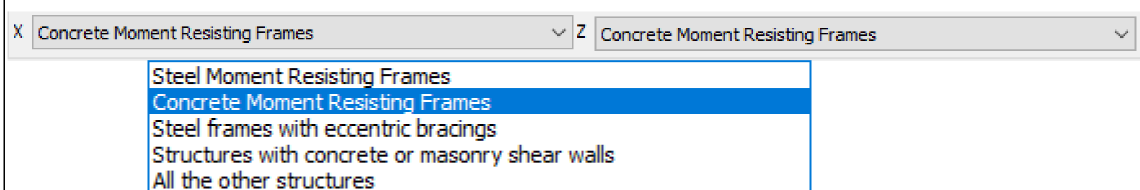


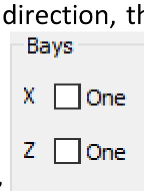
There is now an opportunity to calculate the period in three ways.

The first two are the approximate methods of EC8-1.

§ In the first one 

it is necessary:
To choose, per direction, the structure type

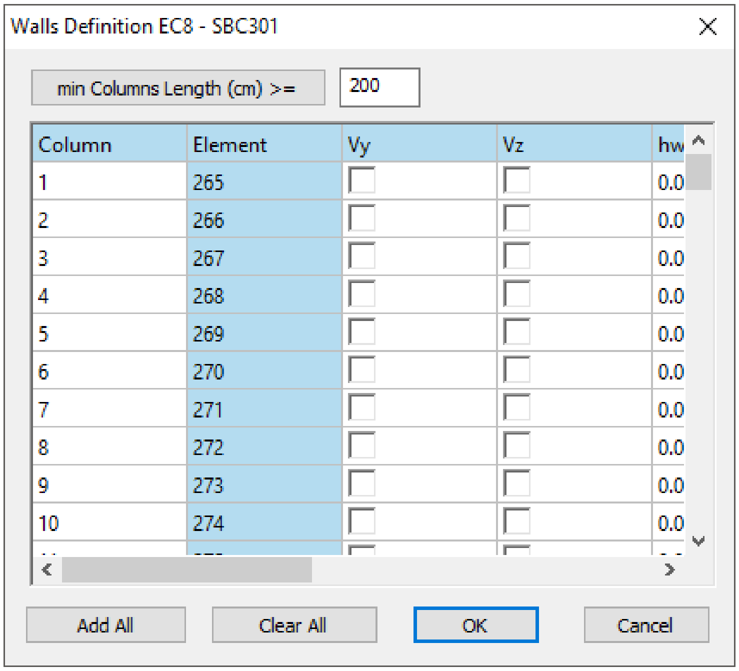


(in case that in X or/and Z direction, the structure only consists of one frame you activate the checkbox in the field “Bays” 

)

Afterwards, choose the command “Walls” 

to assign a value to the minimum length that a vertical member must have to be regarded as a wall instead of a column

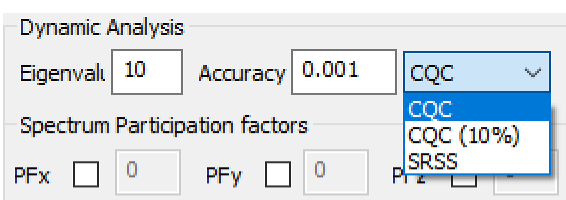


Type the min wall length (cm) and Click the [min Wall Length (cm) >=] button, and automatically, all the walls are checked in each direction, so as T1 is calculated according to paragraph 4.3.3.2.2

- For the second approximate method [EC8-1 par. 4.3.3.2.2 (5)], there is no need to do any further action as long as it is selected.
- The third method includes a Modal Analysis to calculate the periods.

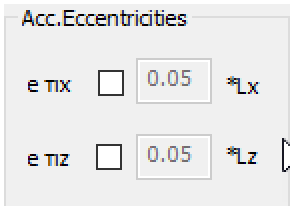
The program takes into consideration the period which corresponds to the dominant modal in each direction. (the modal which has the biggest percentage of the activated mass)

The user can increase or decrease the number of eigenvalues in case of dynamic or static analysis, as long as the calculation of the eigenvalues with Modal Analysis and the percentage of accuracy is chosen.

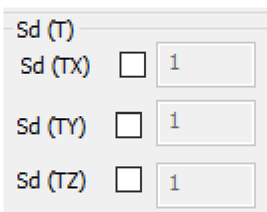


Also, there is also the opportunity to choose the method of combining the modal responses according to Complete Quadratic Combination CQC and CQC (10%)(3.6 EAK), or the square root of the sum of squared (SRSS) method.
Moreover, the results of the modal analysis for the static scenarios are included in the results of seismic action.

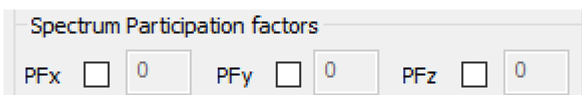
To modify the coefficients of the eccentricities, select the respective checkbox and type the new value on the right.



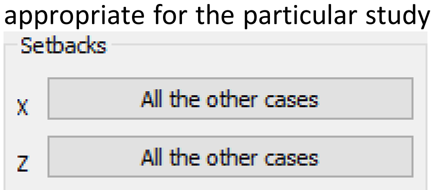
In the same way, the engineer can modify the X, Y, and Z spectrums by typing his values in the respective fields,

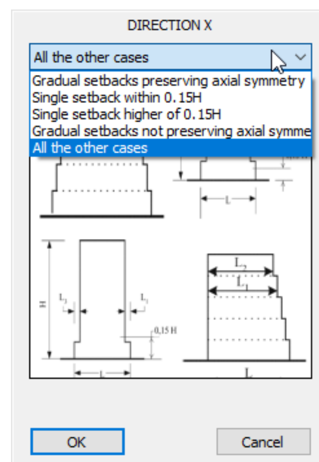


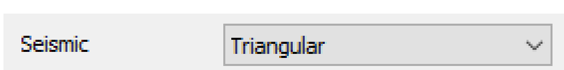
as well as the spectrum participation factors.



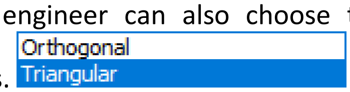
In the **Indents field**, select for each direction the case that is appropriate for the particular study and is defined by the Eurocode.







The engineer can also choose the ***Type of Distribution*** of the seismic force between two options.

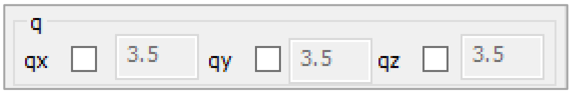


Method of calculating the behavior factor q

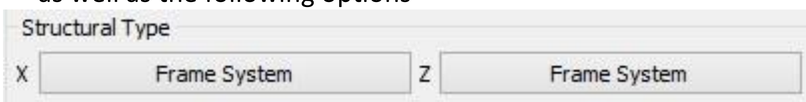
According to the Eurocode, *the “**Behavior factor q**” of the structure is a result of a computation procedure. Additionally, the “**Structure type**” follows certain criteria*.

SCADA Pro calculates automatically the q factor and the type of the structure. To apply the automatic process, you must follow the procedure described bellow:

After having completed all the previously mentioned values, leave the following boxes blank

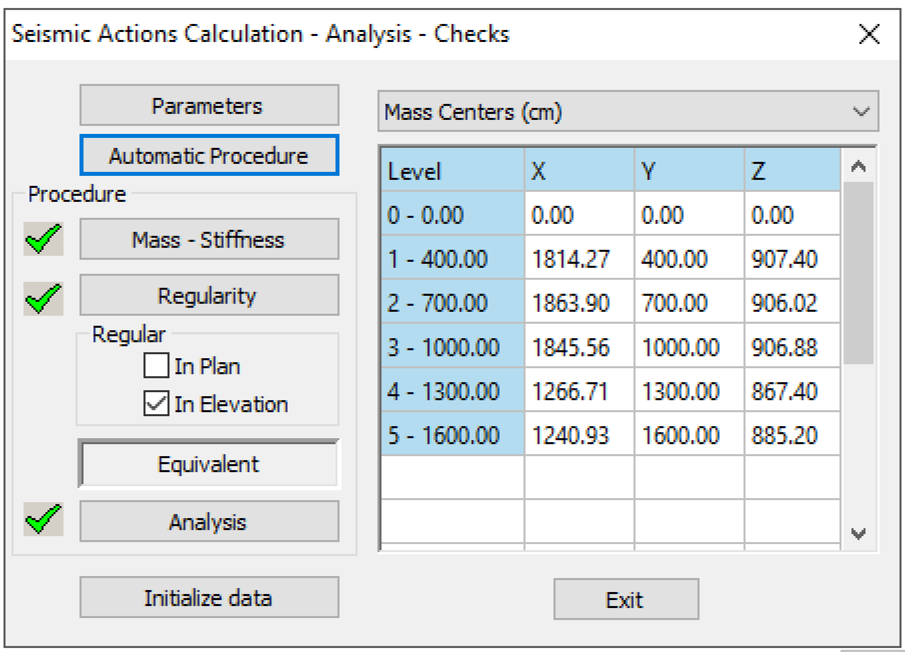


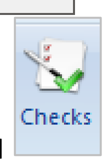
as well as the following options



without any changes.

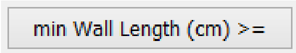
Choose “**Ok**” and use the “**Automatic procedure**” to run an initial analysis.



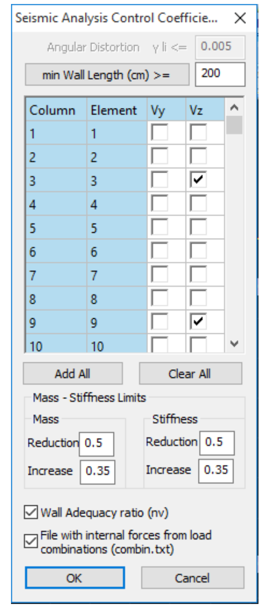
Click “Exit” to close the dialog box and choose the “Checks” command 

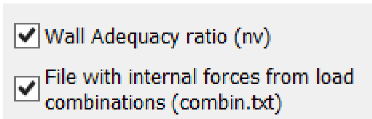
of the “Results” menu at the ribbon, to open the “Seismic analysis control coefficients” dialog box.

In the dialog box "**Seismic analysis control coefficients**" you are asked to assign a value for the minimum length that a vertical member must have to be regarded as a wall instead of a column. Click the



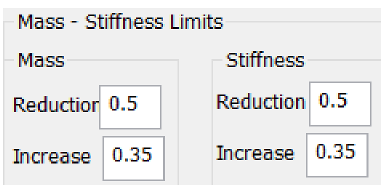
button, and automatically, all the walls are checked in each direction.





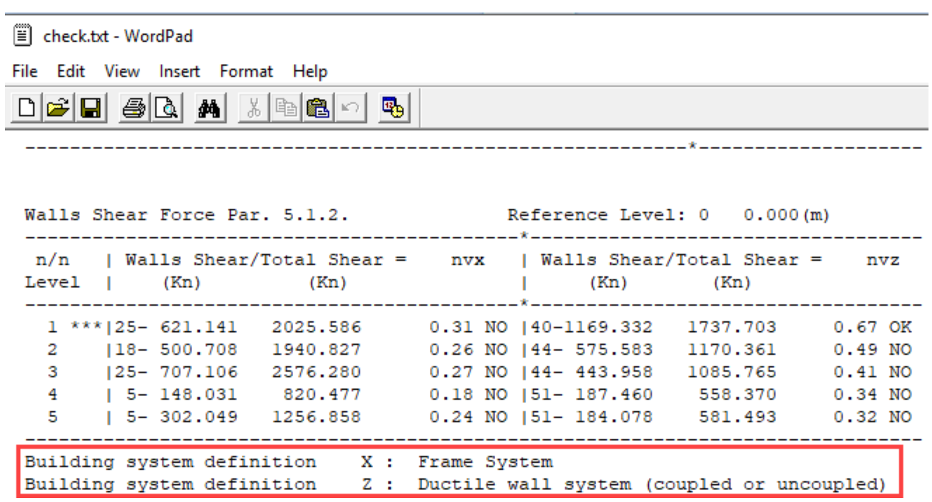
Additionally, by checking the boxes next to the two last options, two .txt files will be created and saved to the folder of the project, ready to be viewed or printed afterwards.

As far as the "**Wall adequacy**" is concerned, the relevant .txt file contains the computation of the shear acting to each wall, at each level of the structure and for all the load combinations considered.



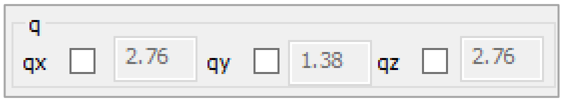
The "**Mass – Stiffness limits**" area, since no specific limitation is prescribed by EC8 (in contrast with EAK – Greek antiseismic regulation), modifications may be incorporated to those limits. Consequently, the building's regularity state in elevation will be altered, too.

In the "Checks" file, the program "defines" the structural type by the base shear undertaken by the walls.

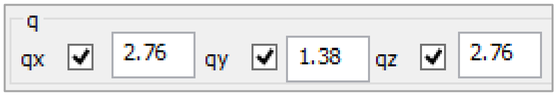


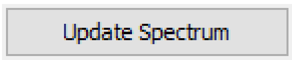

```
-----------------------------------------------------------*--------------------

Walls Shear Force Par. 5.1.2.             Reference Level: 0    0.000(m)
-------------------------------------------*-------------------------------------
 n/n    | Walls Shear/Total Shear =     nvx    | Walls Shear/Total Shear =      nvz
Level   |      (Kn)            (Kn)            |      (Kn)           (Kn)
-------------------------------------------*-------------------------------------
   1 ***|25- 621.141    2025.586      0.31 NO |40-1169.332    1737.703      0.67 OK
   2    |18- 500.708    1940.827      0.26 NO |44- 575.583    1170.361      0.49 NO
   3    |25- 707.106    2576.280      0.27 NO |44- 443.958    1085.765      0.41 NO
   4    | 5- 148.031     820.477      0.18 NO |51- 187.460     558.370      0.34 NO
   5    | 5- 302.049    1256.858      0.24 NO |51- 184.078     581.493      0.32 NO
---------------------------------------------------------------------------------
Building system definition      X :  Frame System
Building system definition      Z :  Ductile wall system (coupled or uncoupled)
```

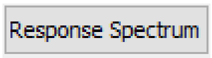
Since the **"Building system definition"** has been determined, it should be included in the **"Parameters"** dialog box. With these changes, conduct the analysis for the second time. Now, the proposed values for the **"Behavior coefficient q"** can be found in the "Parameters" dialog box. For the example considered, in the **"q"** area, one can read.



The proposed values may be kept or altered (the latter one is an option that could be utilized from the beginning of the procedure, however, in this occasion the software would not propose any values, at all).



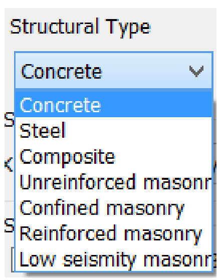
Click 

to update the spectrum by the new values of the q factor and click 

to see it.

Click “Ok” and conduct the analysis one more time, considering the new q values.

⚠ In case of Steel, Composite and Masonry Structures:



Just select the relative Structural Type:

- **Steel structures**: the procedure is the same. The only difference is located at the definition of the "Structural Type”, which is identified according to the type of the structure, thus, the help TXT file is not necessary. The user can select the type from the beginning and continues like before with the definition of the "q" value.

- **Masonry structures**: any "Structural Type" selection. Just select the type of masonry in “Structural Type” and the “q” value is calculated automatically by the program.

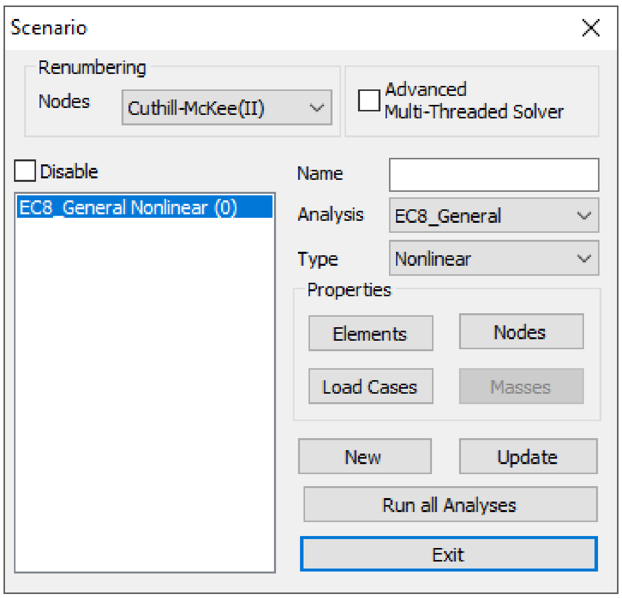
§ 3. EC-8_General Analysis and Inelastic (Pushover) Type

Choose **EC-8_General** Analysis and **Inelastic** Type and press the button *New*.

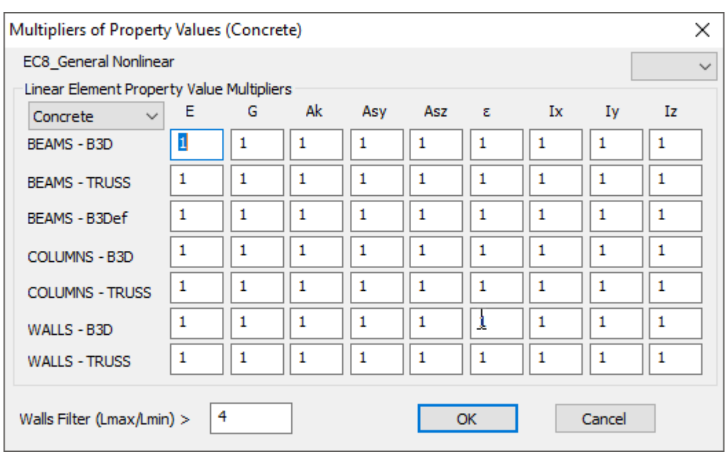
***A prerequisite** for performing all analysis scenarios with **Inelastic** Type is:*

- ***the existance of reinforcement***
- ***the calculation of the respective ultimate moments of resistance.***

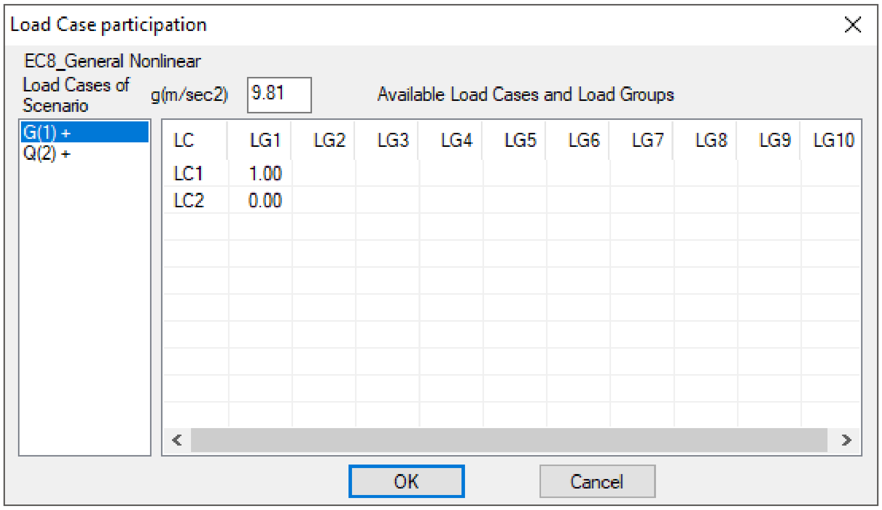
⚠ For Greece, choose EC-8 Greek / Nonlinear and for Cyprus, Italy and Austria take into consideration the Annexes of the Eurocodes, respectively. For all other European countries, choose the EC-8 General and type the parameters of the corresponding annexes.



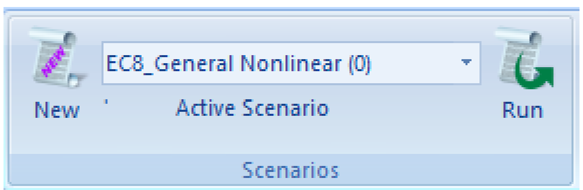
In ***Members,*** you have to update the corresponding coefficients according to the annex of your country for Concrete and Steel respectively



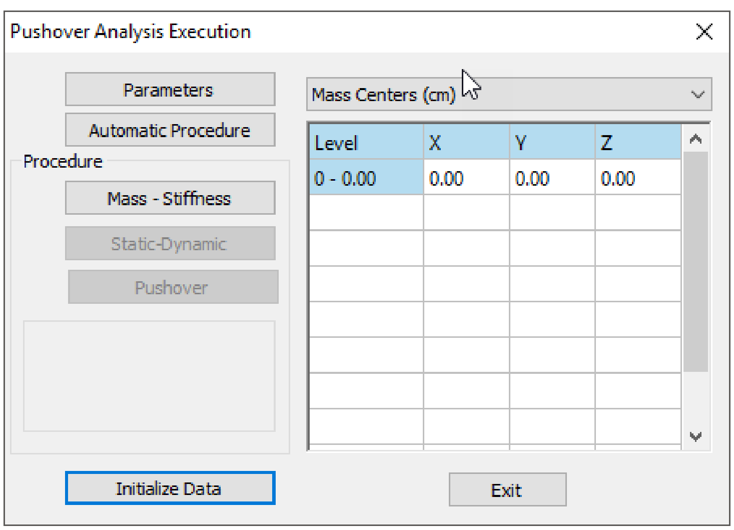
In ***Load Cases***, type 1.00 next to LC1 for “Dead Loads” and 1.00 next to LC2 for “Live Loads” Q and press the button Update.



By activating the **EC-8_General Nonlinear scenario**,



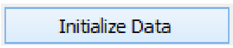
the command *Run* opens the window for the scenario’s run and by pressing **Initialize Data**, the following commands are activated:
:



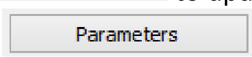
The procedure involves three steps, carried out sequentially, either automatically (“Automatic Procedure”) or selectively (click the procedure’s buttons) and contain:

- Mass and stiffness calculation.
- Static analysis for the internal forces calculation due to permanent and live loads is required for starting a successive pushover analysis.
- Dynamic analysis considering the elastic spectrum of EC8 for the calculation of the eigenperiods and the target displacement.
- Pushover analysis.

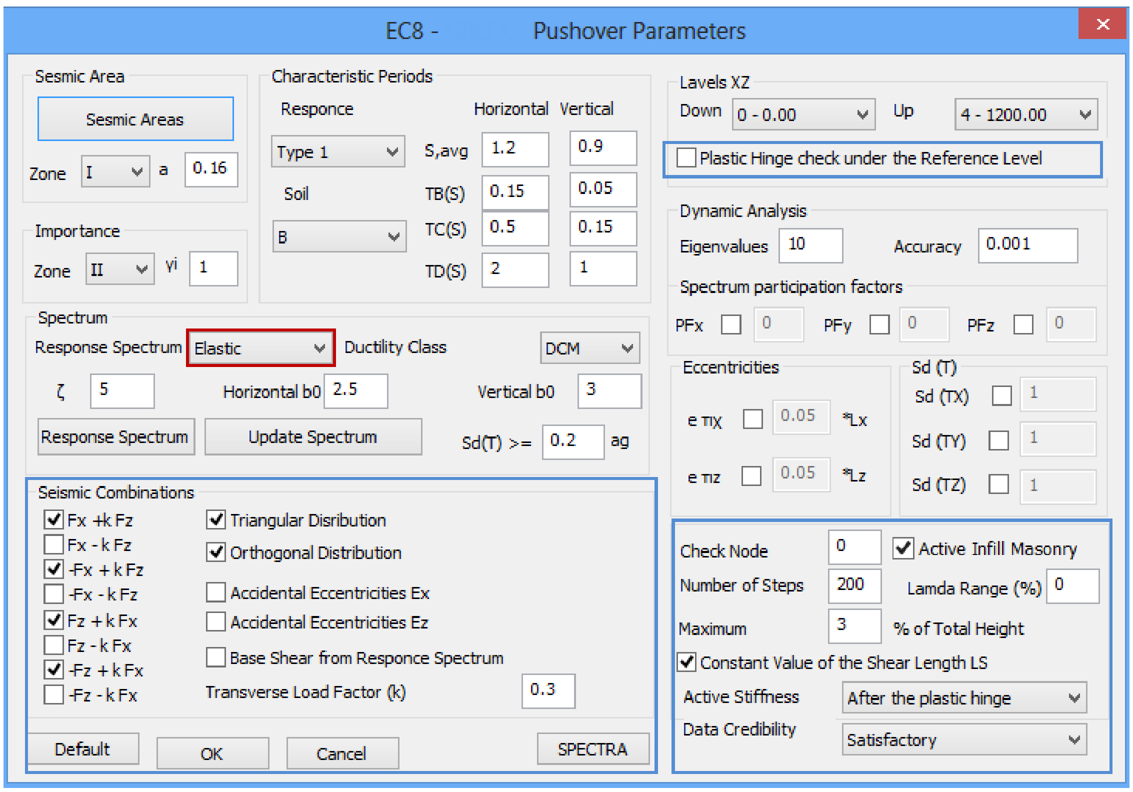
Always define the live loads' coefficient ψ2.
Default value: ψ2=0.30.

Press 

to update the parameters of the current scenario.

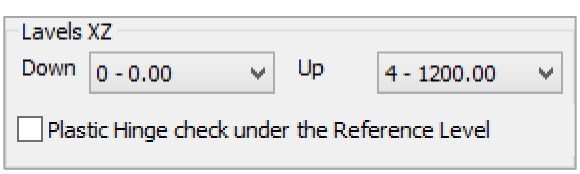
Then press 

to define the parameters of this specific project.



In the dialog box of the Pushover Analysis' Parameters, the definition of the parameters outside the borders are defined as in linear analysis, but in this case, the Response Spectrum must be Elastic.

In "Levels XZ" section:

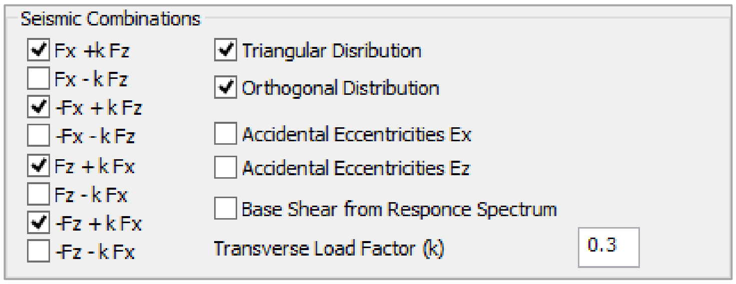


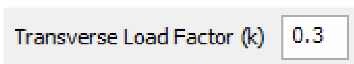
Select the lowest and the highest level in which the seismic actions will be imposed for the analysis' purposes.
It is recommended to define the last complete level (no stairwell termination)

as the highest level. This level contains the Check Node which is the diaphragm node or another node, in the same level but the outer perimeter of the building.

The activation of the following checkbox/means that the elements that belong in lower levels than the reference level are taken into consideration as potential locations of plastic hinges.

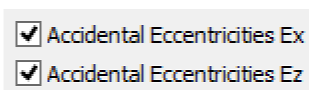
In “Seismic Combinations” section:



Check the load combinations that will be taken into account in the pushover analysis. Each combination implies the application of one seismic force in one direction (X or Z direction) and one seismic force in the corresponding transverse direction multiplied by a factor, defined in 

. The coefficient k is given 0.3 by default.

Check the seismic forces’ distribution along height (Triangular or/and Orthogonal).

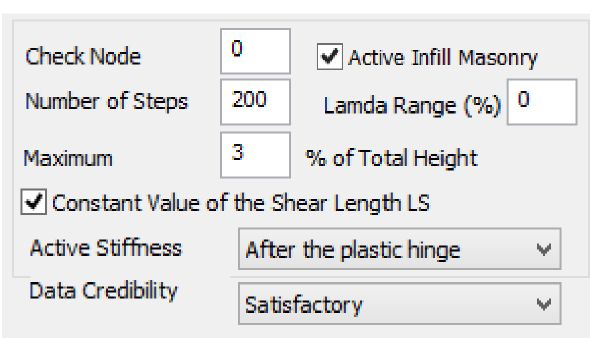
Check 

if you want to consider the accidental Eccentricities due to which moments are developed in x or/and z direction.

The check/means that the base shear is calculated from the dynamic analysis.

Activate all seismic combinations with the accidental eccentricities considered and as a result, 64 load combinations are produced. This means that 64 pushover analyses are conducted, thus, the computational time is increased significantly.

In the last section:



In “Check Node”, type the number of the node for calculation of the Capacity Curve. This is the diaphragm node or another node in the outer perimeter of the last complete level of the building.

In “Number of Steps”, define the maximum number of steps (analyses) for each Pushover analysis.

Pushover is an iterative analysis procedure which is completed when the structure collapses; when plastic hinges are being developed, a collapse mechanism evolves. The “Number of Steps” is an upper limit, to avoid an extraordinary number of analysis steps needed until the structural collapse. Steps default value is 200.

“Maximum displacement” in percentage (%) of the total height of the building is another way of setting an upper limit in the analysis steps. The Pushover iterative process ends when the displacement of the “Check Node” versus the height of the structure gets over the specified percentage. By default, the maximum displacement’s upper value is 3% of the total height of the building.

“Lamda Range (%)” is about λ load factor. In each step and for each element, the λ factor is calculated. At the element with the minimum λ, a plastic hinge will be created.

- The default value 0: the program considers only the minimum λ value, which means that in each step only one element is considered for the formation of a plastic hinge, even if there are elements with λ values very close to that.
- Setting a different value (i.e. λmin=10%): This means that the elements with values of λ between λmin and 10% + λmin will be considered in this step with plastic hinges in the corresponding edge (element node).

EXAMPLE:

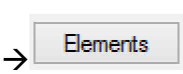
Suppose that, on the first step of the pushover, minimum λ value is 1.0 and corresponds to a certain element with a plastic hinge. Defining 10% Lamda Range, all elements with λ between 1.0 and 1.1 will develop plastic hinges at the same step.

The options about the definition of the “Constant value of the Shear Length Ls” are the following:

- The shear length Ls is calculated as a fixed value based on the length of each element in all steps of the analysis.
- Otherwise, the shear length is calculated on each step, based on the internal forces resulting, with Shear Length = M/V in the end section of the element, i.e. the distance of the end section from the point of zero moments.

“Effective Stiffness” is about the calculation of the elements’ stiffness.

On the first step of the nonlinear analysis, the internal forces are calculated due to dead and live loads, whereas the elements’ stiffness is multiplied by the factors set in 

→ 

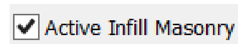
during scenario’s creation.

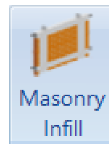
On the second step of the nonlinear analysis considering the seismic loads, you have more than one options for the stiffness calculation:

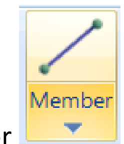
- Initial: the stiffness remains the same as in the first step and invariable during the analysis procedure
- Step by step calculation: the stiffness is calculated in each step, whether plastic hinges are created or not. Stiffness is decreased, compared to the original value.

- After plastic hinge development: the stiffness is calculated as previously, with the difference that the value is decreased since the first plastic hinge has been developed. The initial value of the effective stiffness is considered until that step of the analysis.

“Data reliability”, is the Knowledge Level. Select between Limited, Normal and Full. The parameters that affect the definition of the knowledge levels are geometry, details, material. The knowledge level influences the partial safety factors.

“Active Infill Masonry”: When the checkbox is active 

and model contains Masonry Infills added with the corresponding tool 

or 

(look Chapter 2) allow the program to take them into account during the analysis. Otherwise, even if they are modeled, the program will exclude their influence.

Partial Safety Factors

Based on the knowledge level achieved through the different levels of the survey, inspection, and testing, the following set of partial safety factors (PSF) shown in Table 3.3 is used in the verifications.

Table 3.3: Partial safety factors (PSF) are used for verification, according to the different knowledge levels (KL).

	PSF	
KNOWLEDGE LEVEL	Material	Overstrength
KL1	1.20 (γm)	1.20 (γRd , γov)
KL2	γm as in EN1998-1	γRd , γov as in EN1998-1
KL3	0.80 (γm)	0.80 (γRd , γov)

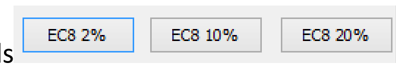
In the dialog box “**Spectra**” the response spectrum is defined, which indicates the structural demand of the structure. The structural demand in combination with a limit state (LS) composes a performance objective. In the dialog box:

Three limit states are considered for the structural design of the structure. The states of damage related to the considered limit states are presented below:
-LS of near collapse (**NC**)
-LS of significant damage (**SD**)
-LS of damage limitation (**DL**)

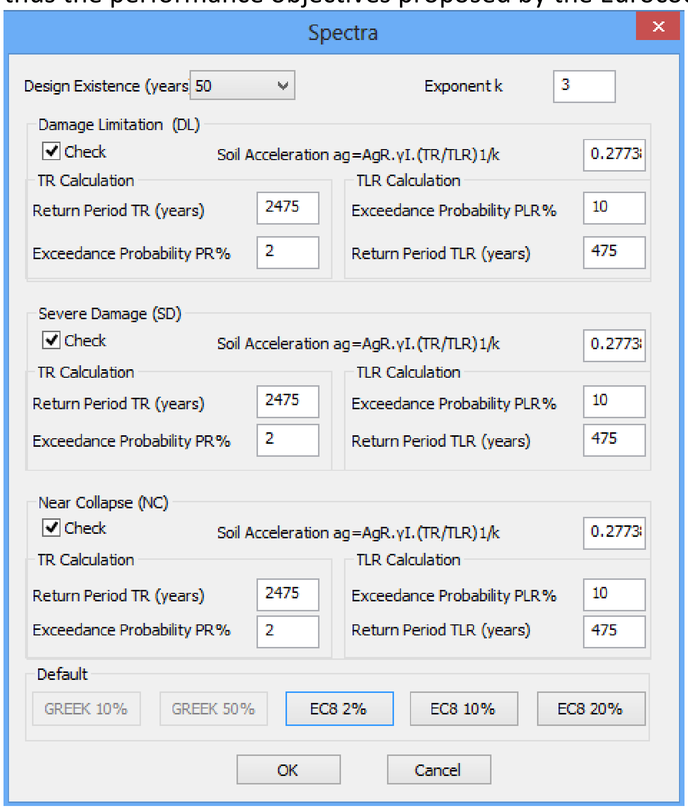
National Authorities define the appropriate performance level, which means they propose a performance level (limit state) against a seismic event. They usually propose more than one performance level (a combination of a limit state with a seismic event).

The seismic events or hazard levels (HL) are described with a return period and a peak ground acceleration. The peak ground acceleration and the incidence of the seismic events are characteristic of the seismicity of the region. The return periods established by the National Authorities are usually the following:

The suggested values for the return periods according to Eurocode are:
- HL with return period 2475 years, corresponding to a probability of exceedance 2% in 50 years.
- HL with return period 475 years, corresponding to a probability of exceedance 10% in 50 years.
- HL with return period 225 years, corresponding to a probability of exceedance 20% in 50 years.

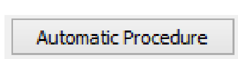
Selecting one of the three default commands 

, the dialog box is filled in automaticaly, with the values of the spectra parameters related to a specific limit state, thus the performance objectives proposed by the Eurocodes are depicted in the dialog box.

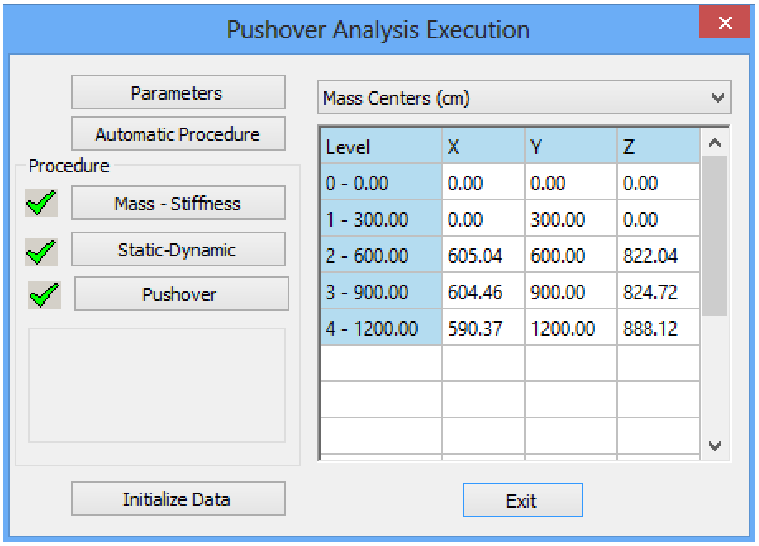


Use National Annex to define the appropriate performance levels; the structural capacity (limit state) related to the seismic demand (hazard level).

Press OK to save the Parameters and close the dialog box.

Then select 

and the program will conduct the analysis and the corresponding calculations, automatically:



Allow the program to calculate Mass and Stiffness, Static and Dynamic analysis and complete the Pushover analysis. The default parameters include four combinations with two types of load distribution and 200 steps for each Pushover analysis, thus 1600 analyses in total!

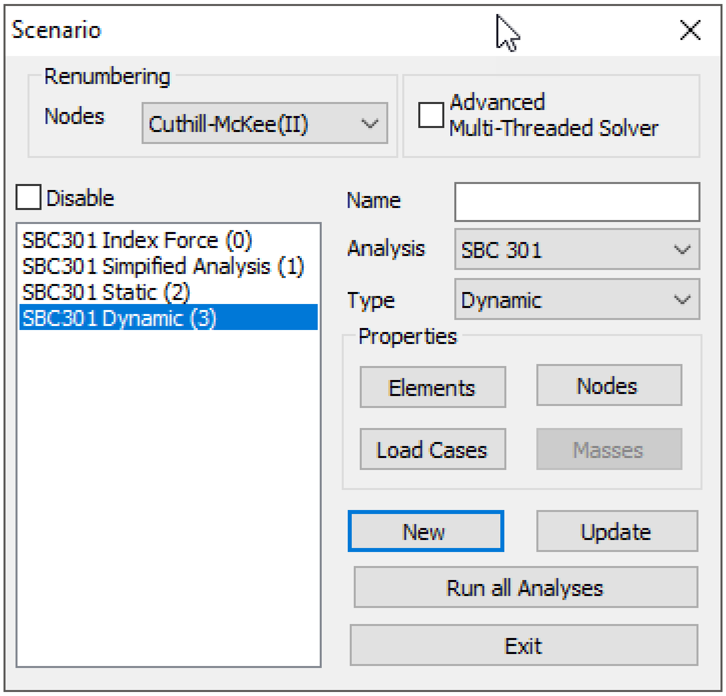
Since the Pushover analysis is applied, the structure is pushed with a gradually increased lateral static load (triangular or rectangular distribution along height) till the collapse of the structure. So, plastic hinges are formed gradually in the ends of the structural elements’ length (beam, column, and wall). Along with the plastic hinges’ formation, the stiffness of the structural elements and the structure in total is decreased step by step. Finally, a plastic mechanism is generated due to the presence of a plastic region in the joints of the structure. The plastic deformations indicate that the strength resistance of the structural elements is overtaken thus the structure collapses.

§ 4. Analysis SBC301

Regulation of Saudi Arabia provides four methods of analysis for calculating seismic loads:

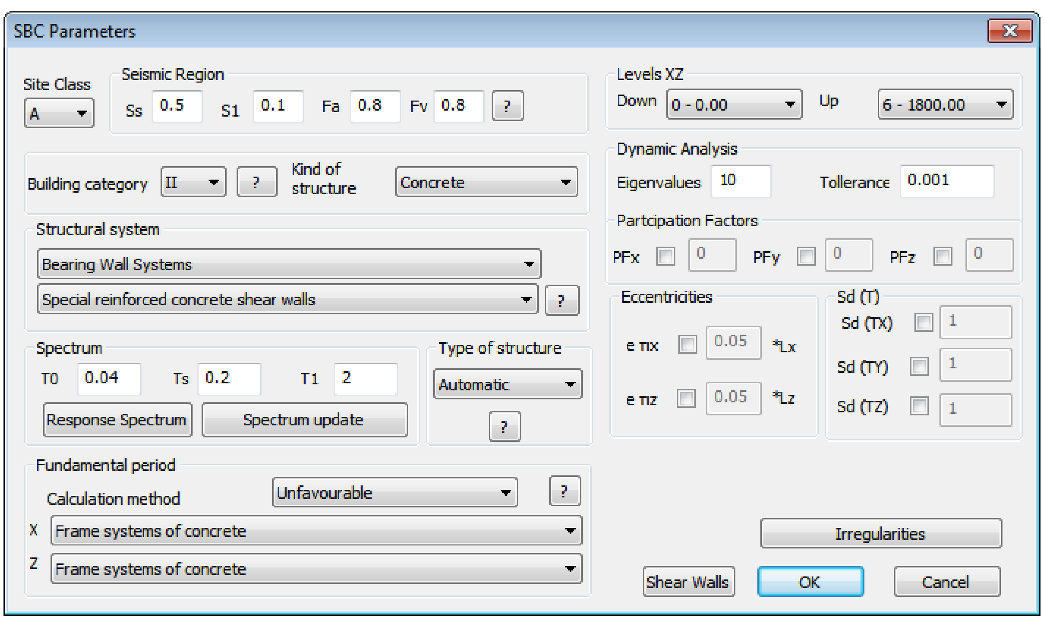
- Index Force Analysis Procedure (Section 10.7)
- Simplified Analysis Procedure (Section 10.8)
- Equivalent Lateral Force Procedure (Section 10.9)
- Modal Analysis Procedure (Section 10.10)

The process of creating these four scenarios is the same as the previously described one for the EC.
Press “New” to create the scenarios:



Select the SBC 301 analysis and then one of the four methods of analysis.

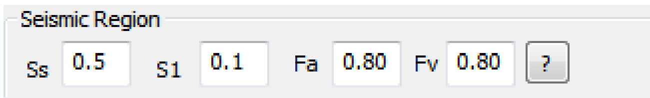
The dialog box to import and edit the parameters is the same for all four methods:



The first parameter concerns the type of soil (Site Class) 

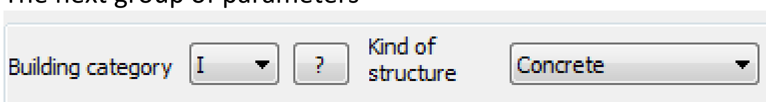
according to par. 9.4.2, which imposes the Fa and Fv factors (Table 9.4.3a & 9.4.3b) as well.

The next group of parameters regards the selection of the mapped maximum considered earthquake spectral response S1 and Ss according to par. 9.4.1.

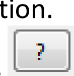


The choice of these two parameters in combination with the type of soil identifies automatically the (user editable) Fa and Fv factor values.

The next group of parameters

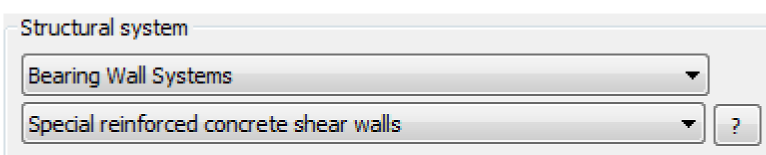


conserns the selection of Building Category according to Table 1.6-1 and the Kind of structure selection.

Press 

to read about all the Kinds of structures according to the corresponding table of SBC 301.

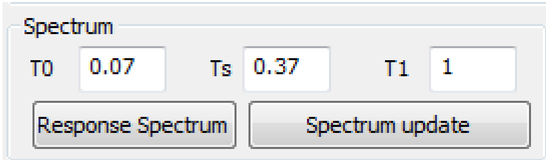
The next section



conserns the selection of the Structural System according to the table 10.2 of SBC 301.

This choice determines:
- the value of the Response modification coefficient R used in various calculations,
- the value of the Deflection amplification factor Cd used according to Sections 10.9.7.1 and 10.9.7.2 and
- the value of the overstrength factor Ωo.
- Finally, according to the seismic category of the structure, restrictions to the selection of Structural System as well as to the maximum height of the building are specified.

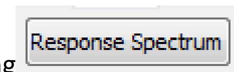
The next section conserns the parameters of the Design Response Spectrum according to par. 9.4.5 of SBC 301



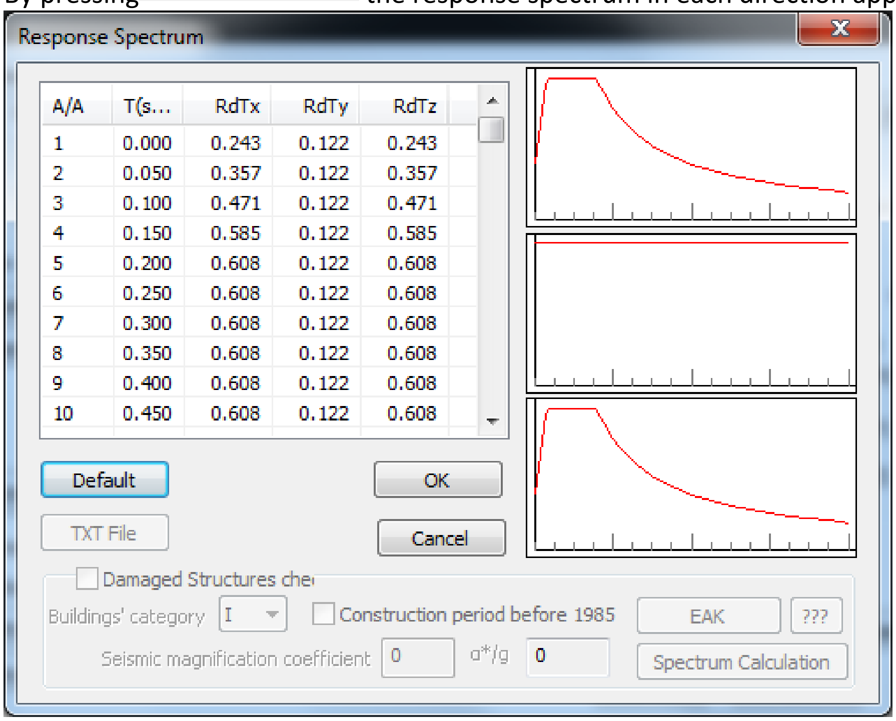
The application calculates automatically the values T0 and Ts based on the previously defined values S1 and Ss. Of course, it is possible to modify them manually.

If any changes are made on these values, either automatically or manually, to update the spectrum, press the button 

.

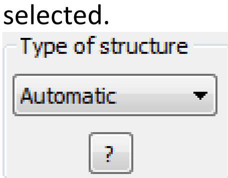
By pressing 

the response spectrum in each direction appears.



A/A	T(s...	RdTx	RdTy	RdTz
1	0.000	0.243	0.122	0.243
2	0.050	0.357	0.122	0.357
3	0.100	0.471	0.122	0.471
4	0.150	0.585	0.122	0.585
5	0.200	0.608	0.122	0.608
6	0.250	0.608	0.122	0.608
7	0.300	0.608	0.122	0.608
8	0.350	0.608	0.122	0.608
9	0.400	0.608	0.122	0.608
10	0.450	0.608	0.122	0.608

Next, the automatic or manual definition of the type of structure according to par. 10.3 is selected.

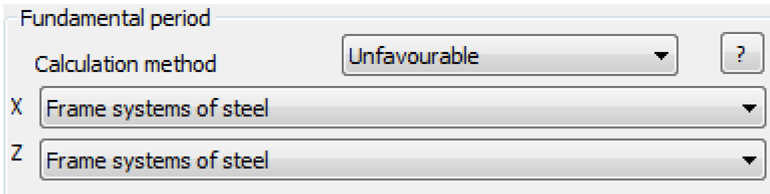


There are three choices:

- Automatic
- Flexible
- Rigid

The first choice identifies the type of structure (Flexible or Rigid) automatically according to par.10.3.1.3 and figures 10.3-1. The user selects the type through the next two choices.

The following parameters concern the calculation of the Fundamental Period.



The par. 10.9.3 of SBC 301 provides three methods for the calculation of this size. For each method to be applied, certain conditions must be met.

The program options are:

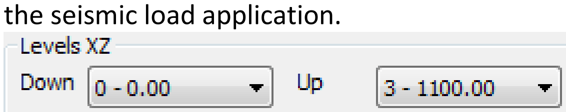
- Unfavourable
- Average
- Method 1
- Method 2
- Method 3

Through the first option, the most unfavorable period of those calculated from the three methods, will be taken into account.
Through the second option, the value is the average of the three values of the three methods.
The next three choices concern the value of the specific method.

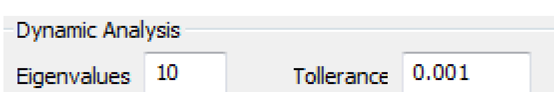
In each case, two values are calculated, one for each direction of the earthquake.

The next option is the type of building based on Table 10.9.3.2 and concerns the first method calculation (Eq. 10.9.3.2-1).

The next section concerns the determination of the lower and upper level to be considered for the seismic load application.

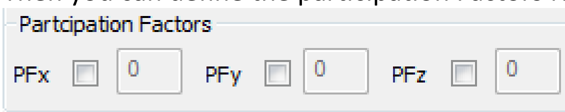


The following parameters are related to the dynamic analysis. More specifically in the following fields:



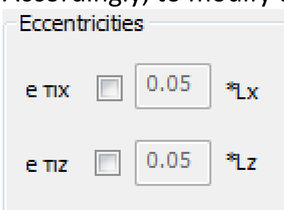
The user determines the number of Eigenvalues to be calculated through the performance of the modal analysis and the respective tolerance.

Then you can define the participation Factors for the seismic forces in each direction.

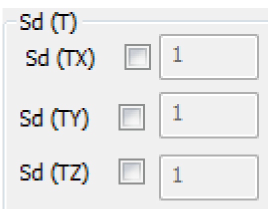


Activate the corresponding checkbox and type the factor for the seismic load. The initial spectrum (for the considered direction) will be multiplied by this factor. The default value is 1, while a value of PFx=2 will double the values of the spectrum along the X direction.

Accordingly, to modify the coefficients for calculating the eccentricities



Activate the corresponding checkbox and type in the value.
Finally under the participation rates of design spectra in each direction

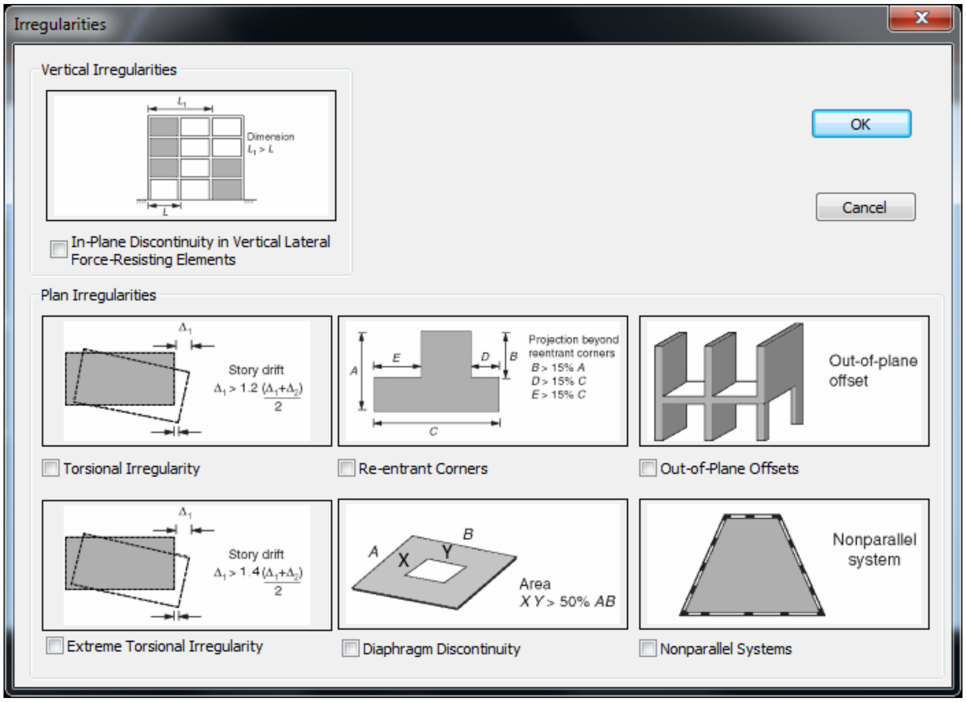


Activate the coefficient for the respective spactra and type in the value. The design spectra acceleration (for the considered direction) will be multiplied by this factor. For example, if you activate the Sd(TX) and enter a value of 2, the design acceleration along the X direction, will be multiplied by 2.

The next section concerns the determination of Plan and Vertical Irregularities of the structure. This definition, among others, determines the choice of the analysis method based on the table 10.6.1.

The SBC 301 with the Table 10.3.2.1 provides 6 Plan Irregularities.

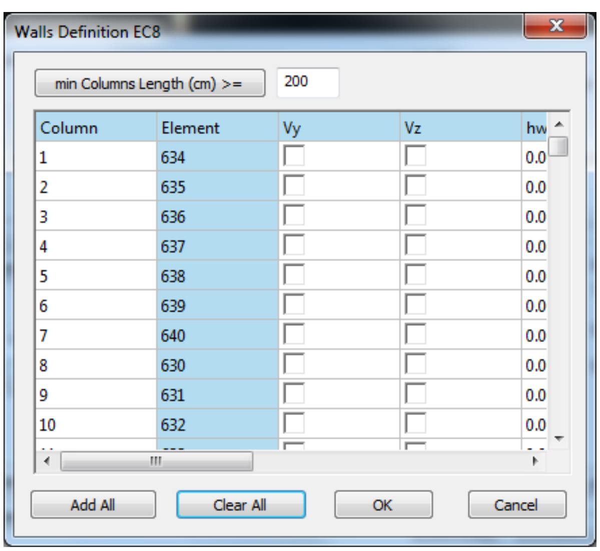
In the following dialog box:



The user determines whether Plan Irregularities meet the relevant criteria by checking the corresponding option.

Regarding Vertical Irregularities, six criteria are met as well.
SCADA Pro checks 5 of them and the user activates the 6th by pressing the respective checkbox if the criteria are fulfilled.
Option/concerns the calculation of sizes which are necessary for determining the eigenperiod with the third method (Eq. 10.9.3.2-3).

Press to open the dialog box.

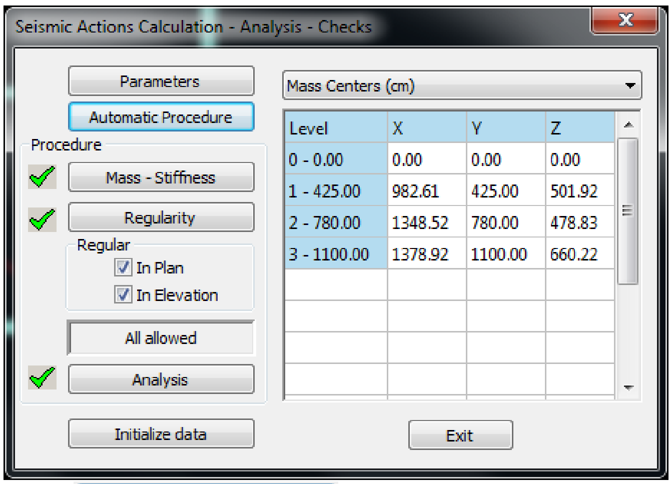


Type the min Column Length (cm) and press the button “min Column Length ” for automatically defining the walls in each direction. The program automatically calculates the required for calculating the eigenperiod values.

NOTE:

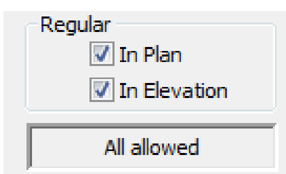
⚠ The definition of the walls is a required step for calculating the eigenperiod using the third method.

After setting the parameters, in the run analysis dialog box



press 

and the program automatically performs the analysis process while making all necessary checks.
Particular attention should be paid to regularity checks

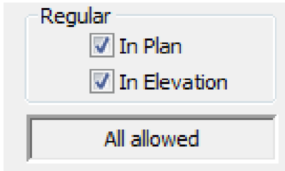


where before performing the final analysis the program gives the result of regularity in plan and height, based on the selection and seismic category in the initial parameters.

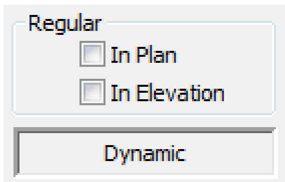
Then, based on the results, the allowed analysis or analyses are suggested (Table 10.6.1).

- On the regularity results, the user is free to select or deselect if he wants, one or two categories of regularity.
- The program changes respectively the prompt type of permitted analysis.

In the previous case, through the automatic control, the building met the criteria for regularity in plan and in elevation, which allows the use of any of the four methods.



However, if both checkboxes are deselected, which means that the building is considered irregular in plan and elevation, then the program proposes the dynamic method for analysis.



“**Checks**” command displays the results of the checks through the automatic procedure.
The first section of the checks concerns the Irregularity criteria:

```
          CHECKS REPORT ACCORDING TO THE MAIN DIRECTIONS OF THE BUILDING
                    SIMPLIFIED STATIC ANALYSIS (SBC 301)
================================================================================

------------------------------------------------------------------------------
         VERTICAL STRUCTURAL IRREGULARITIES (TABLE 10.3.2.2)
------------------------------------------------------------------------------
Check for Stiffness Irregularity per Building Story (Table 10.3.2.2 (1a & 1b))
------------------------------------------------------------------------------
n/n  | Height (m) | Stiffness X (*10^3 KNM) | Stiffness Z (*10^3 KNM) |
Story| Total|Story|   Ki     |  Ki+1    |Ratio|   Ki     |  Ki+1    |Ratio|Result
-----+------+-----+----------+----------+-----+----------+----------+-----+-------
   1 |  4.25| 4.25|  9661.63|  5077.48| 1.90|  9914.30|  4708.94| 2.11|No
   2 |  7.80| 3.55|  5077.48|  4894.28| 1.04|  4708.94|  4010.75| 1.17|No
   3 | 11.00| 3.20|  4894.28|          |     |  4010.75|          |     |
------------------------------------------------------------------------------
n/n  | Height (m) | Stiffness X (*10^3 KNM) | Stiffness Z (*10^3 KNM) |
Story| Total|Story|   Ki     |  KAvg    |Ratio|   Ki     |  KAvg    |Ratio|Result
-----+------+-----+----------+----------+-----+----------+----------+-----+-------
   1 |  4.25| 4.25|  9661.63|  4985.88| 1.94|  9914.30|  4359.85| 2.27|No
   2 |  7.80| 3.55|  5077.48|  4894.28| 1.04|  4708.94|  4010.75| 1.17|No
   3 | 11.00| 3.20|  4894.28|          |     |  4010.75|          |     |
------------------------------------------------------------------------------
Soft Story (i)          : Ki/Ki+1<0.7  or  Ki/Avg((Ki+1)+(Ki+2)+(Ki+3))<0.80
Extreme Soft Story (i) : Ki/Ki+1<0.6  or  Ki/Avg((Ki+1)+(Ki+2)+(Ki+3))<0.70
------------------------------------------------------------------------------

Check for Weight (Mass) Irregularity per Building Story (Table 10.3.2.2 (2))
------------------------------------------------------------------------------
n/n  | Height (m) |      Mass (kN/g)        |      Mass (kN/g)        |
Story| Total|Story|   Mi     |  Mi-1    |Ratio|   Mi     |  Mi+1    |Ratio|Result
-----+------+-----+----------+----------+-----+----------+----------+-----+-------
   1 |  4.25| 4.25|   295.68|     0.00| 0.00|   295.68|   331.70| 0.89|No
   2 |  7.80| 3.55|   331.70|   295.68| 1.12|   331.70|    65.07| 5.10|Yes
   3 | 11.00| 3.20|    65.07|   331.70| 0.20|    65.07|     0.00| 0.00|No
------------------------------------------------------------------------------
Irregular Story (i)  : Mi/Mi+1>1.5  or  Mi/Mi-1>1.5
------------------------------------------------------------------------------

Check for Vertical Geometric Irregularity per Build.Story (Table 10.3.2.2 (3))
------------------------------------------------------------------------------
n/n  | Height (m) |  Plan Dimensions X (m)  |  Plan Dimensions Z (m)  |
Story| Total|Story|   Li     |  Li-1    |Ratio|   Li     |  Li-1    |Ratio|Result
-----+------+-----+----------+----------+-----+----------+----------+-----+-------
   1 |  4.25| 4.25|    19.60|    19.40| 1.00|    13.70|    13.60| 1.00|No
   2 |  7.80| 3.55|    13.20|    19.60| 1.00|     9.30|    13.70| 1.00|No
   3 | 11.00| 3.20|     9.80|    13.20| 1.00|     9.00|     9.30| 1.00|No
------------------------------------------------------------------------------
Irregular Story (i)  : Li/Li+1>1.3  or  Li/Li-1>1.3
------------------------------------------------------------------------------
```

Presented data and test results of each Vertical Irregularity criterion.
At the end of this section, all the above results are presented in summary as well:

```
(1a) Stiffness Irregularity (Soft Story)                                  : No
(1b) Stiffness Irregularity (Extreme Soft Story)                          : No
(2)  Weight (Mass) Irregularity                                           : No
(3)  Vertical Geometric Irregularity                                      : No
(4)  In-Plane Discontinuity in Vertical Lateral Force-Resisting Elements : No
(5)  Discontinuity in Lateral Strength (Weak Story)                       : No
---------------------------------------------------------------------------------
Vertical Structural Irregularity for the total building                   : No
---------------------------------------------------------------------------------
```

Next, the results of the Plan Irregularities according to the user’s choices on the parameters of the analysis, are reported:

```
------------------------------------------------------------------------------
PLAN STRUCTURAL IRREGULARITIES (TABLE 10.3.2.1)
------------------------------------------------------------------------------
(1a) Torsional Irregularity                                            : No
(1b) Extreme Torsional Irregularity                                    : No
(2)  Re-entrant Corners                                                : No
(3)  Diaphragm Discontinuity                                           : No
(4)  Out-of-Plane Offsets                                              : No
(5)  Nonparallel Systems                                               : No
------------------------------------------------------------------------------
Plan Structural Irregularity for the total building                    : No
------------------------------------------------------------------------------
```

The next check indicates the percentage of the seismic forces undertaken by the walls of the structure to determine whether the static system of the structure belongs to one of the following two categories:
- Dual Systems with Special Moment Frames Capable of Resisting at Least 25% of Prescribed Seismic Forces.
- Dual Systems with Intermediate Moment Frames Capable of Resisting at Least 25% of Prescribed Seismic Forces.

So if the rate of seismic forces in the walls is at least 0.25, then the user can select the corresponding static system in one of the above two categories.

```
Concrete Walls Shear Force (TABLE 10.2)          Reference Level: 0    0.000(m)
---------------------------------------------*----------------------------------
 n/n    | Walls Shear/Total Shear =     nvx    | Walls Shear/Total Shear =     nvz
Story  |     (Kn)          (Kn)               |     (Kn)          (Kn)
---------------------------------------------*----------------------------------
  1 ***| 7- 155.362     459.459       0.34    |42- 236.143     624.820      0.38
  2    | 4- 155.835     270.241       0.58    |40- 398.700     493.496      0.81
  3    | 5-  32.667     157.022       0.21    |35- 492.585     553.121      0.89
--------------------------------------------------------------------------------
Recommnended Basic Seismic Force-Resisitng System :
All other Types
--------------------------------------------------------------------------------
```

The next check is related to the control of the P-Delta effects according to par.10.9.7.2
The verification is done in each direction.

```
P-Delta Effects-Stability Coefficient (θ) Check  - Par. 10.9.7.2   Direction X
------------------------------------------------------------------------------
 n/n    | Height (m) | Vertical   |Story Drift|Shear Force|Stability|   θmax
 Story | Total|Story|Load Px(kN)|  (Δ)(mm)   |  Vx (kN)  |Coeff.(θ)|   0.10
-------+------+-----+-----------+-----------+-----------+---------+----------
  1-  3|  4.25| 4.25|   6815.10 |       0.49 |    444.09 |  0.00176|
  2-  5|  7.80| 3.55|   3908.43 |       0.37 |    263.92 |  0.00152|
  3-  5| 11.00| 3.20|    641.81 |       0.29 |    156.43 |  0.00037|
--------------------------------------------------------------------------------

P-Delta Effects-Stability Coefficient (θ) Check  - Par. 10.9.7.2   Direction Z
------------------------------------------------------------------------------
 n/n    | Height (m) | Vertical   |Story Drift|Shear Force|Stability|   θmax
 Story | Total|Story|Load Px(kN)|  (Δ)(mm)   |  Vx (kN)  |Coeff.(θ)|   0.10
-------+------+-----+-----------+-----------+-----------+---------+----------
  1- 39|  4.25| 4.25|   6815.10 |       0.32 |    639.82 |  0.00079|
  2- 39|  7.80| 3.55|   3908.43 |       0.47 |    492.20 |  0.00106|
  3- 39| 11.00| 3.20|    641.81 |       0.84 |    552.32 |  0.00030|
--------------------------------------------------------------------------------
```

The next check concerns the Drift limits according to par. 10.12.1

Drift Limits – Par. 10.12.1 – TABLE 10.12 Direction X

n/n Story	Height (m) Total	Height (m) Story	Story Drift (Δ) (mm)	Allowable (Δa) (mm)	Result
1	4.25	4.25	2.46	106.25	0.0231
2	7.80	3.55	1.83	88.75	0.0206
3	11.00	3.20	1.45	80.00	0.0181

Drift Limits – Par. 10.12.1 – TABLE 10.12 Direction Z

n/n Story	Height (m) Total	Height (m) Story	Story Drift (Δ) (mm)	Allowable (Δa) (mm)	Result
1	4.25	4.25	1.59	106.25	0.0149
2	7.80	3.55	2.37	88.75	0.0267
3	11.00	3.20	4.18	80.00	0.0523

The next check concerns the Building Seperation according to par. 10.12.2

Building Seperation par. 10.12.2 (Eq. 10.12)

Dir X : $\delta xt = sqrt[(\delta x1)^2 + (\delta x2)^2] = 0.1904$ cm
Dir Z : $\delta zt = sqrt[(\delta z1)^2 + (\delta z2)^2] = 0.3102$ cm

The last two checks concern the classification of diaphragm's flexibility of each level of the building.

Diaphragms can be designated as Rigid or Flexible. The classification of the structure in total is determined from the majority of the levels' classification. In the scenarios' parameters, there is an option for choosing to characterize the building either manually or automatically so the program checks the following criteria according to par. 10.3.1.2 & 10.3.1.3

Diaphragm Flexibility
Rigid Diaphragm Condition (par. 10.3.1.2)

n/n Story	Height (m) Total	Height (m) Story	Plan Dimensions (m) (Lx)	Plan Dimensions (m) (Lz)	Ratio Lmax/Lmin	Result
1	4.25	4.25	19.60	13.70	1.43	Yes
2	7.80	3.55	13.20	9.30	1.42	Yes
3	11.00	3.20	9.80	9.00	1.09	Yes

Rigid Diaphragm : Diaphragms of concrete slabs or concrete filled metal deck with Lmax/Lmin <3 and with no plan irregularities

Diaphragm Flexibility
Calculated Flexible Diaphragm Condition (par. 10.3.1.3)

n/n Story	Height (m) Total	Height (m) Story	Maximum Diaphragm Deflection (mm) (X)	Maximum Diaphragm Deflection (mm) (Z)	Average Drift (mm) (X)	Average Drift (mm) (Z)	Result
1	4.25	4.25	0.04	0.06	0.04	0.05	No
2	7.80	3.55	0.11	0.14	0.07	0.08	No
3	11.00	3.20	0.18	0.21	0.07	0.07	Yes

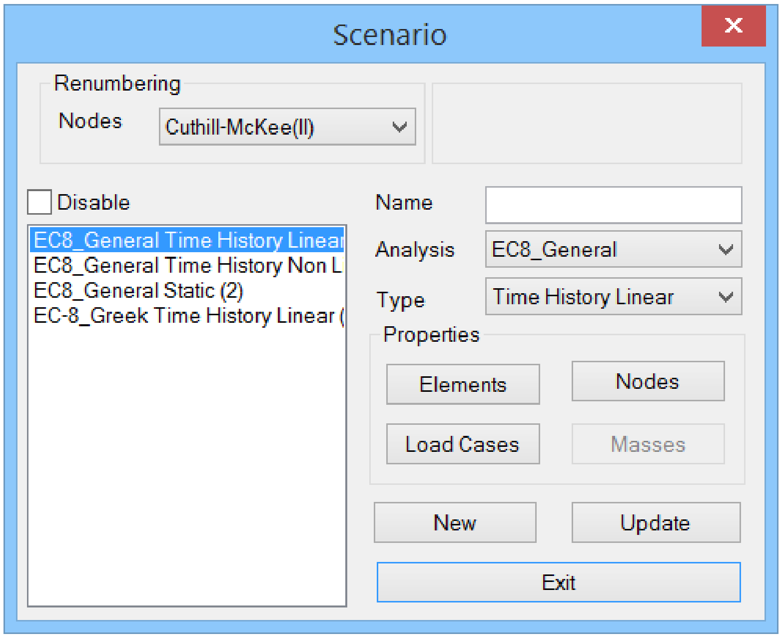
Flexible Diaphragm : Maximum Diaphragm Deflection > 2*(Average Drift)

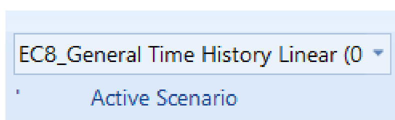
Finally, by choosing "**Seismic Force**", all the data, related to the seismic analysis and loads distribution, is reported

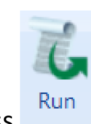
```
SCENARIO : 5 - Modal Analysis Procedure (Section 10.10)

DATA FILE LOAD CASES
---------------------------
Load Case 1 (Dead-G)
Load Case 2 (Live-Q)
MASSES CALCULAT. FROM : G+Ψ2*Q

RESULTS FILE - INTERNAL FORCES
--------------------------------------
Load Case 1 (Dead-ΣG)
Load Case 2 (Live-ΣQ)
Load C. 3 (Horizontal Seismic Force x)
Load C. 4 (Horizontal Seismic Force z)
Load C. 5 (Eccentricity of seism. force x from maxez)
Load C. 6 (Eccentricity of seism. force x from minez)
Load C. 7 (Eccentricity of seism. force z from maxex)
Load C. 8 (Eccentricity of seism. force z from minex)
L. Case 9 (Vertical Seismic Force    y)


GENERAL DATA - CALCULATION PARAMETERS
---------------------------------------
Occupancy Category (TABLE 1.6-1)                : I
Ground Motion Parameters  (%g)                  : S1= 0.200 Ss= 0.800
Site Class   (par. 9.4.2)                       : B
Site Coefficients (TABLE 9.4.3a & 9.4.3b)       : Fa= 1.000 Fv= 1.000
Spectral Accelerations (par. 9.4.3) (%g)        : Sms= 0.800 Sm1= 0.200
Design Spectral Accelerations (par. 9.4.4) (%g) : Sds= 0.533 Sd1= 0.133
Occupancy Importance Factor (TABLE 9.5)         : 1.000
Seismic Design Category (TABLE 9.6a &9.6b)      : A (A - A)
Reliability factor (par. 10.3.3)                : ρ= 1.000
Structural System (TABLE 10.2)                  : R= 4.000  Ωo= 2.500  Cd= 5.000
Bearing Wall Systems
Special reinforced concrete shear walls


Fundamental Periods of Vibration
-------------------------------------------------------------------
Direction Ix : T  (sec)=   0.49228
Direction IIz: T  (sec)=   0.49228
Direction y  : Tv (sec)=   0.00000
-------------------------------------------------------------------


  n/n   Level   Plan Dimensions        Coef.ψ2 Acc. Eccenticities
Height   (m)      LIx (m)   LIIz (m)    L.C.2   etix(m)    etiz(m)
-------------------------------------------------------------------
   0      0.000   19.400    13.600     0.300     0.970      0.680
   1      4.250   19.600    13.700     0.300     0.980      0.685
   2      7.800   13.200     9.300     0.300     0.660      0.465
   3     11.000    9.800     9.000     0.300     0.490      0.450
-------------------------------------------------------------------
eτix = 0.05 *LIx , eτiz = 0.05 *LIIz
-------------------------------------------------------------------
```

§ 5. EC-8_Greek Analysis and Time History Linear Type

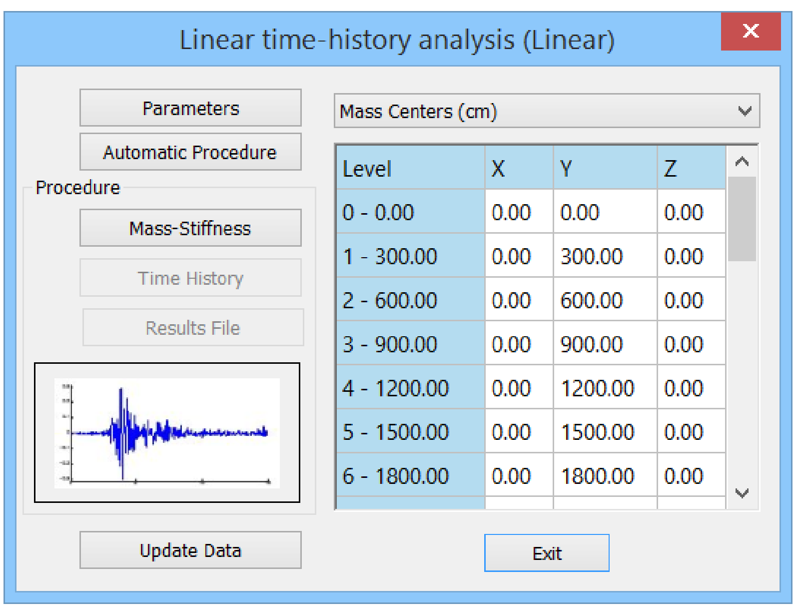
The application of Time History analysis in SCADA Pro starts by defining the respective Analysis Scenario:



Select the Active Scenario 

and press 

:



The total procedure contains three steps:

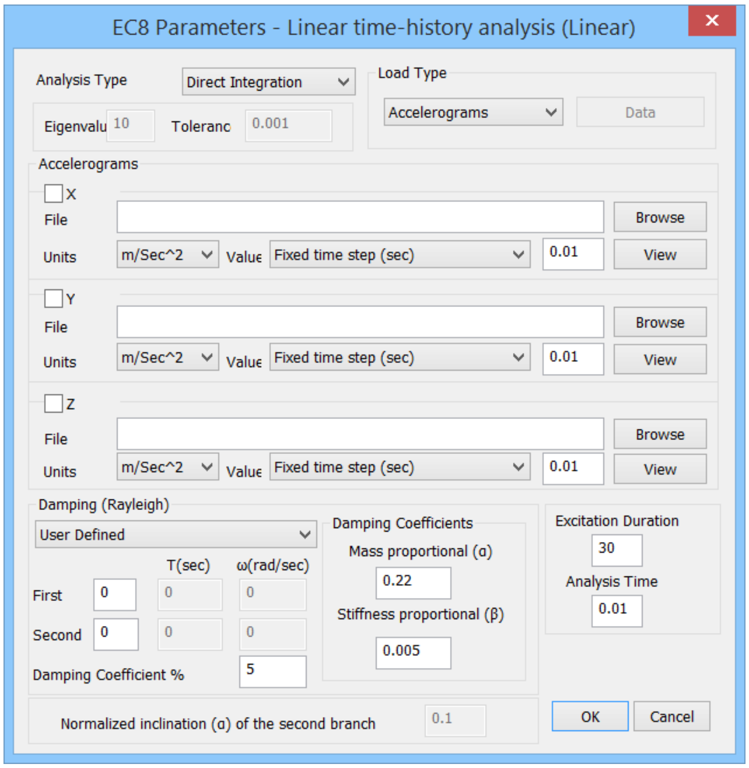
- Definition of the analysis parameters

- Calculation of the masses and stiffness for all members
- Run dynamic analysis for the selected accelerograms

Steps 2 and three are operated either sequentially by pressing the buttons “Mass – Stiffness” and “Time History”, or automatically by selecting the button "Automatic Procedure”.

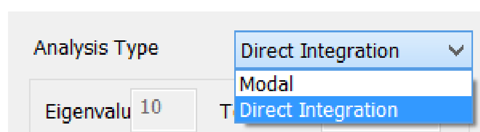
The first step of the process before the analysis is to define its parameters.

Press Parameters :

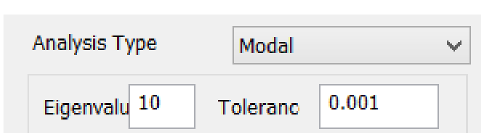


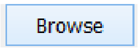
First select Analysis Type between:

- Direct Integration and
- Modal

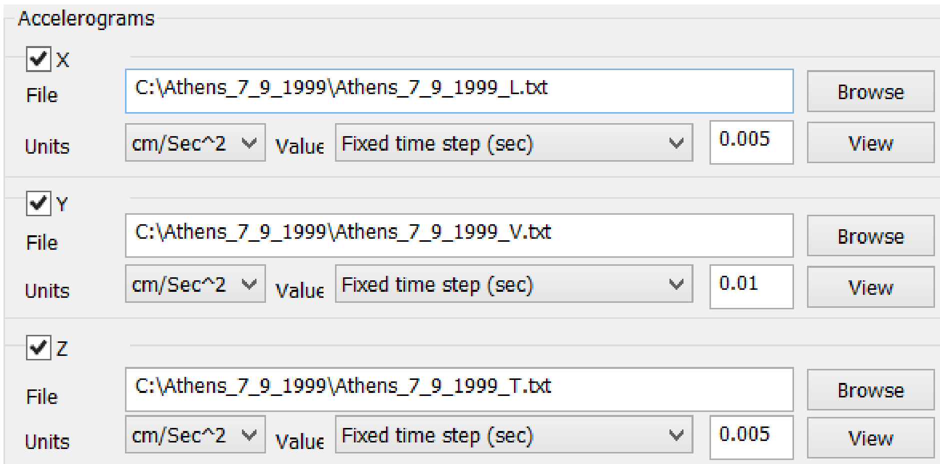


By selecting Modal you must determine the number of Eigenvalues to be taken into account in the modal analysis as well as the calculations’ tolerance.



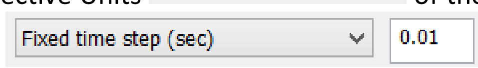
Under "Accelerograms" the user can choose the directions of the seismic excitation, with the possibility of choosing from one to three directions by activating the corresponding checkbox in "X", "Y" or "Z".
Then the user must import the corresponding record file of the seismic stimulation through 

.

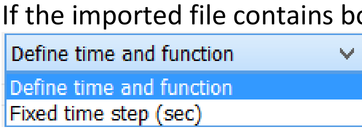


This file must be in a .txt format containing a column with the values of ground acceleration for each time step.

The user must also select the respective Units 

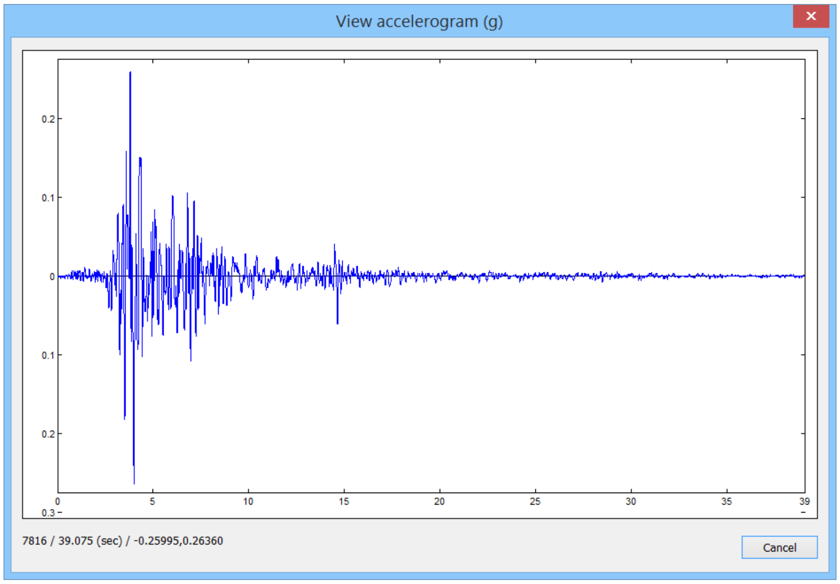
of the ground acceleration and the time step of the recording 

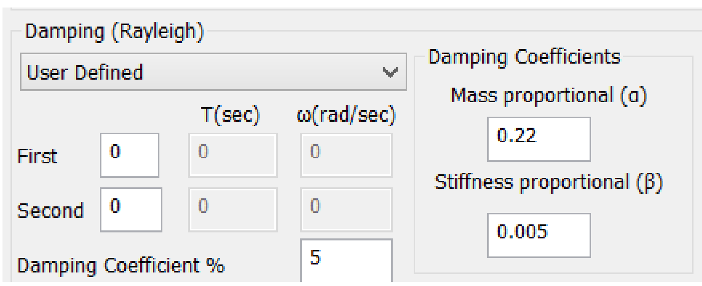
.

If the imported file contains both seismic stimulation and time step, select


and the program will read the time step values.

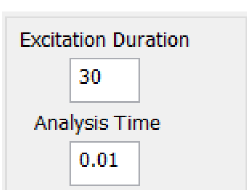
Finally, it is also possible to display each accelerogram by using the "View" button.





In the "Damping Rayleigh" section the user must select the parameter values for the Rayleigh registry.

More specifically the Damping Coefficient % must be defined, along with two modes that will be imposed on this coefficient.
Using these parameters the program calculates the values of the Damping Coefficients (a) and Stiffness proportional (b).

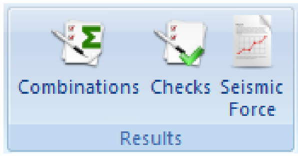


Finally, the user must define the Excitation Duration and the Analysis Time to be considered.

It’s not necessary that the Analysis Time coincides with those of the accelerograms.
In case that the time step of the analysis is smaller than the Accelerograms’ step, the appropriate value is derived from a linear interpolation between the nearest two points.
In case that the time step of the analysis is bigger than the Accelerograms steps, then the structure will perform free oscillation for the remaining time.

After completing the input of the parameters, the user returns to the Run analysis window to proceed to the next steps

2. Results



"Results" command group contains commands about the generation of the load combinations, the design checks, the analysis results and the seismic forces.

2.1 Combinations

SCADA Pro contains all the combination files for all Static and Dynamic scenarios of Elastic and Inelastic Analysis, as "**Defalut Combinations**".

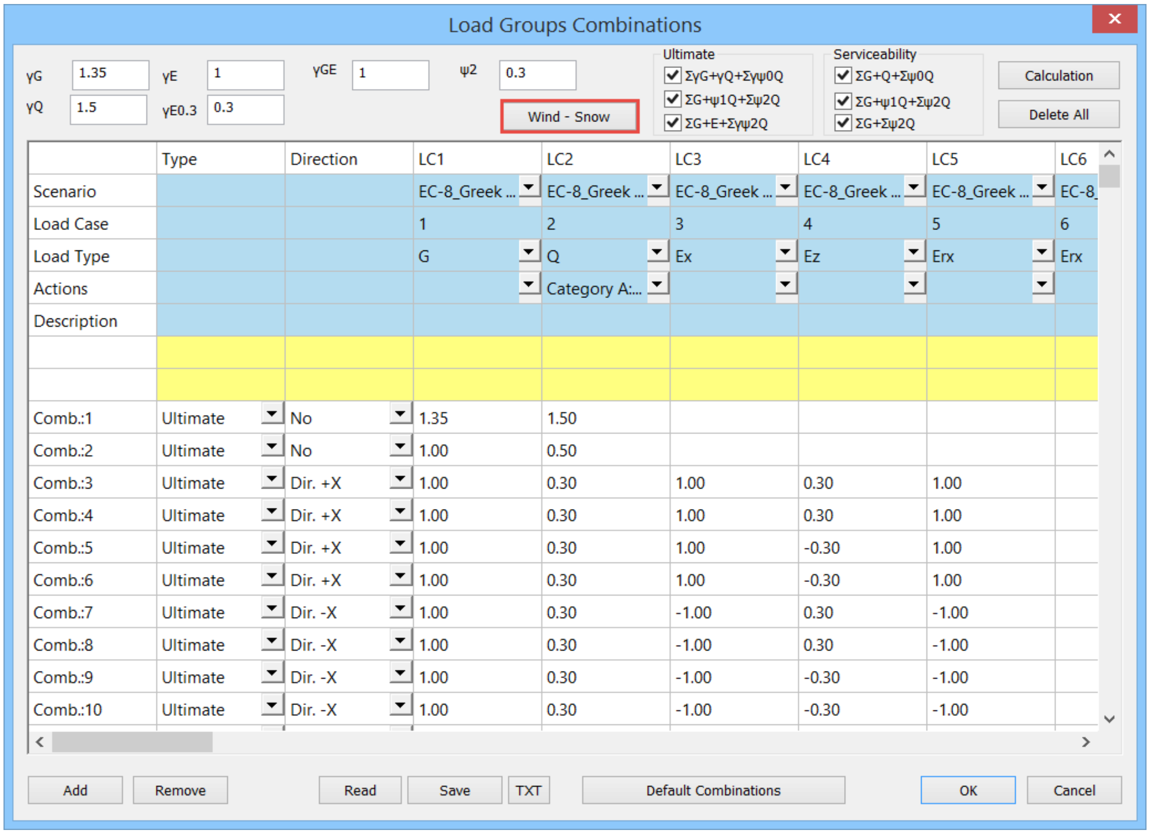
Name	Date modified	Type	Size
eak-dyn.cmb	23/3/2010 1:27 μμ	CMB File	55 KB
eak-dyn-et.cmb	11/1/2010 5:12 μμ	CMB File	48 KB
eak-static.cmb	11/1/2010 5:11 μμ	CMB File	53 KB
Ec8-dyn.cmb	23/3/2010 1:22 μμ	CMB File	48 KB
Ec8-dyn-cypr.cmb	23/3/2010 1:22 μμ	CMB File	48 KB
Ec8-PushOver.cmb	13/5/2013 11:44 πμ	CMB File	7 KB
Ec8-static.cmb	23/3/2010 1:21 μμ	CMB File	53 KB
Ec8-static-cypr.cmb	23/3/2010 1:21 μμ	CMB File	53 KB
ita-dyn.cmb	23/3/2010 1:09 μμ	CMB File	48 KB
itaEc8-dyn.cmb	23/3/2010 1:18 μμ	CMB File	48 KB
itaEc8-static.cmb	23/3/2010 3:12 μμ	CMB File	53 KB
ita-static.cmb	23/3/2010 1:06 μμ	CMB File	53 KB
pal-static.cmb	27/2/2018 11:35 πμ	CMB File	3 KB
sbc-000.cmb	5/5/2017 4:35 μμ	CMB File	91 KB
sbc-001.cmb	5/5/2017 4:35 μμ	CMB File	91 KB
sbc-002.cmb	5/5/2017 4:15 μμ	CMB File	91 KB
sbc-003.cmb	5/5/2017 4:25 μμ	CMB File	91 KB

Default combinations concern seismic scenarios. To create combinations of non-seismic scenarios, you can use both automatic and manual mode.

Since you perform a seismic analysis scenario, its combinations are generated by the program automatically. By selecting the command "Combinations" the window with the combinations of the active seismic scenario opens.
You can achieve the same thing by selecting the command "Default combinations", that is, the program will import the combinations concerning the active seismic analysis scenario.
Predetermined combinations of the "executed" seismic analysis scenarios are automatically recorded by the program.

§ For Linear Analysis

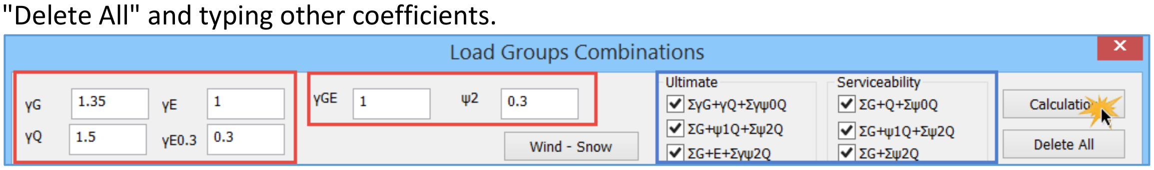
Combinations: creating load combinations for Post-Processor and Members Design as well.



	Type	Direction	LC1	LC2	LC3	LC4	LC5	LC6
Scenario			EC-8_Greek ...	EC-8_Greek ...	EC-8_Greek ...	EC-8_Greek ...	EC-8_Greek ...	EC-8_
Load Case			1	2	3	4	5	6
Load Type			G	Q	Ex	Ez	Erx	Erx
Actions				Category A:...				
Description								
Comb.:1	Ultimate	No	1.35	1.50				
Comb.:2	Ultimate	No	1.00	0.50				
Comb.:3	Ultimate	Dir. +X	1.00	0.30	1.00	0.30	1.00	
Comb.:4	Ultimate	Dir. +X	1.00	0.30	1.00	0.30	1.00	
Comb.:5	Ultimate	Dir. +X	1.00	0.30	1.00	-0.30	1.00	
Comb.:6	Ultimate	Dir. +X	1.00	0.30	1.00	-0.30	1.00	
Comb.:7	Ultimate	Dir. -X	1.00	0.30	-1.00	0.30	-1.00	
Comb.:8	Ultimate	Dir. -X	1.00	0.30	-1.00	0.30	-1.00	
Comb.:9	Ultimate	Dir. -X	1.00	0.30	-1.00	-0.30	-1.00	
Comb.:10	Ultimate	Dir. -X	1.00	0.30	-1.00	-0.30	-1.00	

After running a scenario analysis, combinations are automatically generated by the program. "Combinations" opens the table with the combinations of the active scenarios.

The same results are derived from the "**Default Combination**" button, which fills in the table with the combinations of the active scenario analysis.
The default combinations of the executed analysis, are automatically saved by the program.
You can create your combinations without using the "Default", or add more loads of other scenarios and calculate the new combinations either by modifying the defaults or deleting all "Delete All" and typing other coefficients.



Furthermore, you can type the factors, select the combinations and then press 'Calculation" to fill in the table.

The tool "Load Groups Combinations" works like an Excel file offering possibilities like copy, delete using Ctrl+C, Ctrl+V, Shift and right click.

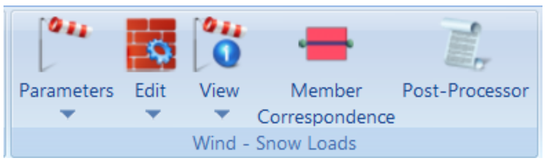
§ Combinations for Wind-Snow

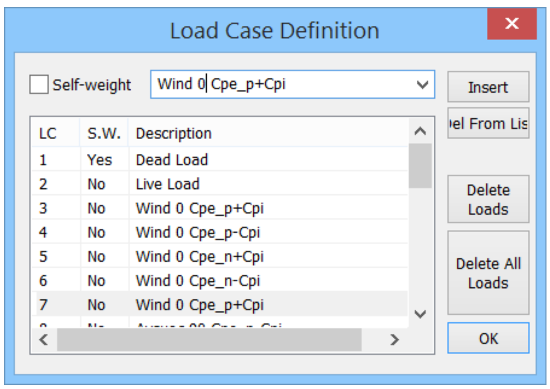
The tool "Laod Groups Combinations" works like an Excel file offering possibilities like copy, delete using Ctrl+C, Ctrl+V, Shift and right click.

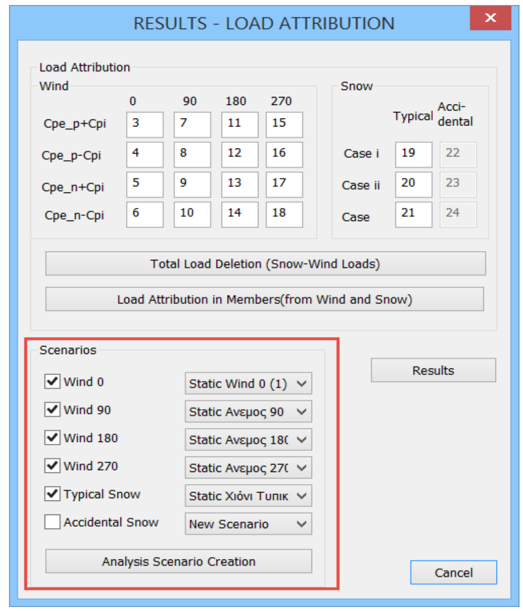
Default combinations concern seismic scenarios. To create combinations of scenarios without seismic loads you can use both automatic and manual mode.

The **automatic mode:**

requires that the automatic procedure for the calculation and distribution of loads of wind and snow as well as the automatic creation of the loads and combinations (see Chapter 6) is already done.





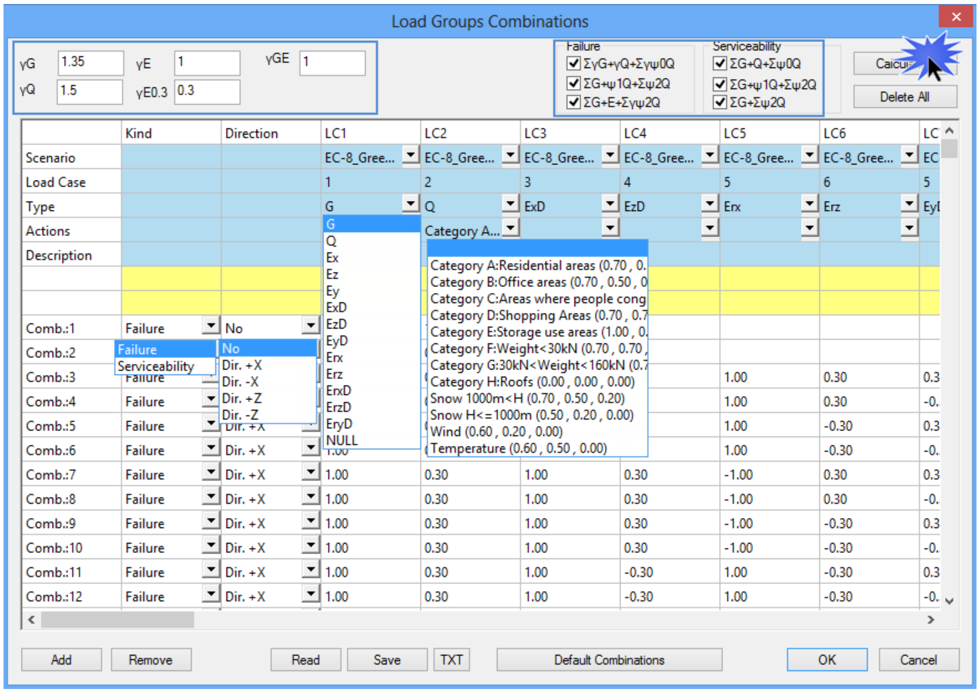


Concerning the above conditions, it is possible to automatically create wind and snow combinations by using the command Wind - Snow.

After running the seismic scenario and all the static scenarios of wind and snow, activate the seismic scenario and choose the command "Combinations". The combinations of the active seismic scenarios are completed automatically. To create automatically the combinations of all wind and snow loads, press Wind - Snow. Automatically the coefficients of all wind and snow scenarios will be filled, offering a complete loads combinations file. Press Save to save the file.

The **manual way** :

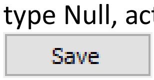
Except for the “Default Combinations,” you can add others with loads from other scenarios.



First type in the partial safety factors γG, γQ, γE, and check the Failure and Serviceability equations you want to be considered.

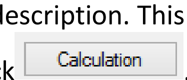
Then fill in the “LC” columns:

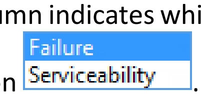
- Select the previously created Scenario from the scenarios list.
- Select the Load Type between Dead, Live, Seismic, or Null. Null is selected for another type of loads like Wind, Temperature, etc.
- To add to the “Default Combinations,” those of snow loads choose LC10, load case 1, type Null, action Snow and “Calculate”.

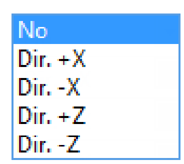
  

  of the combinations file.

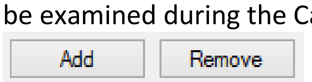
- In the Actions, line select the corresponding action from the list.
- In the Description line, you can type in a description. This is optional.

Do the same thing for all Load Cases and then click  . Combinations’ lines are filled in with the appropriate coefficients automatically.

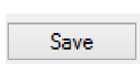
“Type” column indicates which type of limit state is examined through the defined combination  .



“Direction” column indicates in which direction the capacity of the structure will be examined during the Capacity Design procedure for the specific combination.



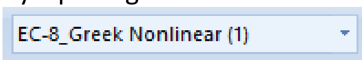
buttons allow to add or remove lines and columns since they have been selected, as in an excel file.

You always have to  the combination as a CMB file in the project’s folder. You will need this file during “Post-Processor” and “Members Design”.

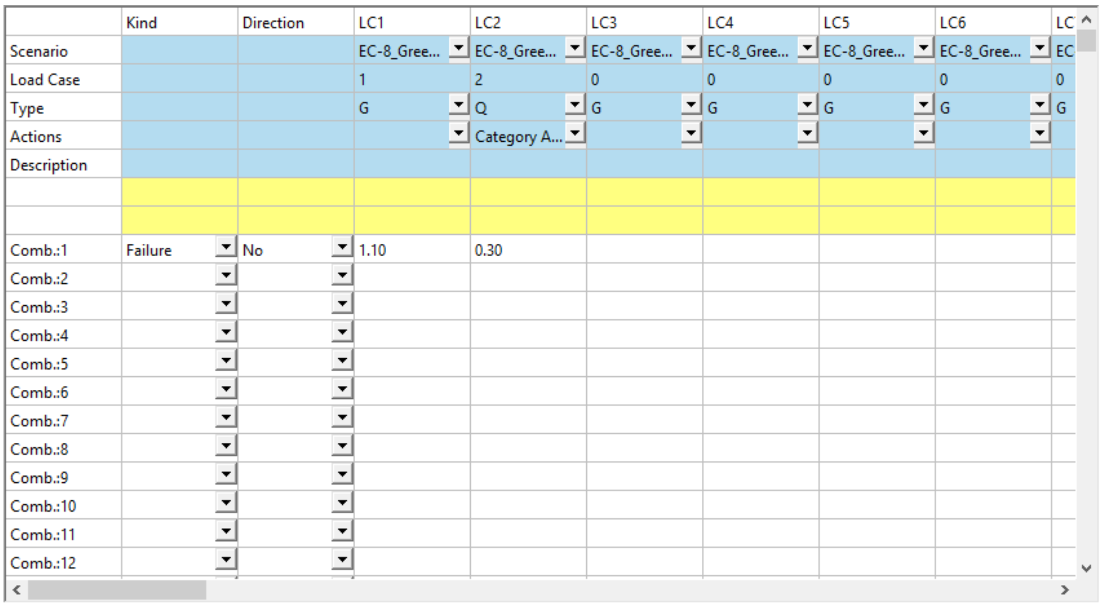
Press Read to open a previously saved CMB file, or TXT to open a TXT file that contains the load combinations.

§ For Nonlinear (Pushover) Analysis

Nonlinear analysis combination is a unique combination of Dead and Live loads. It is displayed by Opening the “Combinations” when a Nonlinear Active Scenario

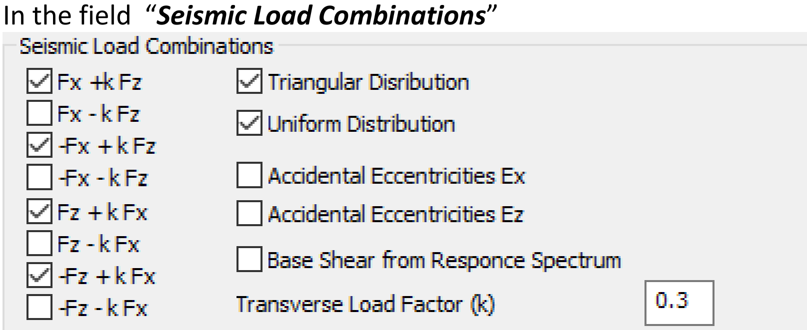


is activated.



Afterwards, through the Parameters of the Inelastic analysis scenario

In the field “***Seismic Load Combinations***”

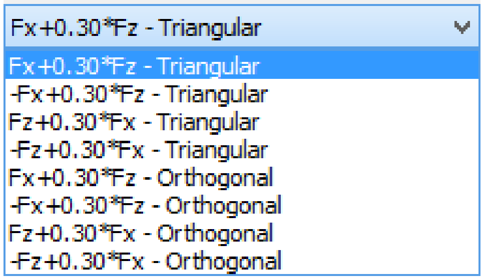


We define the combinations for which inelastic analyzes will be performed. Each combination means that a seismic force will be applied along the specified direction (x or z) by a factor 1 as well as a vertical seismic force in the vertical direction by a factor defined in the field “Transverse Load Factor”.

⚠ The default value is 0.3.

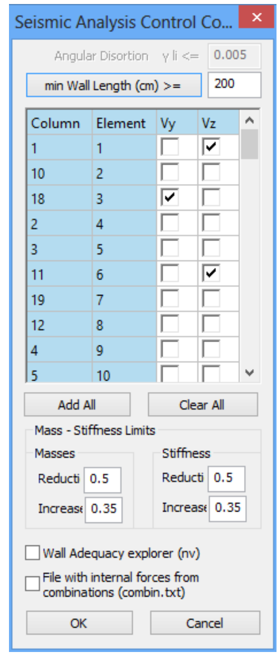
Moreover, we define the type of height distribution of seismic force (Triangular or Orthogonal). KANEPE demands both types of seismic force distribution.
Also, in case we want, except for the seismic forces, the moments resulting from the accidental eccentricities to be taken into consideration, then we activate the fields “Accidental eccentricities Ex and Ez”.

Then, to design the reinforcement, you should define another combination as well as the distribution, selecting the command “**Checks**” in the field “***Analysis method for the Reinforcing Model Design***” (you can see **§2.2 “Checks”**)



2.2 Checks

§ For Linear Analysis

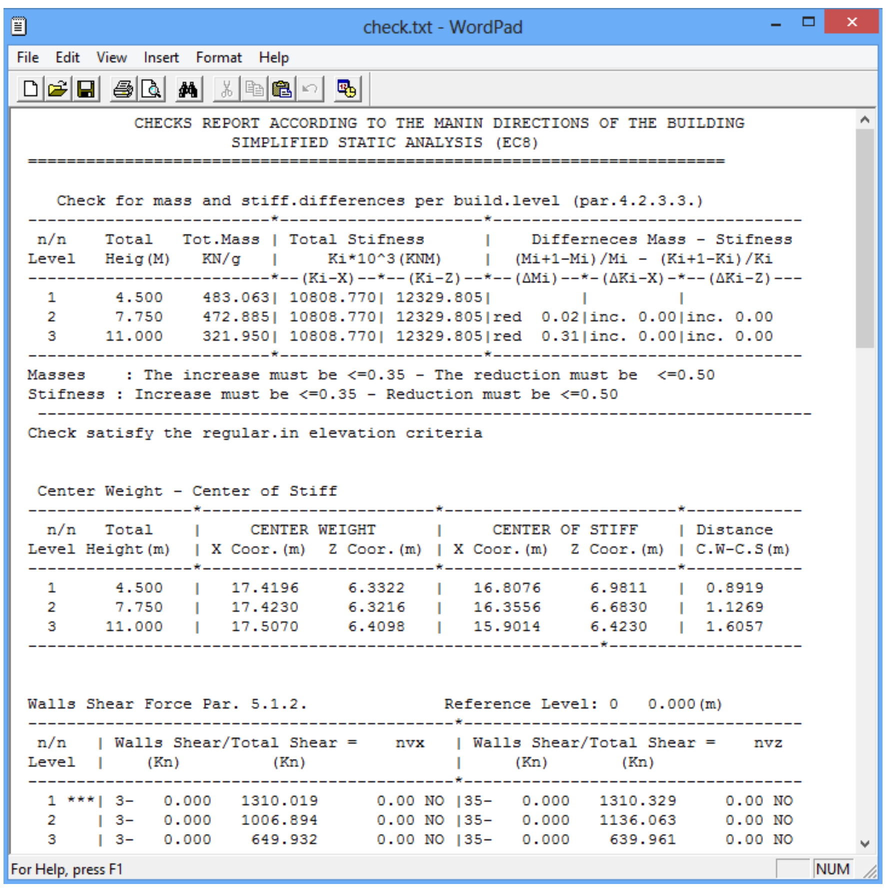


Press “Checks” and in the dialog box:

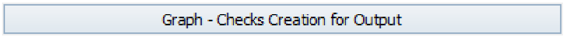
- Type in the minimum length for defining the walls and click the corresponding button,
- set limits on the mass and the stiffness considering the regularity conditions of the building,
- press “OK”.

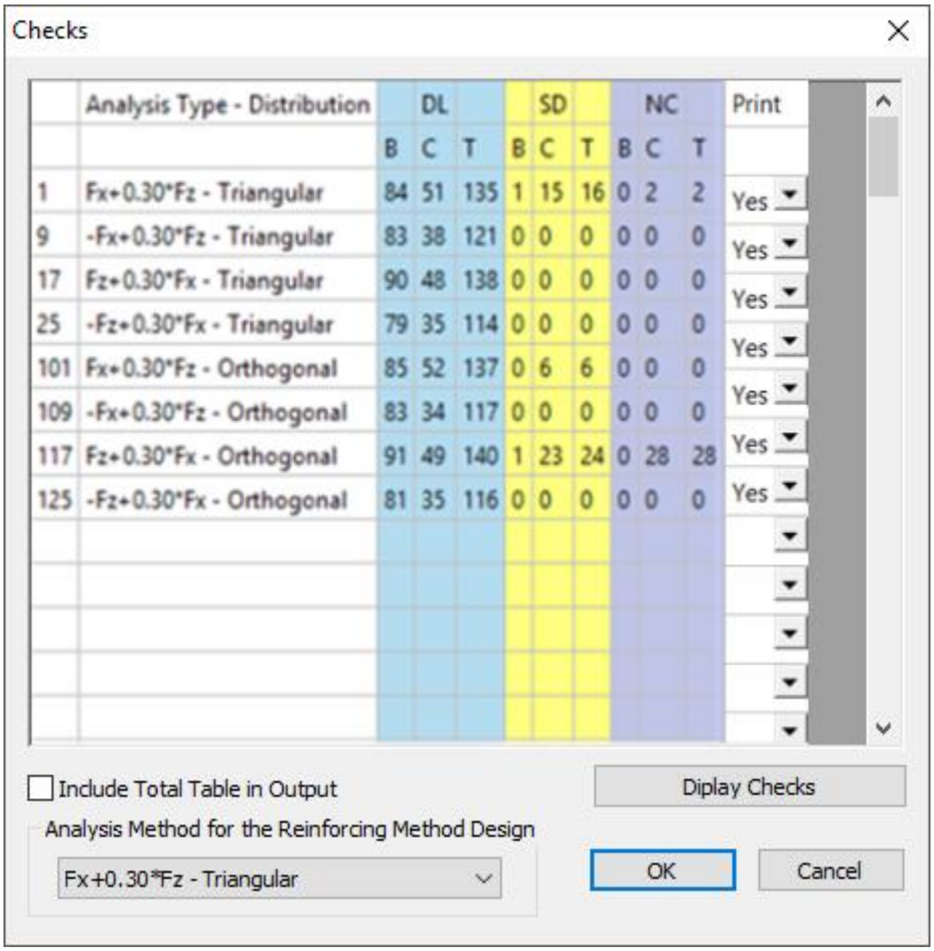
Automatically a TXT file opens, that contains design check’s results according to the “active scenarios”:

- Regularity
- Second Order effects
- Interstory Drift Limitation
- Interstory Drift sensitivity coefficient θ
- Walls Shear Force ratio nv,z
- Seismic joint’s calculation


```
check.txt - WordPad
File  Edit  View  Insert  Format  Help

            CHECKS REPORT ACCORDING TO THE MANIN DIRECTIONS OF THE BUILDING
                      SIMPLIFIED STATIC ANALYSIS (EC8)
========================================================================


   Check for mass and stiff.differences per build.level (par.4.2.3.3.)
-------------------------*----------------------*--------------------------------
 n/n    Total   Tot.Mass | Total Stifness       |    Differneces Mass - Stifness
Level   Heig(M)   KN/g   |      Ki*10^3(KNM)    |  (Mi+1-Mi)/Mi  - (Ki+1-Ki)/Ki
-------------------------*--(Ki-X)--*--(Ki-Z)--*--(ΔMi)--*-(ΔKi-X)-*--(ΔKi-Z)---
  1      4.500    483.063| 10808.770| 12329.805|         |         |
  2      7.750    472.885| 10808.770| 12329.805|red  0.02|inc. 0.00|inc. 0.00
  3     11.000    321.950| 10808.770| 12329.805|red  0.31|inc. 0.00|inc. 0.00
-------------------------*----------------------*--------------------------------
Masses    : The increase must be <=0.35 - The reduction must be  <=0.50
Stifness : Increase must be <=0.35 - Reduction must be <=0.50
 --------------------------------------------------------------------------------
Check satisfy the regular.in elevation criteria


 Center Weight - Center of Stiff
-----------------*-------------------------*-------------------------*-----------
  n/n   Total     |     CENTER WEIGHT       |     CENTER OF STIFF     | Distance
Level Height(m)   | X Coor.(m)  Z Coor.(m) | X Coor.(m)  Z Coor.(m) | C.W-C.S(m)
-----------------*-------------------------*-------------------------*-----------
  1      4.500    |   17.4196     6.3322   |   16.8076     6.9811   |  0.8919
  2      7.750    |   17.4230     6.3216   |   16.3556     6.6830   |  1.1269
  3     11.000    |   17.5070     6.4098   |   15.9014     6.4230   |  1.6057
------------------------------------------------------------*-------------------


Walls Shear Force Par. 5.1.2.             Reference Level: 0    0.000(m)
---------------------------------------*----------------------------------------
 n/n    | Walls Shear/Total Shear =    nvx   | Walls Shear/Total Shear =    nvz
Level   |     (Kn)          (Kn)             |     (Kn)          (Kn)
---------------------------------------*----------------------------------------
  1 ***| 3-    0.000    1310.019      0.00 NO |35-    0.000    1310.329     0.00 NO
  2    | 3-    0.000    1006.894      0.00 NO |35-    0.000    1136.063     0.00 NO
  3    | 3-    0.000     649.932      0.00 NO |35-    0.000     639.961     0.00 NO
For Help, press F1                                                          NUM
```

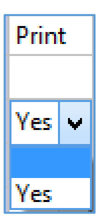
§ For Nonlinear (Pushover) Analyses

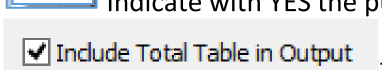
The precondition in order the checks of Inelastic Analyzes to open is that since the analysis is over, select the command Mass Distribution so as the window **Report** opens and then press the button 



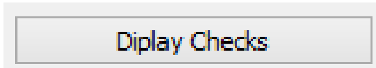
	Analysis Type - Distribution	DL			SD			NC			Print
		B	C	T	B	C	T	B	C	T	
1	Fx+0.30*Fz - Triangular	84	51	135	1	15	16	0	2	2	Yes
9	-Fx+0.30*Fz - Triangular	83	38	121	0	0	0	0	0	0	Yes
17	Fz+0.30*Fx - Triangular	90	48	138	0	0	0	0	0	0	Yes
25	-Fz+0.30*Fx - Triangular	79	35	114	0	0	0	0	0	0	Yes
101	Fx+0.30*Fz - Orthogonal	85	52	137	0	6	6	0	0	0	Yes
109	-Fx+0.30*Fz - Orthogonal	83	34	117	0	0	0	0	0	0	Yes
117	Fz+0.30*Fx - Orthogonal	91	49	140	1	23	24	0	28	28	Yes
125	-Fz+0.30*Fx - Orthogonal	81	35	116	0	0	0	0	0	0	Yes

The table above contains the total number of beams (B) and columns (C) which have lower capacity than the demanded. For each pushover and each LS. The symbol “T” means total.

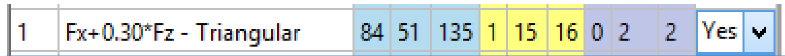


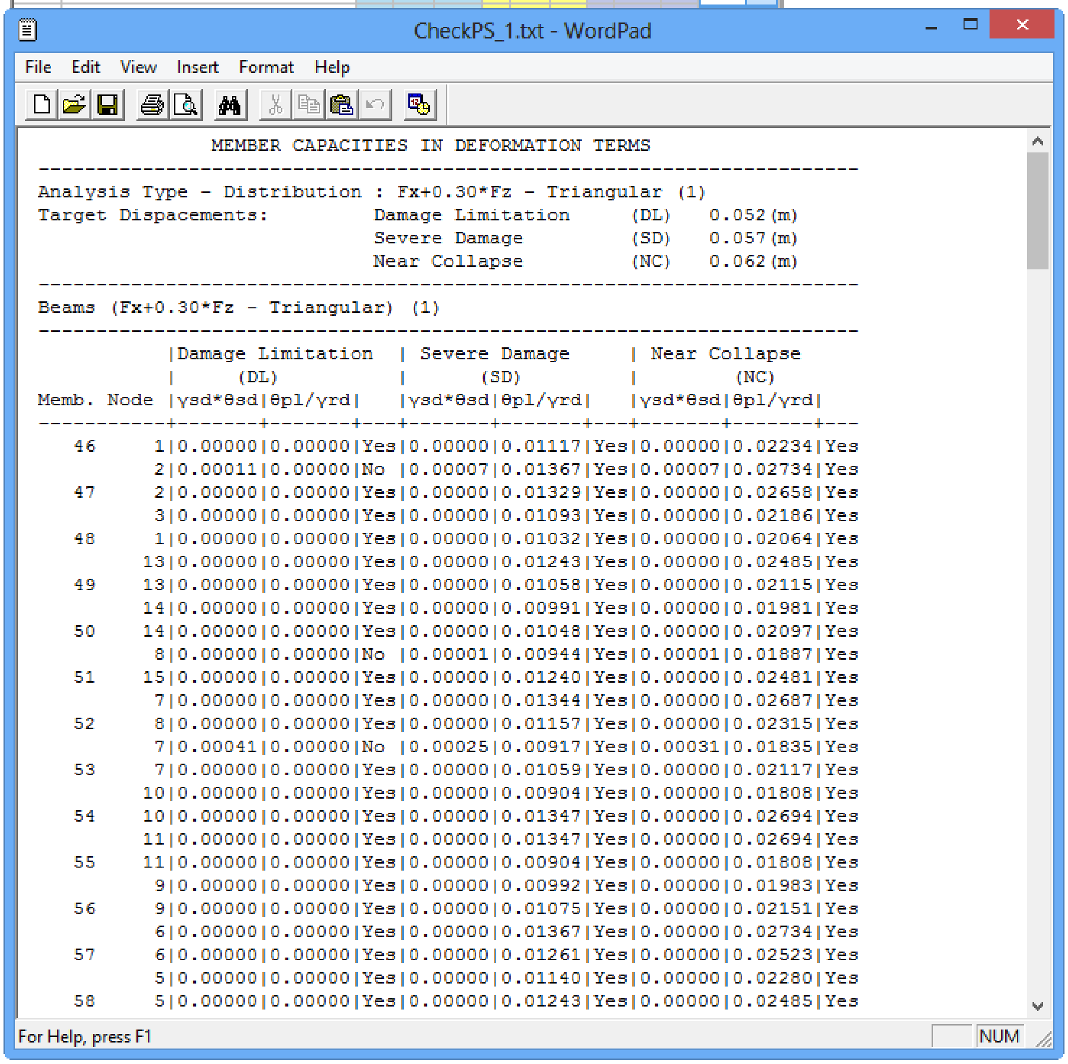
Indicate with YES the pushover analysis to be included in the prints, otherwise check 

to consider all of them.

The 

button opens the TXT file that contains all members' capacity regarding deformation for the pushover analysis with a "YES" indicator in the "Print" column:






```
                    MEMBER CAPACITIES IN DEFORMATION TERMS
 -----------------------------------------------------------------------
 Analysis Type - Distribution : Fx+0.30*Fz - Triangular (1)
 Target Dispacements:          Damage Limitation      (DL)    0.052 (m)
                               Severe Damage          (SD)    0.057 (m)
                               Near Collapse          (NC)    0.062 (m)
 -----------------------------------------------------------------------
 Beams (Fx+0.30*Fz - Triangular) (1)
 -----------------------------------------------------------------------
             |Damage Limitation  | Severe Damage      | Near Collapse
             |      (DL)         |       (SD)         |        (NC)
 Memb. Node  |γsd*θsd|θpl/γrd|   |γsd*θsd|θpl/γrd|   |γsd*θsd|θpl/γrd|
 ------------+-------+-------+---+-------+-------+---+-------+-------+---
    46      1|0.00000|0.00000|Yes|0.00000|0.01117|Yes|0.00000|0.02234|Yes
            2|0.00011|0.00000|No |0.00007|0.01367|Yes|0.00007|0.02734|Yes
    47      2|0.00000|0.00000|Yes|0.00000|0.01329|Yes|0.00000|0.02658|Yes
            3|0.00000|0.00000|Yes|0.00000|0.01093|Yes|0.00000|0.02186|Yes
    48      1|0.00000|0.00000|Yes|0.00000|0.01032|Yes|0.00000|0.02064|Yes
           13|0.00000|0.00000|Yes|0.00000|0.01243|Yes|0.00000|0.02485|Yes
    49     13|0.00000|0.00000|Yes|0.00000|0.01058|Yes|0.00000|0.02115|Yes
           14|0.00000|0.00000|Yes|0.00000|0.00991|Yes|0.00000|0.01981|Yes
    50     14|0.00000|0.00000|Yes|0.00000|0.01048|Yes|0.00000|0.02097|Yes
            8|0.00000|0.00000|No |0.00001|0.00944|Yes|0.00001|0.01887|Yes
    51     15|0.00000|0.00000|Yes|0.00000|0.01240|Yes|0.00000|0.02481|Yes
            7|0.00000|0.00000|Yes|0.00000|0.01344|Yes|0.00000|0.02687|Yes
    52      8|0.00000|0.00000|Yes|0.00000|0.01157|Yes|0.00000|0.02315|Yes
            7|0.00041|0.00000|No |0.00025|0.00917|Yes|0.00031|0.01835|Yes
    53      7|0.00000|0.00000|Yes|0.00000|0.01059|Yes|0.00000|0.02117|Yes
           10|0.00000|0.00000|Yes|0.00000|0.00904|Yes|0.00000|0.01808|Yes
    54     10|0.00000|0.00000|Yes|0.00000|0.01347|Yes|0.00000|0.02694|Yes
           11|0.00000|0.00000|Yes|0.00000|0.01347|Yes|0.00000|0.02694|Yes
    55     11|0.00000|0.00000|Yes|0.00000|0.00904|Yes|0.00000|0.01808|Yes
            9|0.00000|0.00000|Yes|0.00000|0.00992|Yes|0.00000|0.01983|Yes
    56      9|0.00000|0.00000|Yes|0.00000|0.01075|Yes|0.00000|0.02151|Yes
            6|0.00000|0.00000|Yes|0.00000|0.01367|Yes|0.00000|0.02734|Yes
    57      6|0.00000|0.00000|Yes|0.00000|0.01261|Yes|0.00000|0.02523|Yes
            5|0.00000|0.00000|Yes|0.00000|0.01140|Yes|0.00000|0.02280|Yes
    58      5|0.00000|0.00000|Yes|0.00000|0.01243|Yes|0.00000|0.02485|Yes
```

At the bottom of the file, the Shear Resistance Check appears only for members that fail in shear.

SHEAR RESISTANCE CHECK

Beams (Fx+0.30*Fz - Triangular) (1)

Elem.	Node	Vrd,s	Vrd,max	Vr	Ved	Ratio	A - DL	B - SD	Γ - NC

Columns (Fx+0.30*Fz - Triangular) (1)

Elem.	Node	Vrd,s	Vrd,max	Vr	Ved	Ratio	A - DL	B - SD	Γ - NC
4	42	y:1369.81	2227.03	171.52	171.71	1.0011			
	5	y:1369.81	2227.03	171.51	171.71	1.0011			
18	56	y: 276.60	128.48	251.99	2.68	0.0209	NO	NO	NO
	19	y: 276.60	128.48	254.75	2.68	0.0209	NO	NO	NO
38	1	y: 276.60	128.48	284.94	54.80	0.4265	NO	NO	NO
	56	y: 276.60	128.48	286.33	54.80	0.4265	NO	NO	NO

At the end of this file and if the parameters of the Masonry Infills scenarios are active, the checkbox/displays the results of masonry infills capacities in deformation terms for each infill element.

For tensioned members, results are not displayed because these effects are not taken into account in the project's model.

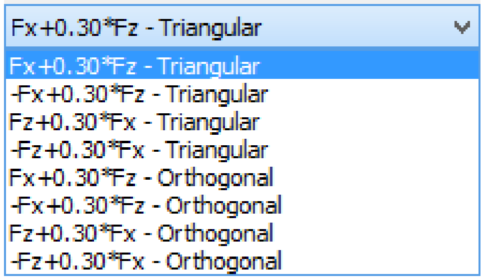
MASONRY INFILLS CAPACITIES IN DEFORMATION TERMS

Member	Damage Limitation (DL) γsd*εf	εy		Severe Damage (SD) γsd*εf	εu/γrd		Near Collapse (NC) γsd*εf	εu	
121 Comp.	0.00440	0.00150	No	0.00493	0.00269	No	0.00534	0.00350	No
122 Tens.									
123 Comp.	0.00262	0.00150	No	0.00278	0.00269	No	0.00291	0.00350	Yes
124 Tens.									
125 Comp.	0.00426	0.00150	No	0.00474	0.00269	No	0.00511	0.00350	No
126 Tens.									
127 Comp.	0.00291	0.00150	No	0.00306	0.00269	No	0.00317	0.00350	Yes
128 Tens.									

Check ☑ Include Total Table in Output to include the table in the print output.

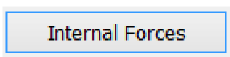
- Checks document help you to assess in which Pushover analysis, the structural elements present lower capacity than the one defined in the considered LS, i.e. it can be easily observed in which Pushover analysis, the defined performance level is not satisfied. In such case, the structure must be strengthened, for example through the reinforcement of some structural elements, and be redesigned.
- First select from the list the analysis, which indicates the redesign of the existing structure.

Analysis Method for the Reinforcing Method Design



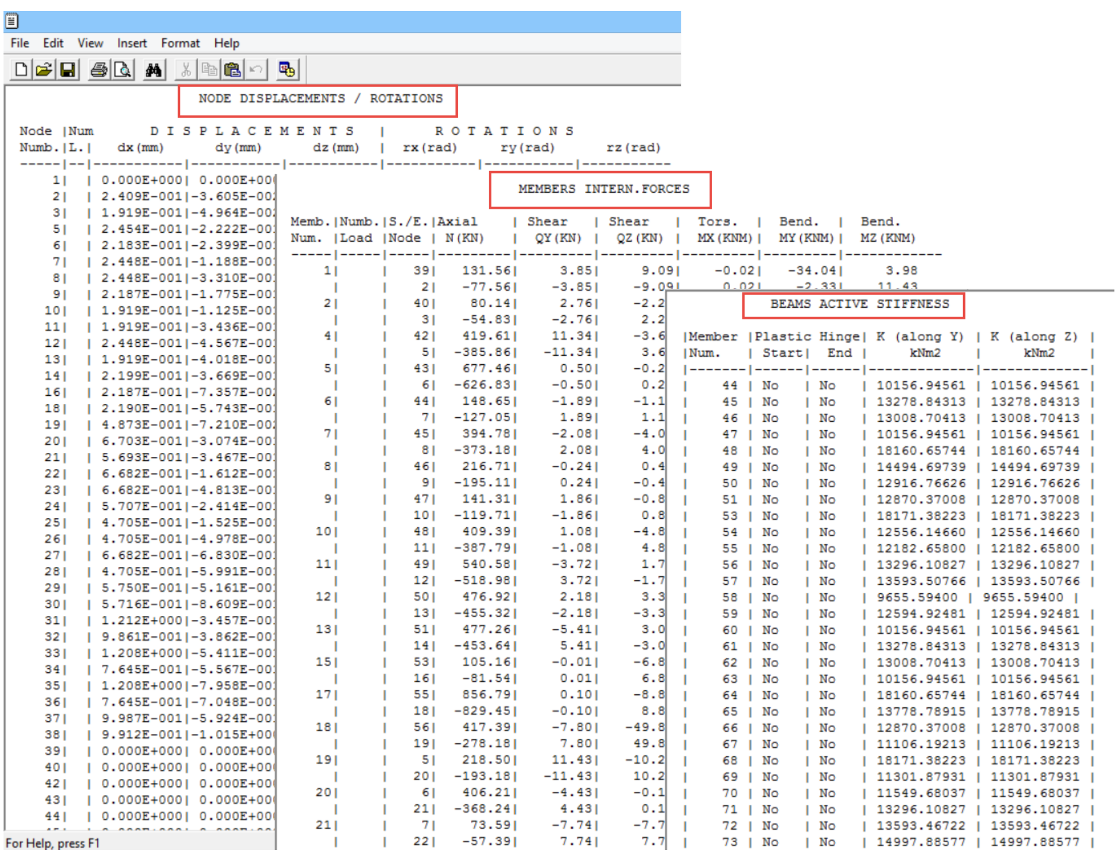
⚠ The Interventions procedure is explained in “Members Design” chapter.

⚠ In both processes of evaluation and intervention, no elements with lack of structural integrity must be met for any analysis type of the considered Limit State.

Press the 

command to open the txt file containing:

- NODE DISPLACEMENTS AND ROTATIONS
- MEMBER INTERNAL FORCES
- BEAMS ACTIVE STIFFNESS



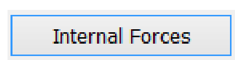
2.3 Seismic Force:

§ Seismic Action of Scenarios Of seismic elastic Static analyzes

§ SCENARIO : 0 – DATA AND RESULTS OF SEISMIC FORCE
§ DATA FILE LOAD CASES
§ RESULTS FILE – INTERNAL FORCES
§ SEISMIC ACTION ALONG THE MAIN DIRECTIONS OF THE BUILDING
§ Calculations Parameters
§ Fundamental Periods (Proximity Type)
§ Height Distribution Of Seismic Force (Shear-Moment)

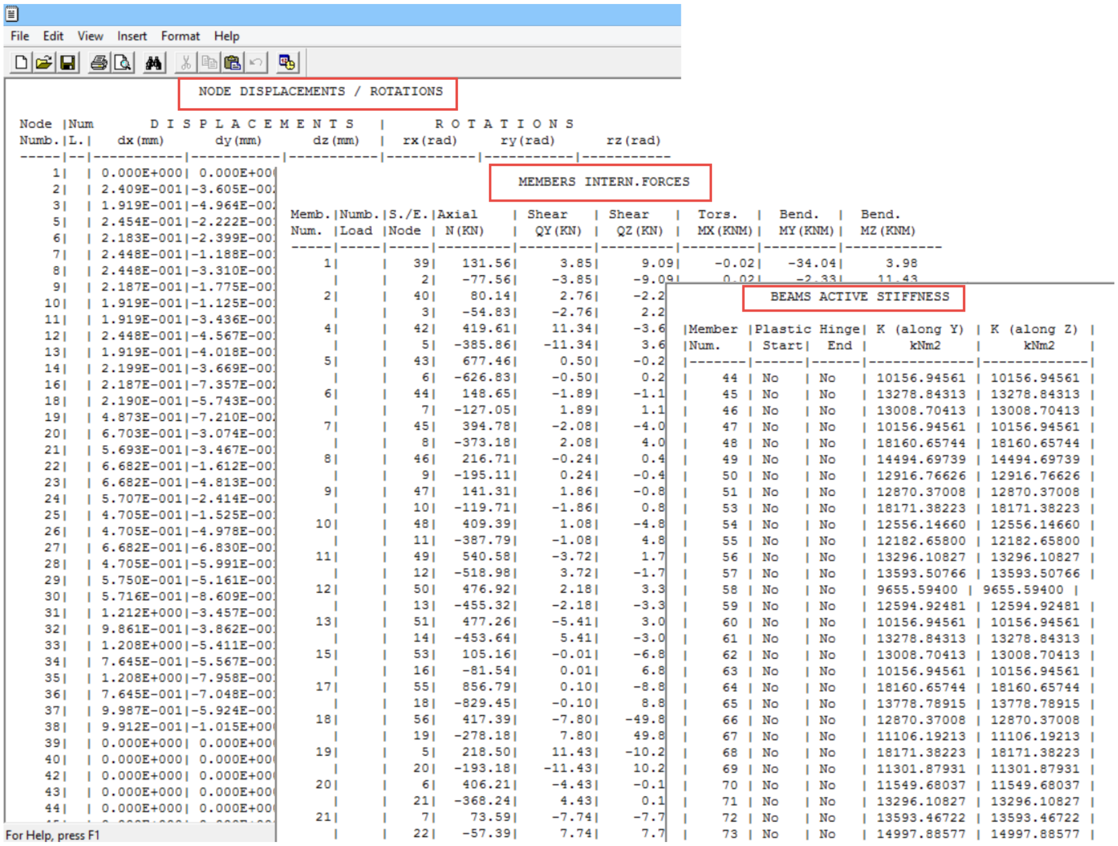
§ Seismic Action of seismic inelastic analyzes

§ SCENARIO : 0 – DATA AND RESULTS OF SEISMIC FORCE
§ DATA FILE LOAD CASES
§ RESULTS FILE – INTERNAL FORCES
§ SEISMIC ACTION ALONG THE MAIN DIRECTIONS OF THE BUILDING
§ Calculations Parameters
§ Fundamental Periods (Modal Resp.Spect.analysis)
§ Eigenvalues Participation Factors
§ Mass Participation Factors / Direction
§ Active Modal Masses
§ Acceleration Response Spectrum Matrix Values
§ Limit States - Elastic Response Spectra

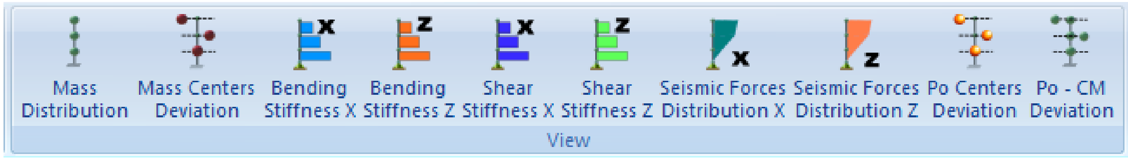
Moreover, through the window ***Report***, the button 

Press the command to open the txt file containing:

- NODE DISPLACEMENTS AND ROTATIONS
- MEMBER INTERNAL FORCES
- BEAMS ACTIVE STIFFNESS


File Edit View Insert Format Help
NODE DISPLACEMENTS / ROTATIONS
Node Numb. | Num L. | D I S P L A C E M E N T S dx (mm) dy (mm) dz (mm) | R O T A T I O N S rx (rad) ry (rad) rz (rad)
MEMBERS INTERN.FORCES
Memb. Num. | Numb. Load | S./E. Node | Axial N (KN) | Shear QY (KN) | Shear QZ (KN) | Tors. MX (KNM) | Bend. MY (KNM) | Bend. MZ (KNM)
BEAMS ACTIVE STIFFNESS
Member Num. | Plastic Hinge Start | End | K (along Y) kNm2 | K (along Z) kNm2
For Help, press F1

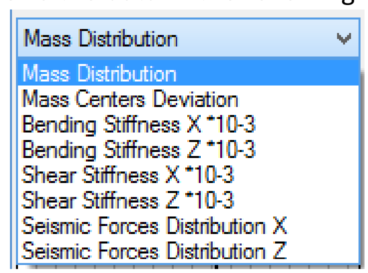
3. VIEW



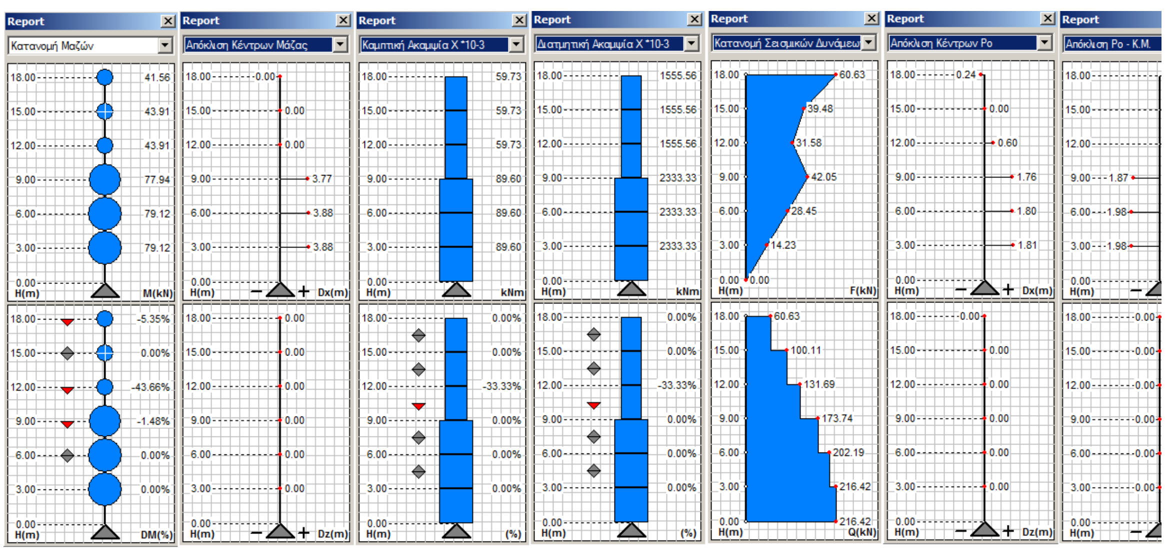
“View” command group is used in a different way for Linear and Nonlinear Scenarios.

§ For Linear Analysis

For Linear Scenarios: the commands are used so as the user reviews the data in the following drop-down list.

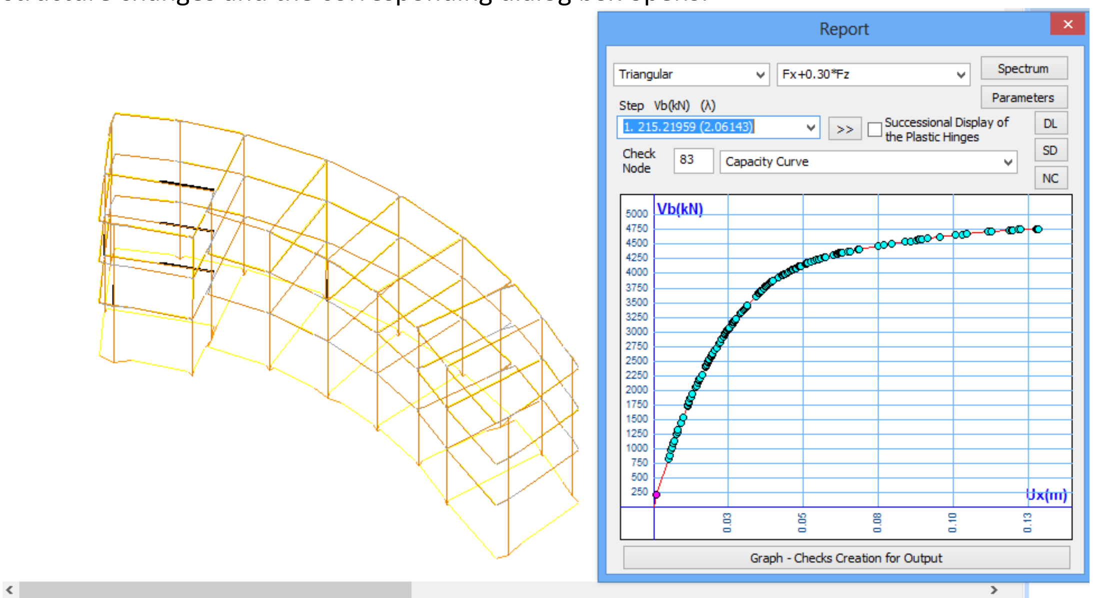


By each selection, the diagram with the corresponding title opens.

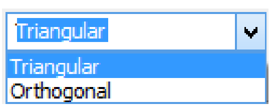


§ For Nonlinear (Pushover) Analysis

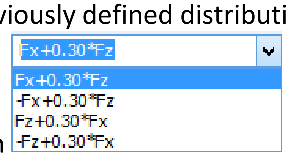
For Nonlinear (Pushover) Scenarios:
Select one of the "View" commands (for example "Mass Distribution"). The depiction of the structure changes and the corresponding dialog box opens:



It’s a new tool used for the presentation of the capacity curve of all Pushover analyses and the corresponding view of the deformed and undeformed shape of the structure in each step. In the top of the window select one of the previously defined distributions

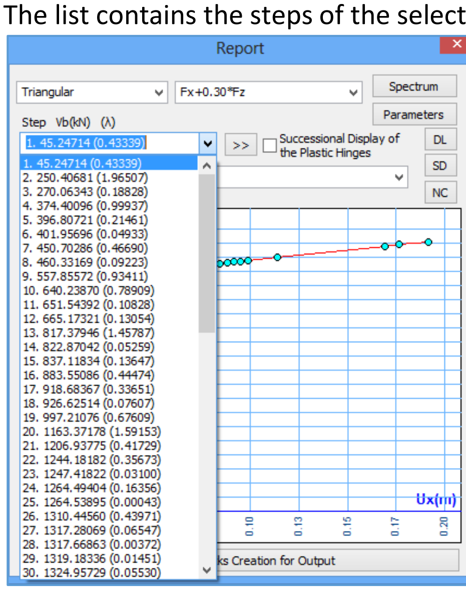


and a load combination



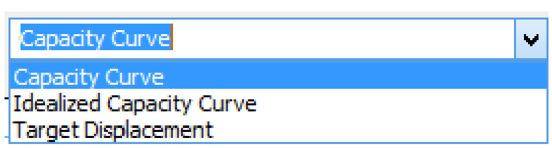
.

The list contains the steps of the selected nonlinear analysis.



Each step provides the corresponding shear value Vb(kN) and the λ load factor, while the following data can be shown graphically:

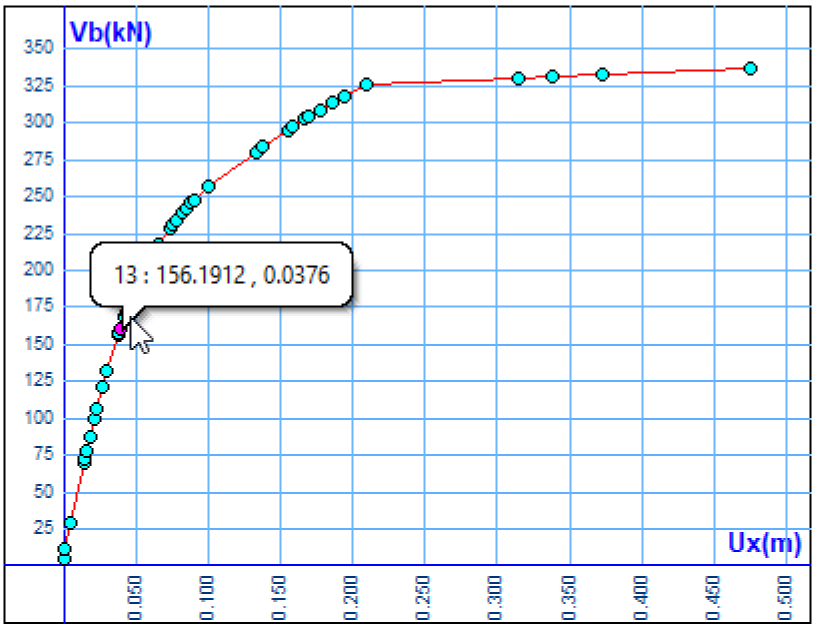
- § Capacity Curve
- § Idealized Capacity Curve
- § Target Displacement

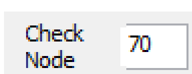


§ 1.Capacity Curve

The capacity curve represents the nonlinear relationship between the base-shear force and the displacement of the control (Check) node.
The points depicted in the capacity curve are the “Steps” of the pushover analysis. The selected step is displayed with pink color and represents the time of a plastic hinge creation.

Approaching the mouse to the points of the steps the indication of the step’s number and the corresponding values of Vb and Ux is displayed.

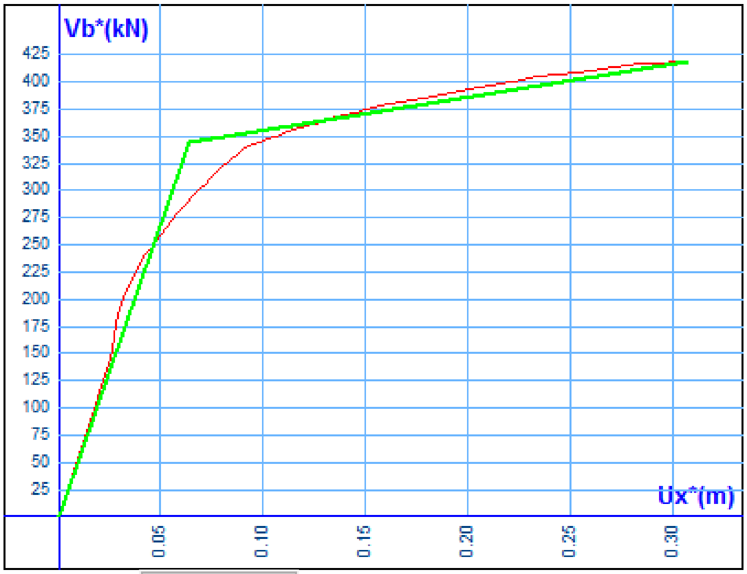


In 

you can change the check node and derive the results for a different check node, without repeating the analysis. The results’ presentation is automatically updated.

§ 2.Idealized Capacity Curve

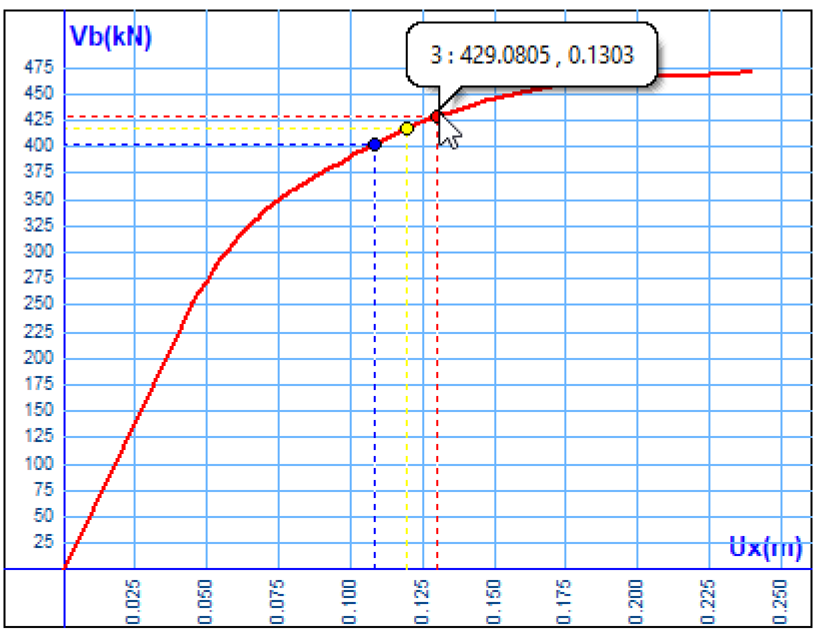
It is the linearized capacity curve by the procedure given in Annex B of EN 1998-1. The idealized elasto-plastic force- displacement’s relationship is calculated.



The button Parameters is used in the definition of the parameters on how to make the capacity curve bilinear. This bilinear curve is necessary in order the slopes of its two branches to be used to calculate the period and the corresponding spectral acceleration.

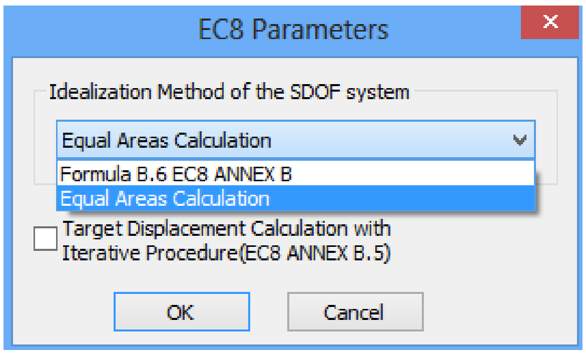
§ 3.Target Displacement

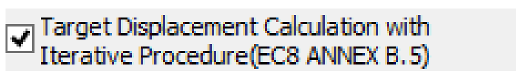
The target displacement of the check node, for the tree different Limit States, is determined according to the Displacement Coefficient Method given in Annex B of EN 1998-1.



Approaching the mouse to the points, the values for the three target displacements are displayed, one for each level of performance and the corresponding shear to the check node.

is similar to the Spectrum command explained in the section regarding the parameters of the scenario. Note that these parameters are considered in the calculation of the target displacement related to the seismic demand and not to the structural capacity. For that reason, these parameters are irrelevant to the capacity curve and so they can be defined or modified before or after the implementation of the analysis procedure.
In *Parameters* select the method for the derivation of the capacity curve ("Idealization Method") between the two methods proposed in Annex B of EN 1998-1.



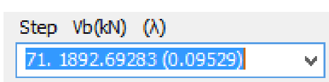
Activate


and the program uses the Iterative Procedure proposed in EC8 for the calculation of the target displacement.

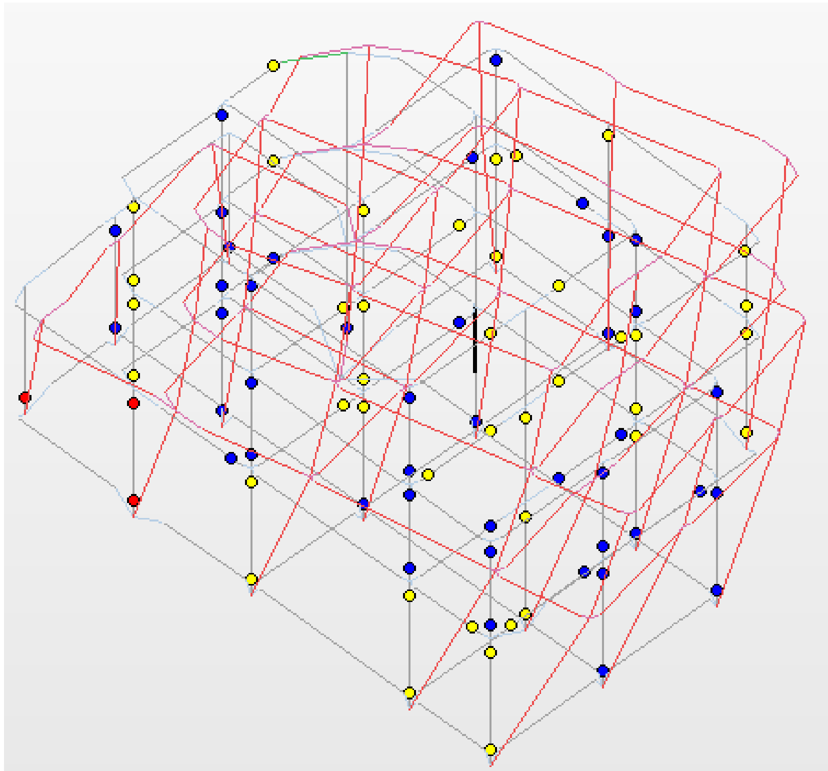
§ Representation of the structure

The deformed shape of the structure as well as the location of the plastic hinges, formed in the edges of the structural elements for each step, are depicted in real time.
There are two methods:

1. Select the step from the list



and you will see the corresponding deformed shape of the structure in the selected step and the location of the plastic hinges.
The undeformed shape of the structure is displayed along with the deformed one, in gray and red color respectively.
Colored dots indicate the edges with the plastic hinges and the different colors refer to the damage levels.



According to the chord rotation of the plastic hinges, dots have three different colors:

- Blue color, when the chord rotation is less than the (θum – θy)/2γRd value.
- Yellow color, when the chord rotation is between the (θum – θy)/2γRd and (θum - θy) values.
- Red color, when the cord rotation is more than the (θum – θy) value.

2. Otherwise, select the first step and press >> to see the structure's motion and the plastic hinges' formation. Press >> to stop it. You can have the same effect by clicking on a "step" in the list and turning the mouse wheel.

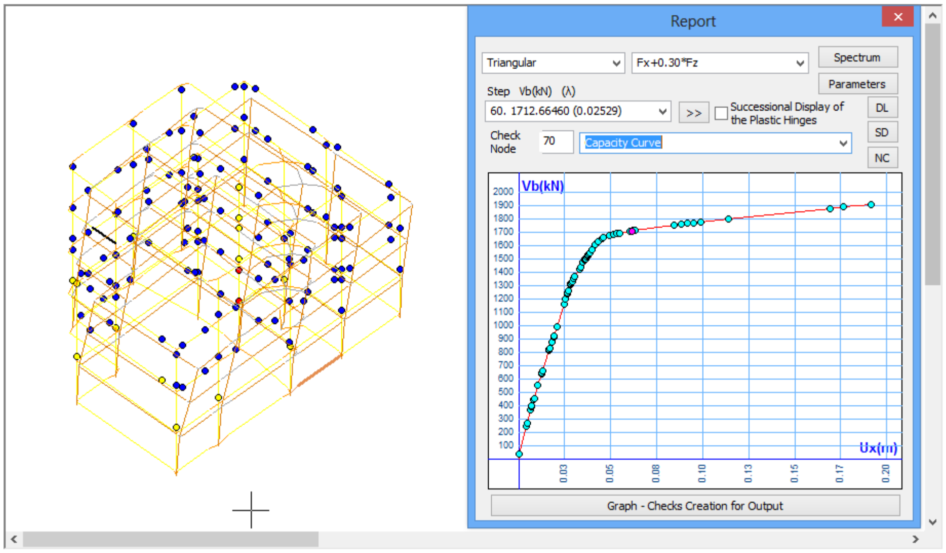
A-DL, B-SD, and Γ-NC buttons gives you the deformed shape of the structure for the three Limit States. This means that the structure is displayed on the step, on which the deformation of the check node is equal to the corresponding target displacement.

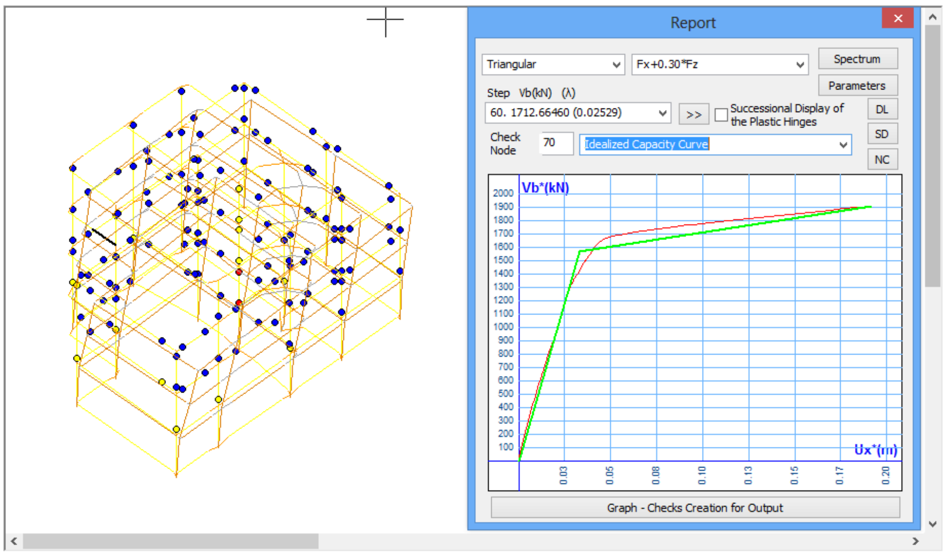
The check points on the curve correspond to the three LS:

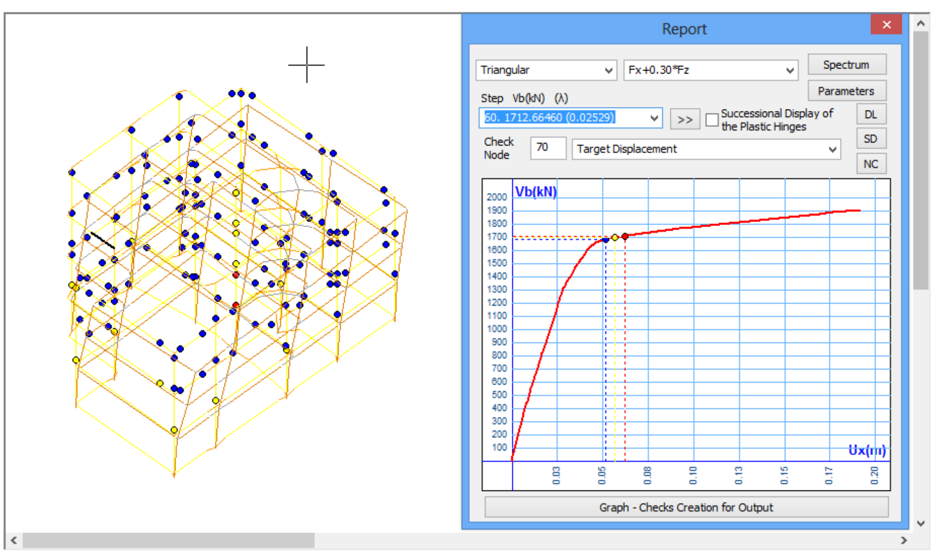
A-DL : blue

B-SD : yellow

Γ-NC : red

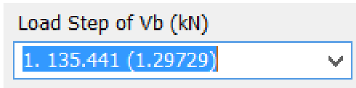

Report
Triangular
Fx+0.30*Fz
Spectrum
Parameters
Step Vb(kN) (λ)
60. 1712.66460 (0.02529)
Successional Display of the Plastic Hinges
DL
SD
NC
Check Node
70
Capacity Curve
Vb(kN)
Ux(m)
Graph - Checks Creation for Output


Report
Triangular
Fx+0.30*Fz
Spectrum
Parameters
Step Vb(kN) (λ)
60. 1712.66460 (0.02529)
Successional Display of the Plastic Hinges
DL
SD
NC
Check Node
70
Idealized Capacity Curve
Vb*(kN)
Ux*(m)
Graph - Checks Creation for Output


Report
Triangular
Fx+0.30*Fz
Spectrum
Parameters
Step Vb(kN) (λ)
60. 1712.66460 (0.02529)
Successional Display of the Plastic Hinges
DL
SD
NC
Check Node
70
Target Displacement
Vb(kN)
Ux(m)
Graph - Checks Creation for Output

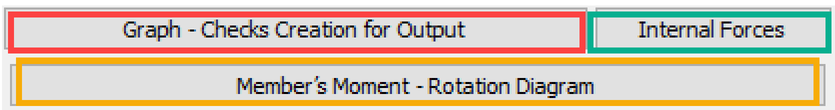
- The graphics are generated per distribution (Rectangular, Triangular) and seismic combination.

- By choosing a type of distribution and seismic combination from the list,

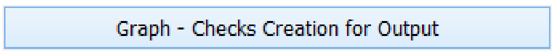


the steps of this inelastic analysis for each step, the Shear Vb(kN) λ load factor and the corresponding minimum (λ) factor are displayed. The corresponding point is also marked on the capacity curve, in pink

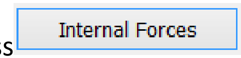
At the bottom side of the window



- The selection of



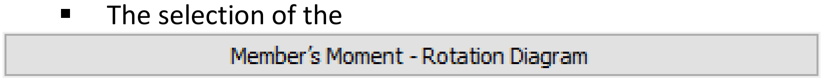
command is necessary for the syntax of the documents, which contain the graphs and the design checks of the project, as well as any updates in case of modifications.

- Press 

command to open the txt file containing:

NODE DISPLACEMENTS AND ROTATIONS
MEMBER INTERNAL FORCES
BEAMS ACTIVE STIFFNESS

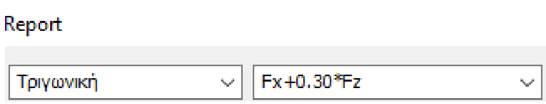
- The selection of the



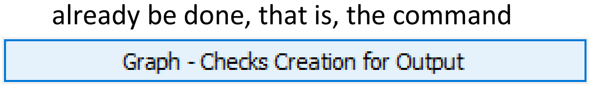
command displays the member's moment-rotation diagram per member (start-end) as well as direction.

§ Member's Moment-Rotation Diagram

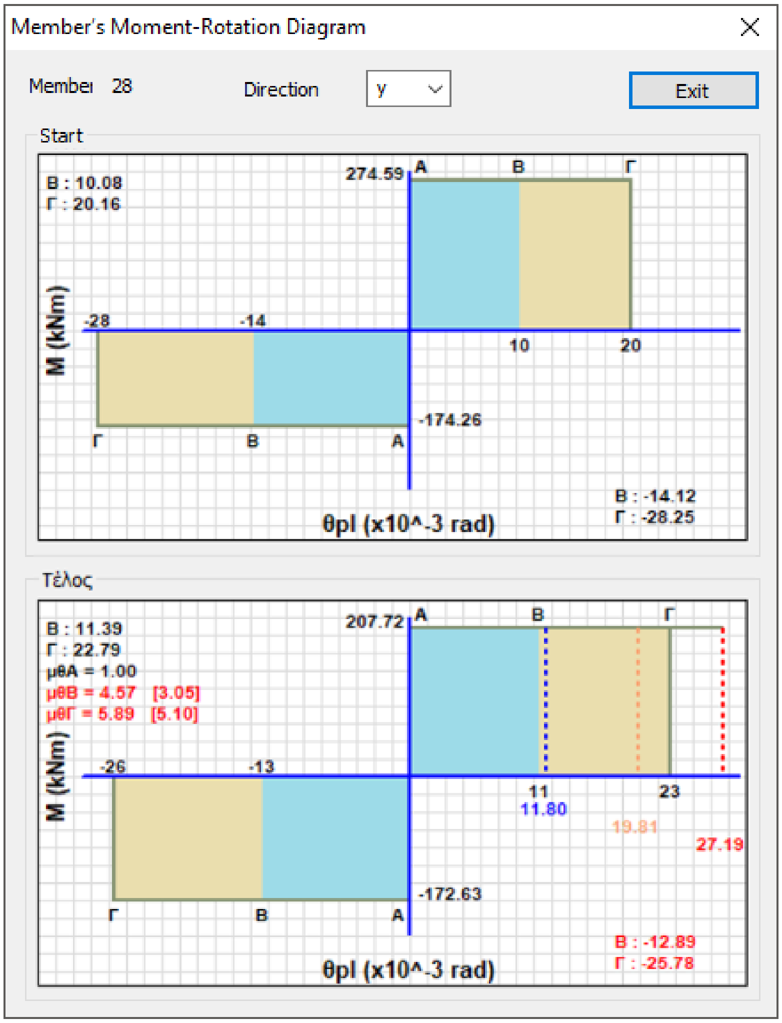
Selecting/command and pointing by left clicking a beam or column member, the member's moment-rotation diagram is displayed per member (start-end) as well as direction for the selected distribution.



A prerequisite to display the member's moment-rotation diagram is that the checks have already be done, that is, the command



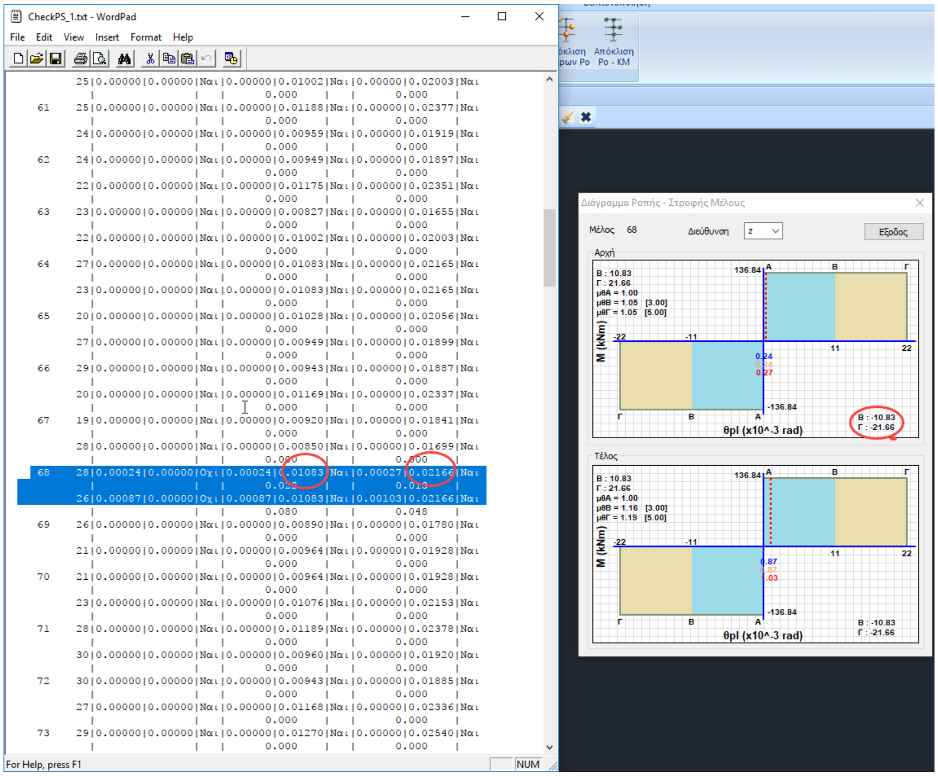
has been selected



This diagram is based on the following assumptions:

- The calculation of moment My is made according to the relation (A.6) of the Kanepe's annex 7A.
- The value of My is different for each step due to the axial force that is implicated in its calculation.
  For this diagram, however, a fixed value was used and is the one derived from the axial force of the vertical loads.
- Two values of My (positive and negative) are calculated and two areas with the (different) limits for the performance levels are designed respectively.
  For columns, due to symmetrical reinforcement, the two values will always be the same.
- As well as it is already known the diagram doesn't have an elastic branch, so only the plastic area is displayed.
- The **θ** values have been divided by the corresponding safety coefficients. The ultimate θ_{pl} limits corresponding to the respective performance levels have been divided by the safety coefficient γ_{rd}=1.8, while the angles of rotation θ_{sd} have been multiplied by the coefficient.

This is done to make sure that compatibility with the corresponding print results has been achieved.



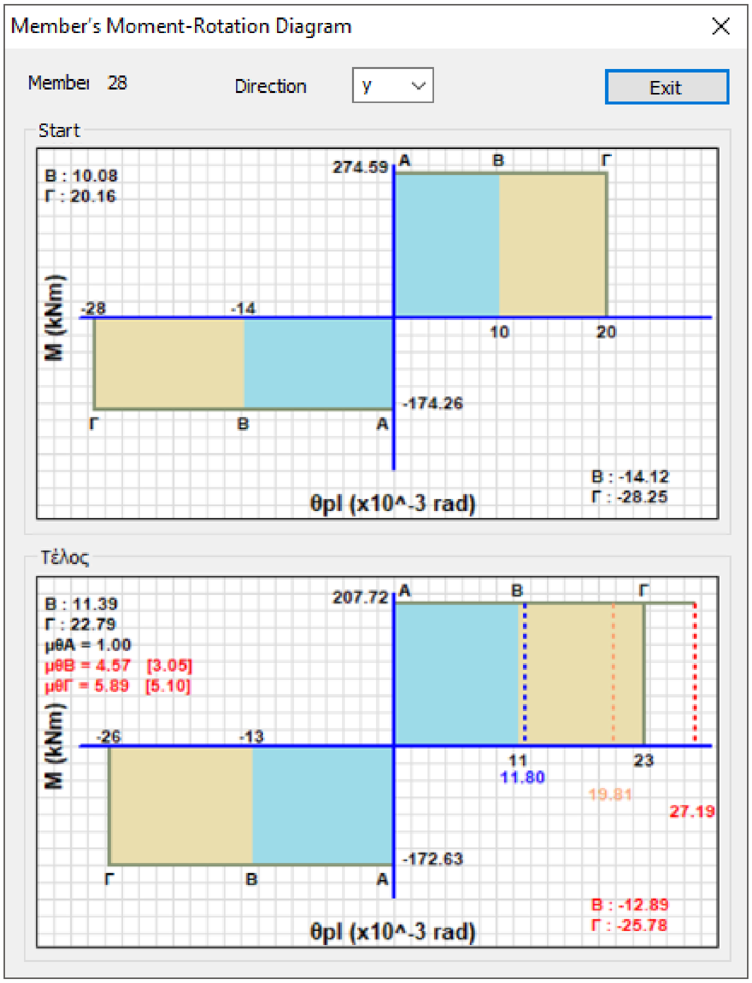
The diagram shows the angle of rotation of the plastic joint (requirement) for the three steps of the analysis corresponding to the three levels of performance:

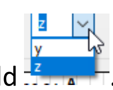
A: blue **B**: orange **C**: red

The values are displayed, depending on the sign of the angle, in the corresponding area.

In the dialog box that appears:

The corresponding diagram for each edge (start-end) is displayed.



The direction is selected through the corresponding direction field .

⚠ Especially for beams, the predetermined direction is the principal direction z assuming however that the angle of rotation of the plastic joint is the worst magnitude of both directions.

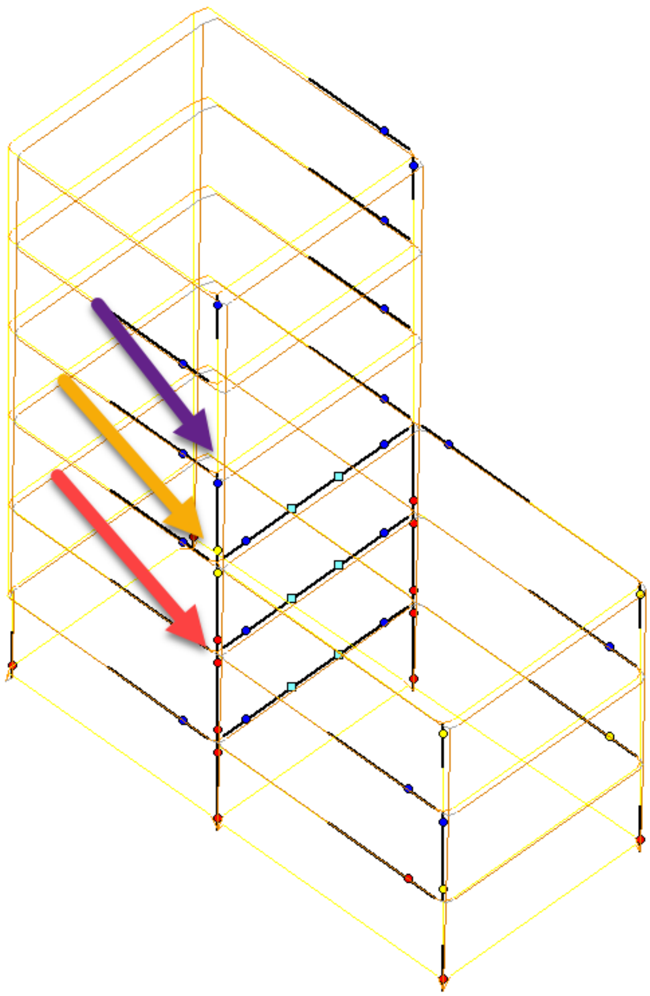
Two colored areas appear, one for the positive and one of the negative values of the axis, where the **blue** represents the **B** performance level, while the **brown** one represents the **C**.

The **black** values are the **limits** for each level of performance.

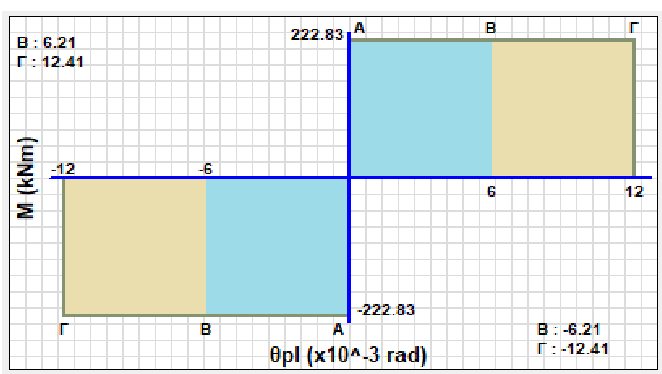
⚠ In the diagram, they appear as an integer. But in the bottom left for the negative and in the upper left for the positive they are written with their decimal digits.

The colors appearing in the circles at the ends of each member in the three-dimensional structure depend on the corresponding angle of rotation of the plastic joint.

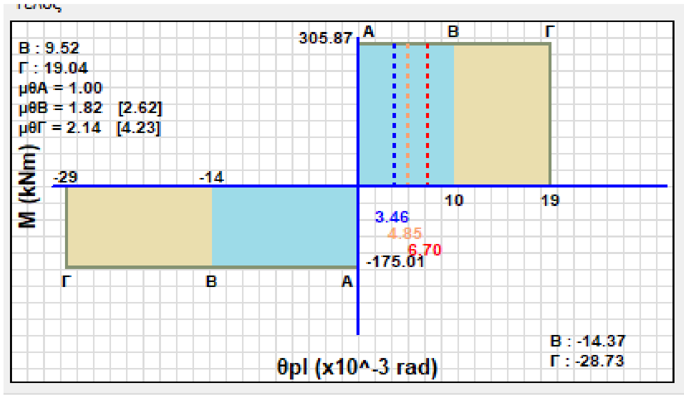
More specifically:



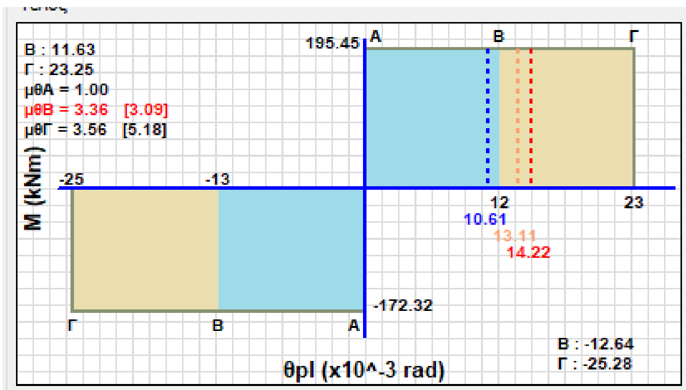
No value means that the end has not developed a plastic joint:



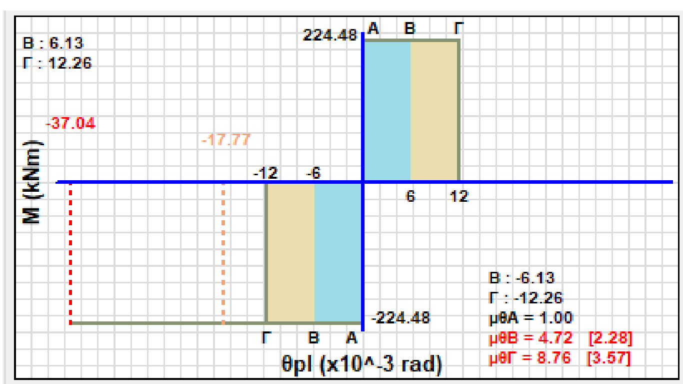
The **blue** color means that the respective blue line is inside the blue area, that is, the limit A (which is value 0) is exceeded, but both this and the other two values have not exceeded the boundary of the B (blue area).



The yellow color means that the respective value (orange line) has entered the brown area and the respective red line has not gone out of the brown area.



Finally, the **red** color means that the respective red value has gone out of the brown area.



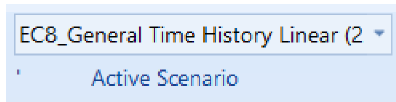
⚠ All of the above are valid as long as the structure is on the step that corresponds to the performance level C, so as all of the above are applied.

Also, the ductility indices regarding angle of rotation μθ for each level of performance are also given. First, the required is indicated and then the one available is written in brackets.

The values are displayed with red color if the first value is bigger than the second one.

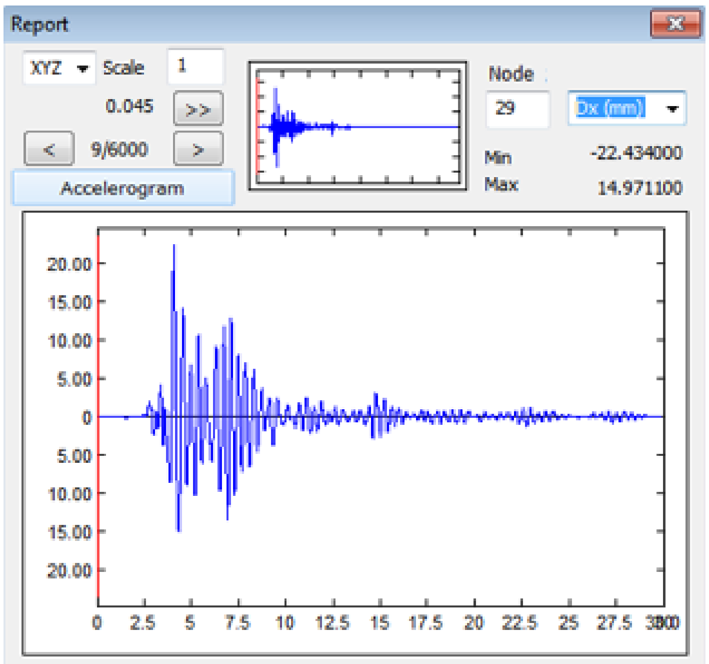
For the first performance level μθA=1.

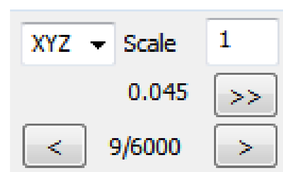
§ For Time History Analysis

“View” commands activated 

:

After completing the analysis, press a command from the "View" menu that opens the following window:





where the user can choose the direction of the seismic action (X, Y, Z or XYZ) and the scale that the result of the analysis will be visualized. Type the number of the node to see the response.

The graph of the response of the selected node over time with the maximum and minimum value is automatically displayed.
At the same time in the upper window, the selected Accelerogram of the selected seismic action is displayed.
Finally, it is possible to view the deformed state of the structure for each time step of the analysis. For this purpose, the model appears in the following three-dimensional representation, where the undeformed shape is presented in parallel with the movement of the deformed shape.