Atir Software Development LTD

STRAP - Seismic Analysis

Step by step

13 Kehilat saloniki, Tel Aviv, Israel
Tel: +972-3-6480129
Website: http://www.atir.com
Email: atir@atirsoft.com

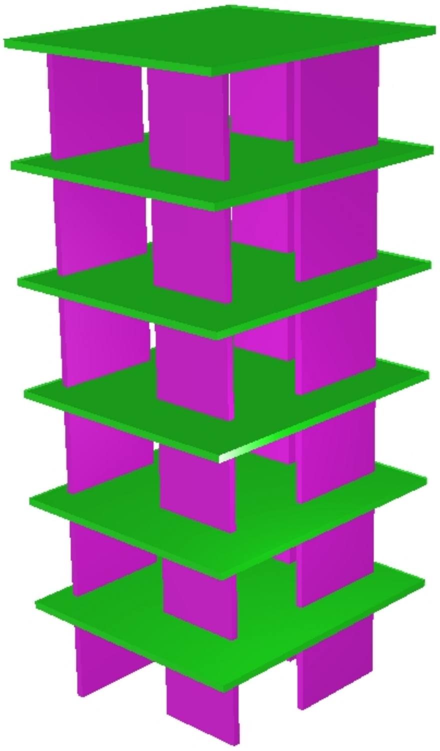
Table of contents

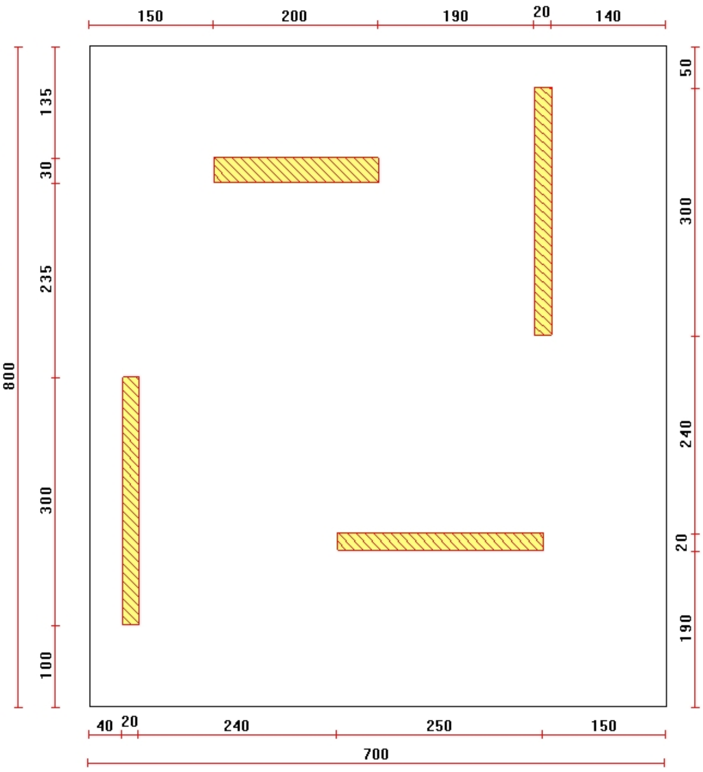
1. Abstract

The aim of seismic analysis is to calculate the reaction of the model to earthquakes. This example describes the method to analyze the building using Modal Analysis. Modal Analysis is the accepted method and is recommended by mode design codes. The reaction of the model for each mode shape is calculated according to the response spectrum given by the seismic code. Because the aim of this example is to describe the method to do Modal Analysis, we will use a simple six-story structure, with a story height of 3 meters. The seismic load carrying system consists of four shear walls.

2. Geometry Definition

We will define the model geometry and part of the loads using the AutoSTRAP program.



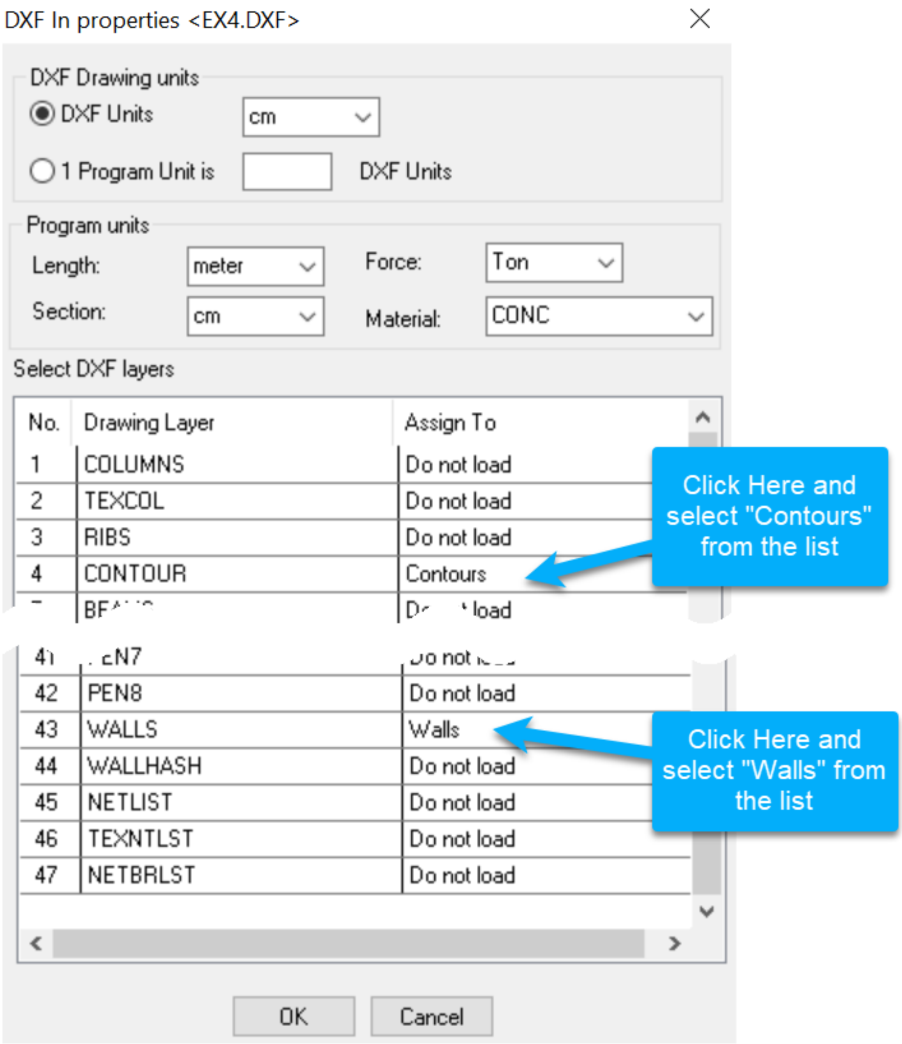


The DXF file can be downloaded from: EX4.DXF

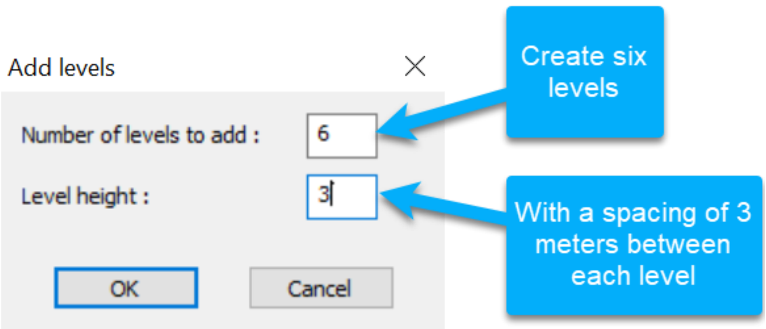
AutoSTRAP

- click the AutoSTRAP  icon on the desktop or from STRAP's models tab click *Utilities > AutoSTRAP*.
- click the "New" icon  or click *File > New-import DXF file*.
- select the DXF file that you downloaded and click *open*.

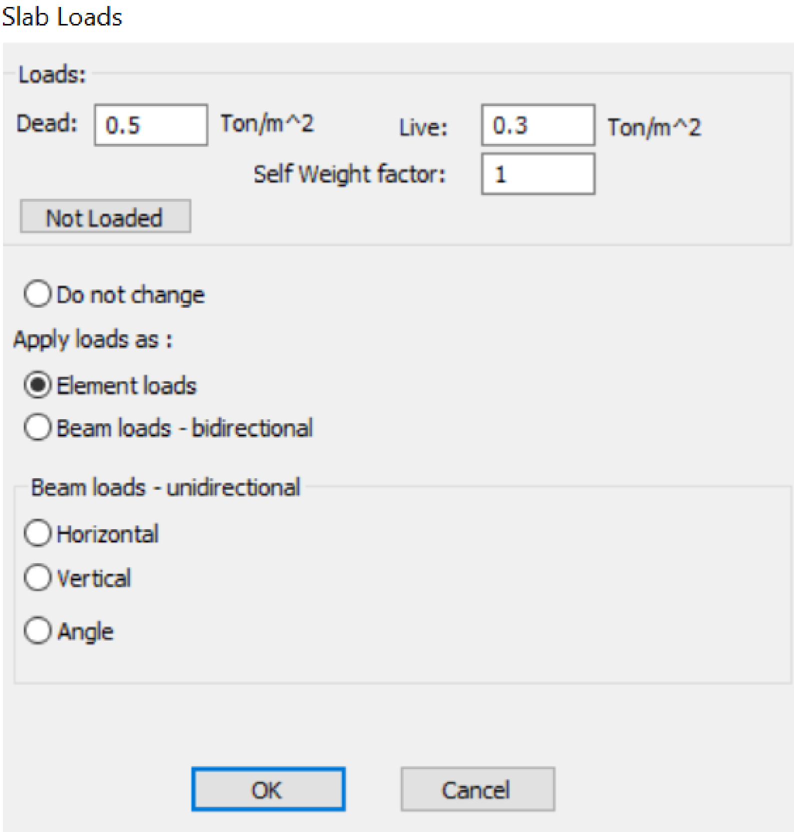
the program displays the following menu. The lines representing the contours are found in Layer #4 and the lines forming the walls are in Layer #43 and these layers must be identified and assigned:



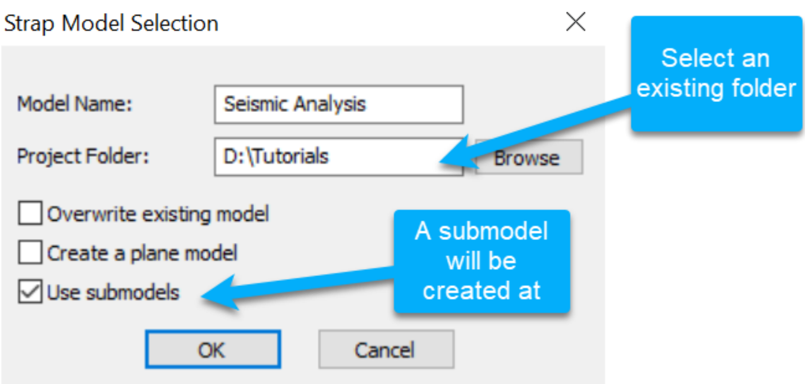
- the elements identified by the program are displayed on the screen. To display the slab element grid, click on the icon.
- click on Levels in the side menu and *Add Level* at the bottom of the dialog box:



- The program adds the list of levels to the Dialog box. click *End* to continue.
- Click **Loads** in the lower side menu, then select **Slab load** in the upper side menu.
- Click *Select all spaces* and define the loads:



- Create the STRAP model: click **STRAP** in the bottom side menu.
- Click **Defaults** in the top side menu and check ☑ ***Create rigid links in plane.*** This option creates infinite rigidity in the plane of the floor slab.
- Click **Create model** in the top side menu



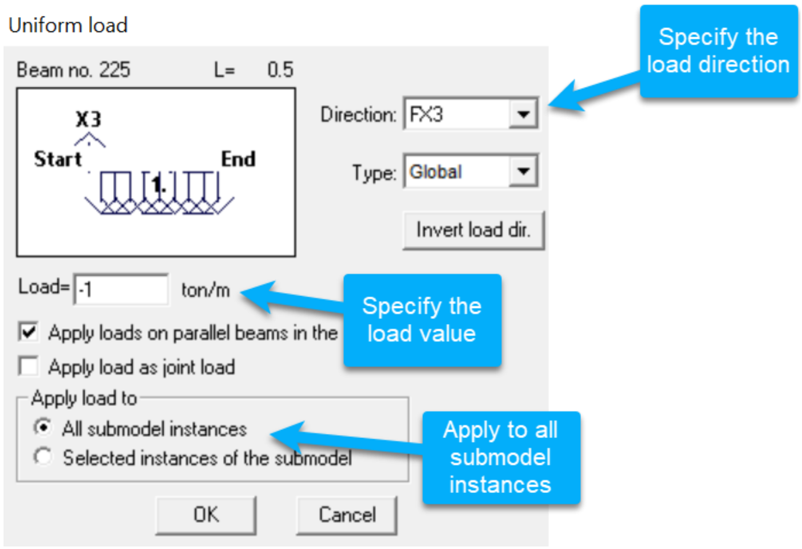
- Run STRAP and open the created model Seismic Analysis.
- click in the icon bar to display an isometric view.
- select the submodel to display from the list in the side menu: EX4
- click in the icon bar to display the walls.
- define "dummy beams" on the slab perimeter (for applying line loads to the slab): click Beams in the lower side menu, then click Define in the upper side menu.
- Set the following two parameters:
  - Prop. = 0 in the bottom dialog box (indicates that the beams are 'dummy').
  - ☑ Chain with previous beam in the options at the top of the side menu.
- Select the four corner nodes in sequence, then select the first node again to end the beam definition..
- display the Main model: Main model

3. Loads Definition

Add the walls self-weight and the masonry line loads along the slab perimeter (the slab self-weight was defined as part of the slab dead load):

- click the Loads tab.
- select Existing load in the side menu, select the "Dead" load case and click Revise load.
- select Wall loads in the lower side menu and select Define in the upper side menu.
- click on Self weight, select the local x3 direction and define a factor = -1.0.
- click on *Select all walls*; the load is applied to all the walls in the main model.
- select the first instance of the submodel: EX4#1
- select Beam loads in the lower side menu and select Define in the upper side menu.

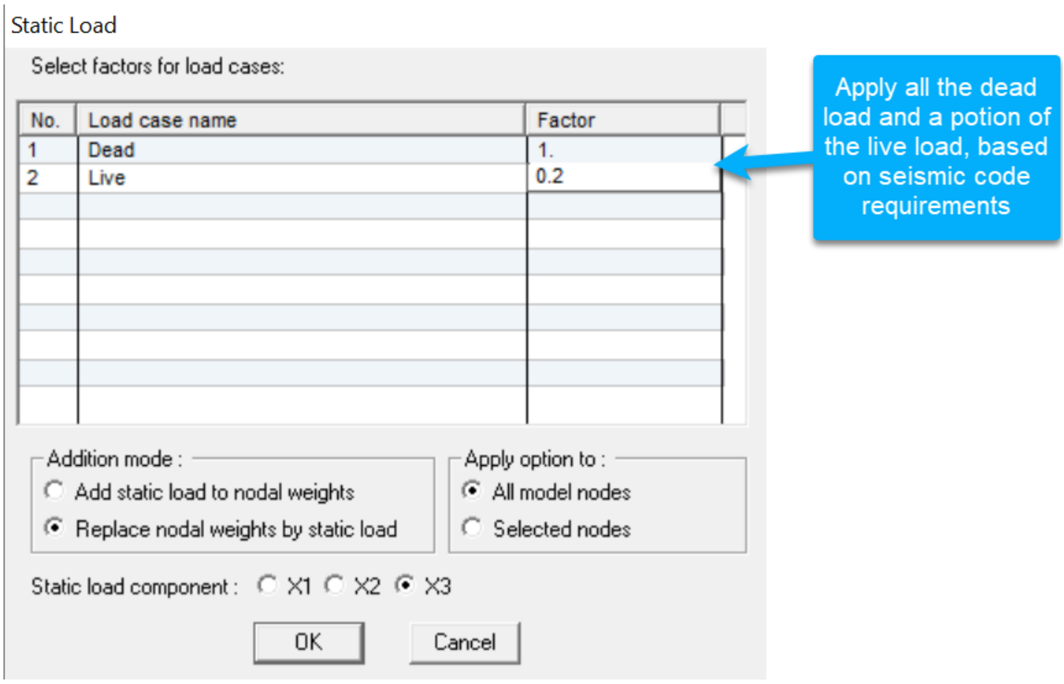
- click on Uniform, then *Select all beams* and define the loads:



- return to the Main model: Main model
- click End load case
- click 1+2= Solve to solve the model for static loads

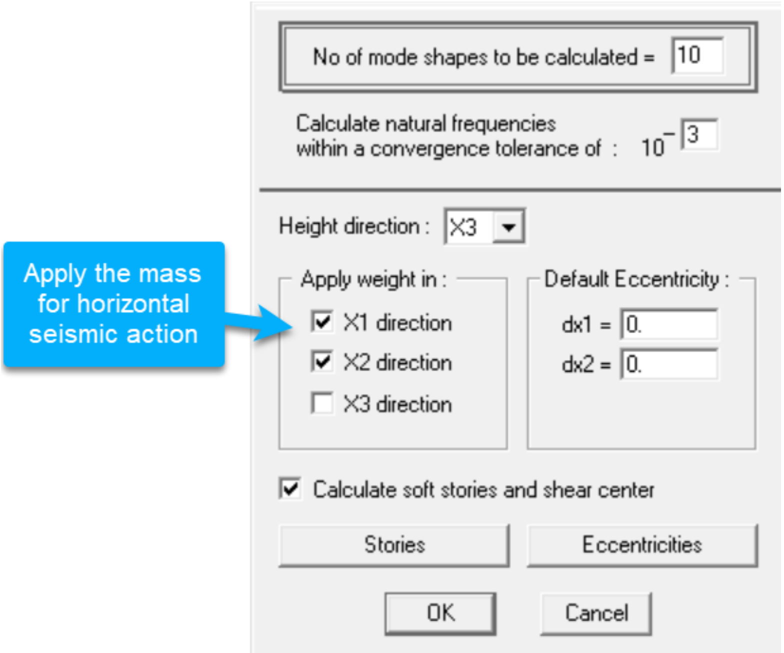
4. Modal Analysis

- click the Weights tab. The masses are defined from the static loads.
- select the Static load option:



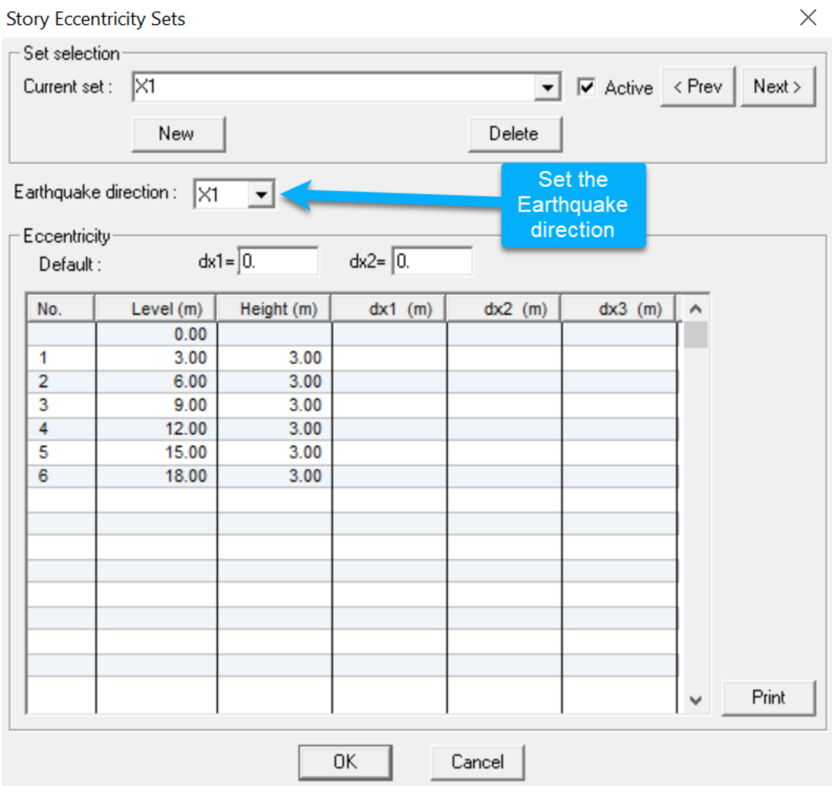
- select 

in the side menu:

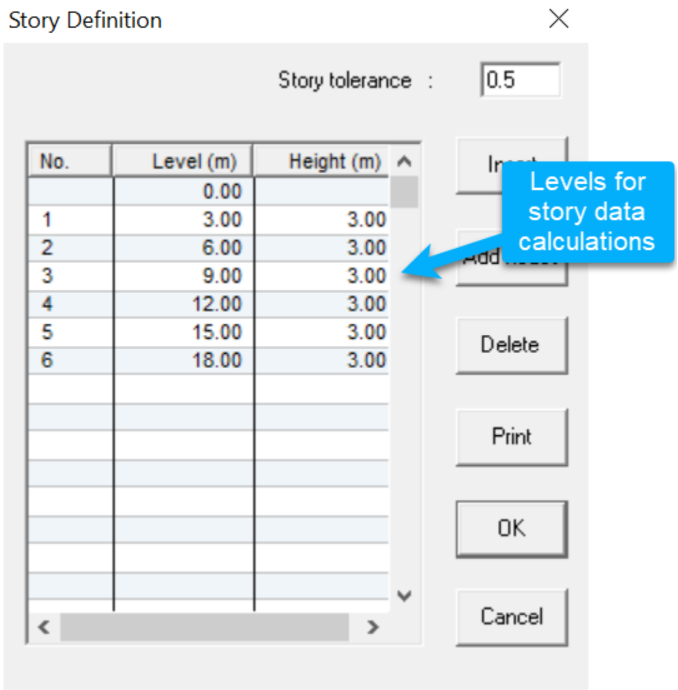


- click the 

button to define an earthquake in the X1 direction.



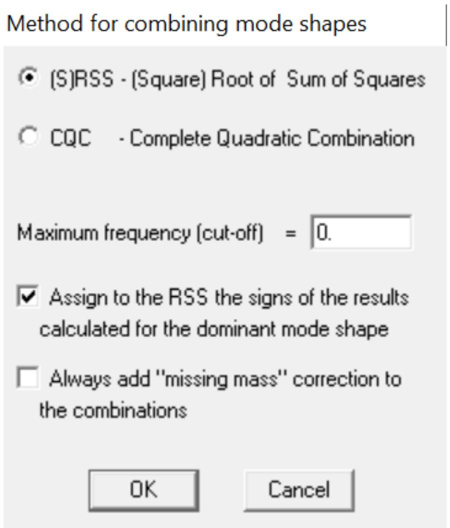
- click the Stories to verify the levels for Story data calculations.



- click 1+2= Solve to calculate the mode shapes. After the modal analysis is solved, you will get into the Dynamics tab.
- display the results graphically: select a mode shape and then click *Start Animation / End Animation* at the bottom of the screen to start/stop animation; repeat for other mode shapes.

5. Seismic Analysis

- ·Select the method for combining the modes: select *Seismic analysis* in the menu bar and *Method for combining modes* in the pulldown menu:



- **SRSS**: square root of sum of squares. The estimated response R (force, displacement, etc) at a specified coordinate is expressed as:

$$R = \sqrt{\sum_{i=1}^{N} R_i^2}$$

where Ri is the corresponding maximum response of the ith mode at the coordinate.

- **CQC**: complete quadratic combination. The estimated response is expressed as:

$$R = \sqrt{\sum_{i=1}^{N}\sum_{j=1}^{N} R_i \rho_{ij} R_j}$$

where ρ = the cross-modal damping coefficient.

Note:

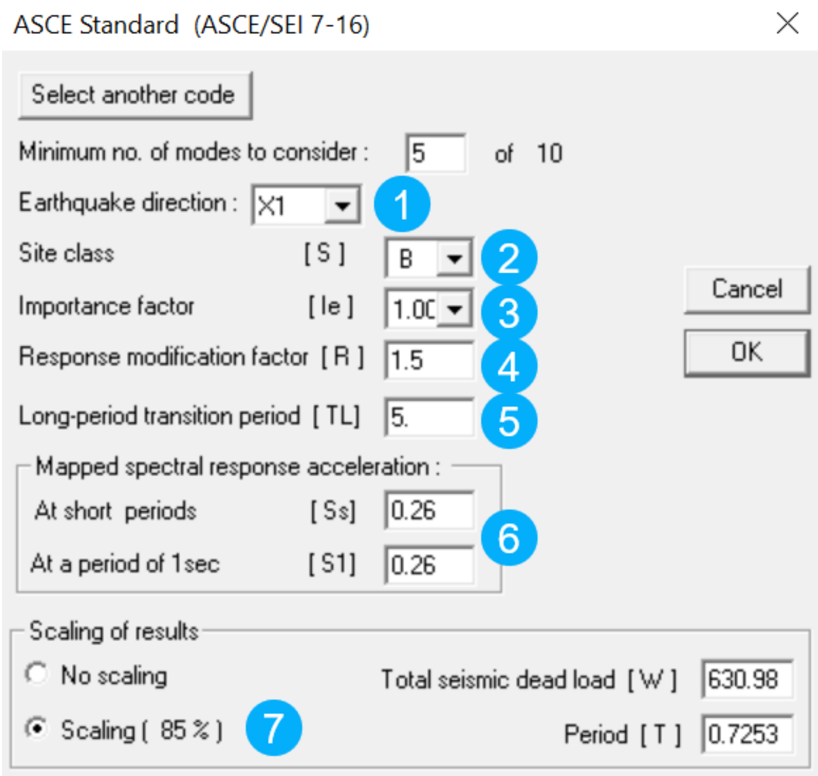
**** When some of the modes are closely spaced, the SRSS method may grossly underestimate or overestimate the maximum response. Large errors have been found in particular in space models in which the torsional effects are significant. The term "closely spaced" may be arbitrarily defined as the case where the difference between two natural frequencies is less than 10% of the smaller frequency.***

**** The CQC method is a more precise method of combining the maximum values of modal response.***

**** The two methods are identical for undamped models (x = 0).***

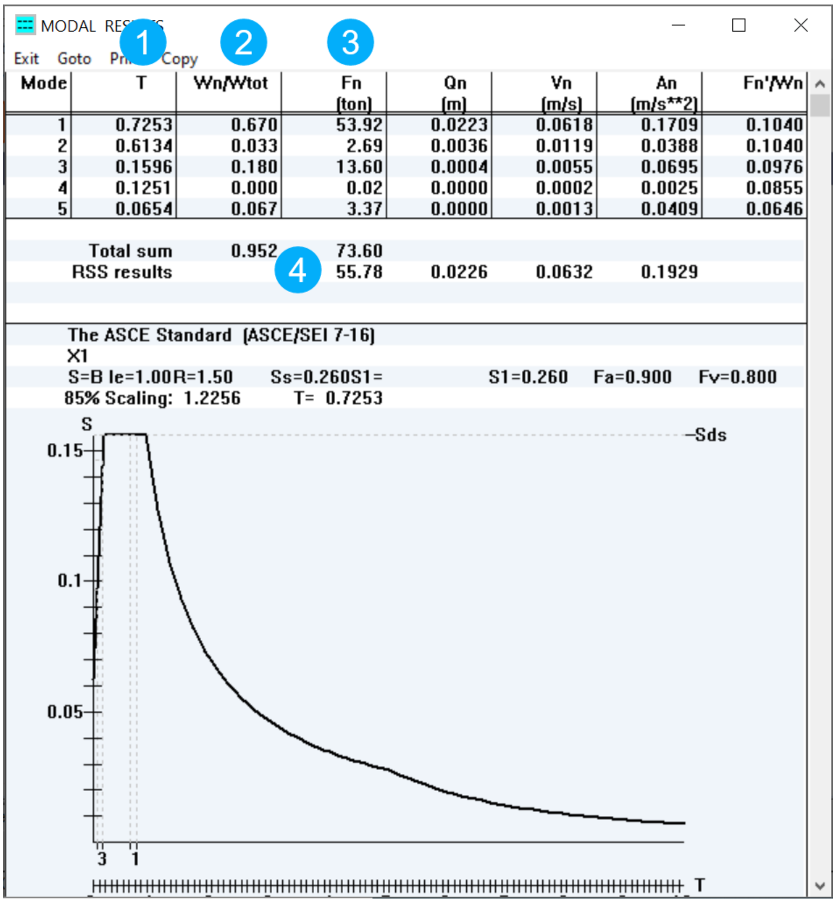
The seismic analysis for this example is done according to the ASCE/SEI 7-16 Code; you may select other Codes, e.g., Eurocode 8, NBC-Canada, etc.

- select Parameters in the side menu:



1. Specify the direction that the earthquake is applied. Select one of the global directions or define a vector as a combination of the three global directions. All mode shapes are used no matter in which direction the earthquake is applied. However, the modes which have deflections in the applied direction will dominate.
2. Specify the soil type as per Table 20.3.1.
3. Define the Importance factor as per Table 11.5.1.
4. Define the "Response modification coefficient" as per Table 12.2-1.
5. Specify a value of TL, calculated according to Section 11.4.5 and the Figures in Chapter 22.
6. Specify the mapped maximum considered earthquake spectral response acceleration at short periods (Ss) and at 1s (S1) as detailed in Section 11.4 and the Maps in Section 20.
7. Referring to Section 12.9.4, when the base shear calculated from the modal shape analysis is different than the base shear calculated according to the Equivalent lateral force procedure of Section 12.8, all corresponding responses, including moments and forces are adjusted accordingly.

- to display the modal results, click [Display ta...] and select [Display modal results].



1. period (seconds)
2. Participation factor: a factor reflecting the relative influence of the mode shape. The sum should be greater than 90%.
3. sum of external forces in all global directions.
4. the root of the sum of the squares of the horizontal forces.

- click **Story data** to display the story data results at each level:
  - Drift Calculations

Story drift calculations

The ASCE Standard (ASCE/SEI 7-16)

Amplification factor: 4. Max. drift/h : 0.025 Height direction : X3

No.	Level (m)	Height (m)	Drift (mm)	Max. Defl.	Min. Defl.	X1-Drift	X2-Drift	Weight(ton)
	0.00							
✓ 1	3.00	3.00	5.5	1.4	1.4	5.4	1.3	106.60
✓ 2	6.00	3.00	12.9	4.6	4.6	12.6	3.1	106.60
✓ 3	9.00	3.00	17.3	8.9	8.9	16.7	4.4	106.60
✓ 4	12.00	3.00	19.2	13.7	13.7	18.5	5.1	106.60
✓ 5	15.00	3.00	19.4	18.6	18.6	18.6	5.5	106.60
✓ 6	18.00	3.00	18.7	23.2	23.2	17.8	5.5	97.98

A check mark that indicates that the story drift is less than the allowable

The story drift calculation result

  - Stiffness and mass center

RIGIDITY AND MASS CENTERS (Units: ton, meter)

Exit Goto Print Copy

No.	Level	Mass X1	Mass X2	Rigidity X1	Rigidity X2	Difference DX1	Difference DX2
0	0.00						
1	3.00	3.442	4.056	2.998	4.238	-0.444	0.182
2	6.00	3.442	4.056	2.998	4.097	-0.444	0.041
3	9.00	3.442	4.056	3.001	4.094	-0.441	0.038
4	12.00	3.442	4.056	2.996	4.093	-0.446	0.037
5	15.00	3.442	4.056	2.973	4.047	-0.469	-0.009
6	18.00	3.468	4.030	3.201	4.288	-0.267	0.258

- Story shear forces and moments

SHEAR FORCES/MOMENTS (Units: ton, meter)

Exit Goto Print Copy

No.	Level	Story forces		Base shear		Story moments	
		F1	F2	V1	V2	M2	M1
0	0.00			55.78	16.69	740.57	230.61
1	3.00	3.48	0.34	54.07	16.38	577.47	180.58
2	6.00	7.68	1.11	49.55	15.32	420.18	131.50
3	9.00	10.22	2.13	42.32	13.22	275.45	85.59
4	12.00	12.20	3.31	32.22	9.94	150.66	45.94
5	15.00	14.73	4.56	18.21	5.38	54.62	16.14
6	18.00	18.21	5.38				

- Stability coefficient (Theta)

Stability Coefficient (theta)

The ASCE Standard (ASCE/SEI 7-16)

Reduction factor : 0.25

Height direction : X3

No.	Level (m)	Height (m)	Drift (mm)	Weight(ton)	Total Shear(ton)	Theta
	0.00					
✓ 1	3.00	3.00	5.5	630.98	55.78	0.0052
✓ 2	6.00	3.00	12.9	524.38	54.07	0.0105
✓ 3	9.00	3.00	17.3	417.78	49.55	0.0121
✓ 4	12.00	3.00	19.2	311.18	42.32	0.0118
✓ 5	15.00	3.00	19.4	204.58	32.22	0.0103
✓ 6	18.00	3.00	18.7	97.98	18.21	0.0084

A check mark that indicates that Theta is less than the allowable

The stability coefficient, Theta

- Week story

Weak stories calculations

The ASCE Standard (ASCE/SEI 7-16)

Allowable shear stress (concrete) : 0.6 mPa

Allowable shear stress (steel) : 150. mPa

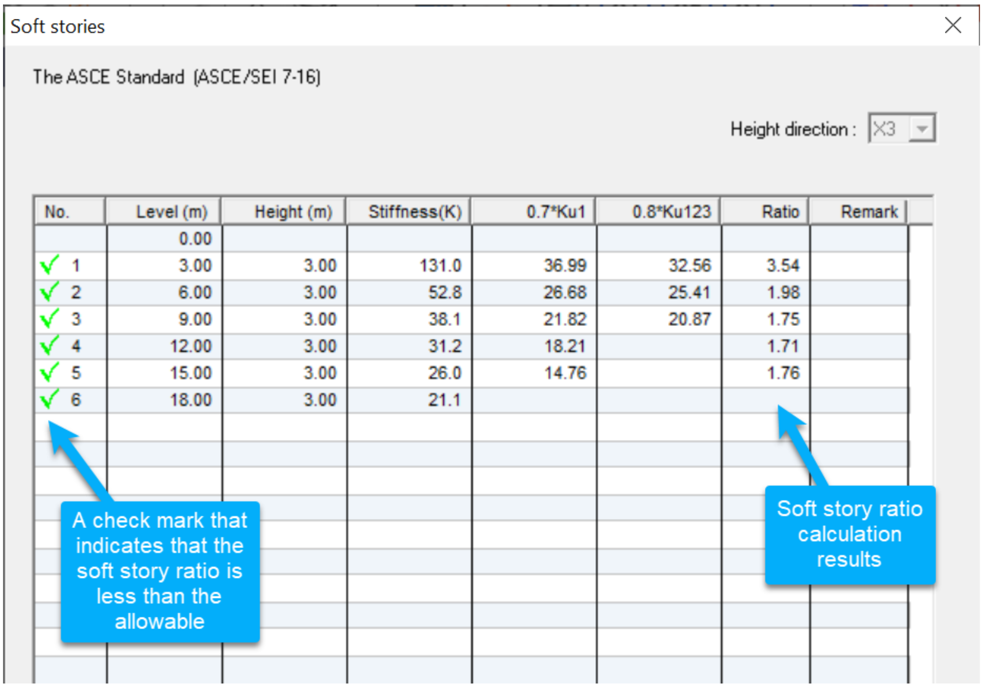
Height direction : X3

No.	Level (m)	Height (...	X1-Shear	Ratio	X2-Shear	Ratio
	0.00					
✓ 1	3.00	3.00	67.29	1.00	73.37	1.00
✓ 2	6.00	3.00	67.29	1.00	73.37	1.00
✓ 3	9.00	3.00	67.29	1.00	73.37	1.00
✓ 4	12.00	3.00	67.29	1.00	73.37	1.00
✓ 5	15.00	3.00	67.29	1.00	73.37	1.00
✓ 6	18.00	3.00	67.29		73.37	

A check mark that indicates that the week story ratio is less than the allowable

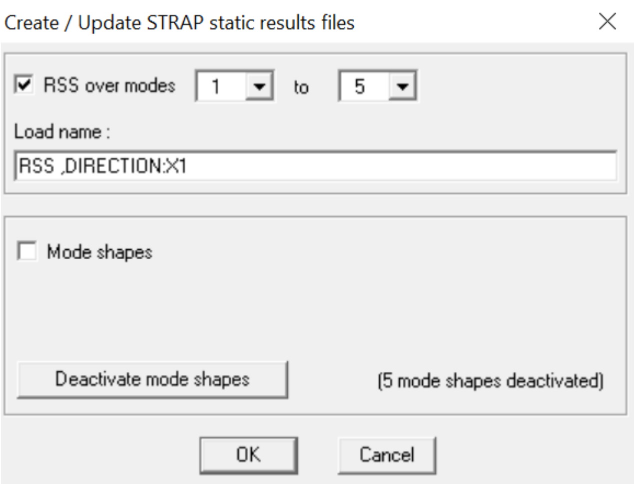
Week story ratio calculation results

- Soft story



No.	Level (m)	Height (m)	Stiffness(K)	0.7*Ku1	0.8*Ku123	Ratio	Remark
	0.00						
√ 1	3.00	3.00	131.0	36.99	32.56	3.54	
√ 2	6.00	3.00	52.8	26.68	25.41	1.98	
√ 3	9.00	3.00	38.1	21.82	20.87	1.75	
√ 4	12.00	3.00	31.2	18.21		1.71	
√ 5	15.00	3.00	26.0	14.76		1.76	
√ 6	18.00	3.00	21.1				

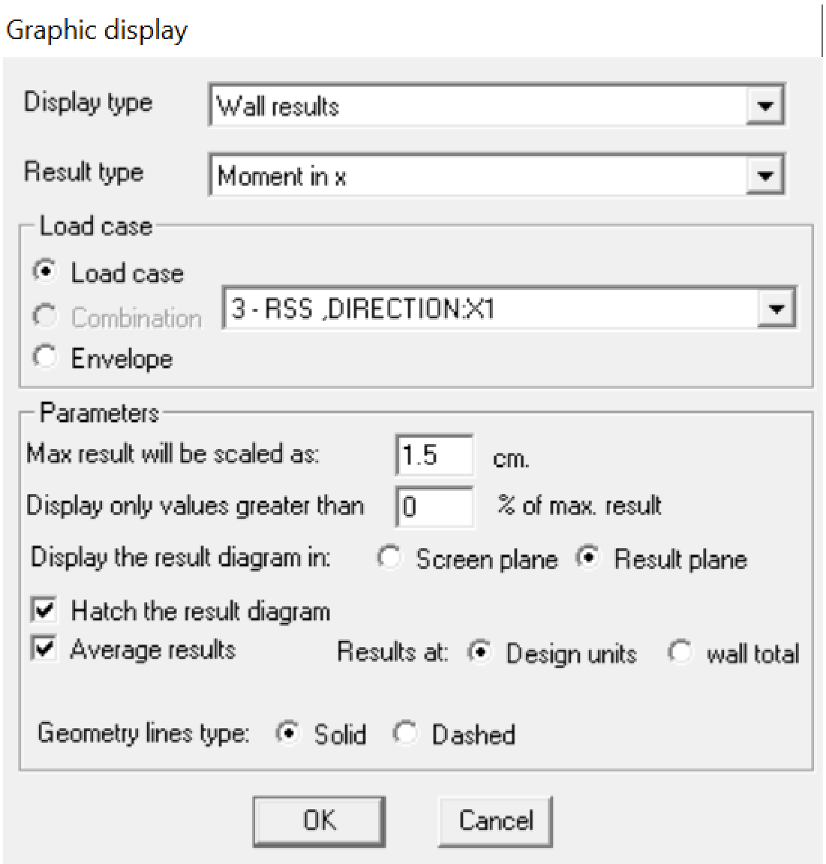
- click [Update re...] in the side menu to Create static load cases from the modal results and append them to the regular load cases:



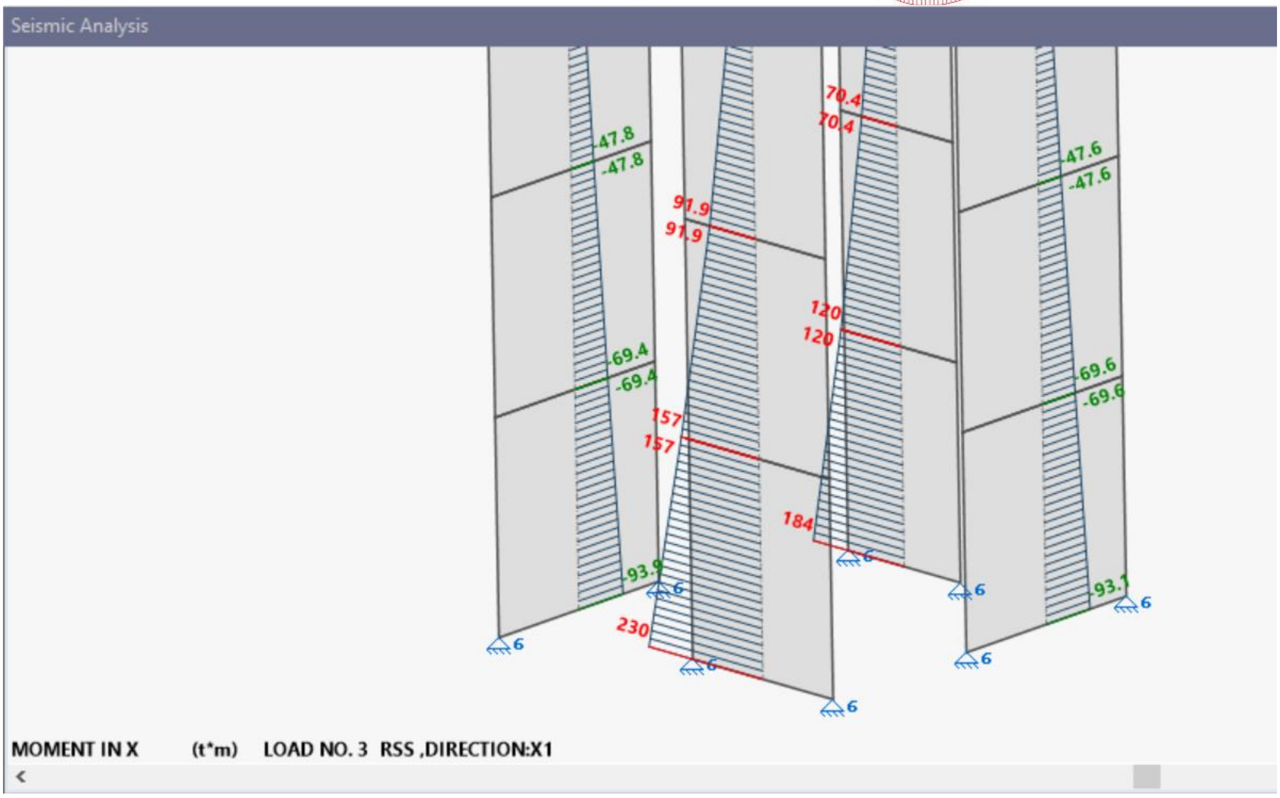
The program creates a fictitious load case representing an earthquake acting in the X1 direction, created by combining the mode shapes according to the code. The step should be repeated for the X2 direction, but the following steps assume that only this one case was created. In addition, we have not considered the minimum eccentricity required by the code.

6. Results

- click the Results tab.
- display only the main model: select *Display*, *Submodel instances* and *remove all*; click *OK*..
- click Draw result and select the options in the following menu:



The program displays Moment in the X direction for each wall design unit.



- create combinations: select Combinations in the lower side menu, Define/rev... in the upper side menu and create the following combinations:

No.	Title	1:Dead	2:Live	3:RSS ,DIRECTION:X1
1	ULS	1.4	1.6	
2	EQ_+X1	1	0.2	1
3	EQ_-X1	1	0.2	-1