

This document gives a detailed summary of the new features and modifications of FEM-Design version 13. We hope you will enjoy using the program and its new tools and possibilities. We wish you success.

Strusoft, the developers

Legend



Pay attention / Note



Useful hint



Example



Clicking left mouse button



Clicking right mouse button



Clicking middle mouse button

Text Italic words marked in cyan are linked to their definition.

WHAT'S NEW IN FEM-DESIGN 13

- Improved eigenfrequency and seismic calculation
- Turkish seismic code including Performance based design
- Automatic load combination generation from load groups
- Load case, load combinations and load group export/import
- Multiple point support (group) generation
- Improved rectangle definition

Table of content

- 1. *STRUCTURE* 4
 - 1.1. Storey properties 4
 - 1.2. Stiffness reduction for dynamic analysis (material) 4
 - 1.3. Support definition 5
- 2. *LOADS* 5
 - 2.1. Load groups 5
 - 2.2. Load combinations 7
 - 2.3. Import / Export of load cases, load groups and load combinations 9
 - 2.4. Seismic load 10
- 3. *MESH* 10
 - 3.1. Settings 10
- 4. *ANALYSIS* 12
 - 4.1. Eigenfrequencies 12
 - 4.2. Seismic analysis 12
 - 4.3. Resultant 17
- 5. *PERFORMANCE BASED DESIGN* 18
 - 5.1. Model definition and calculation 18
 - 5.2. Results 20
- 6. *CAD/USER INTERFACE* 23
 - 6.1. Rectangle definition 23

1. STRUCTURE

1.1. Storey properties

Two properties are added to Storeys dialog.

- **Ground level**

For automatic wind load generation, the height of the storeys are calculated from ground level

- **Bottommost level**

This is the storey's origin height, so the storeys are starting from this level.

The 'Storeys' dialog box is shown with the following settings:

- Horizontal size of building [m] ... 50.0 X 30.0
- Ground level [m] 1.50
- Bottommost level [m] -6.00

The table below shows the storey configuration:

#	Storey	Height [m]	Level [m]
1	Storey 1	3.000	-3.000
2	Storey 2	3.000	0.000
3	Storey 3	3.000	3.000
4	Storey 4	3.000	6.000
5	Storey 5	3.000	9.000
6	Storey 6	3.000	12.000
7	Storey 7	3.000	15.000

The structural grid diagram shows levels from -6.00 to +15.00. The 'Ground level' is marked at 0.00 and the 'Bottommost level' is marked at -6.00.

1.2. Stiffness reduction for dynamic analysis (material)

For reinforced concrete structure the user has the possibility to reduce the element stiffness in order to model the cracking's effect in eigenfrequency calculation.

The 'Material' dialog box is shown with the following settings:

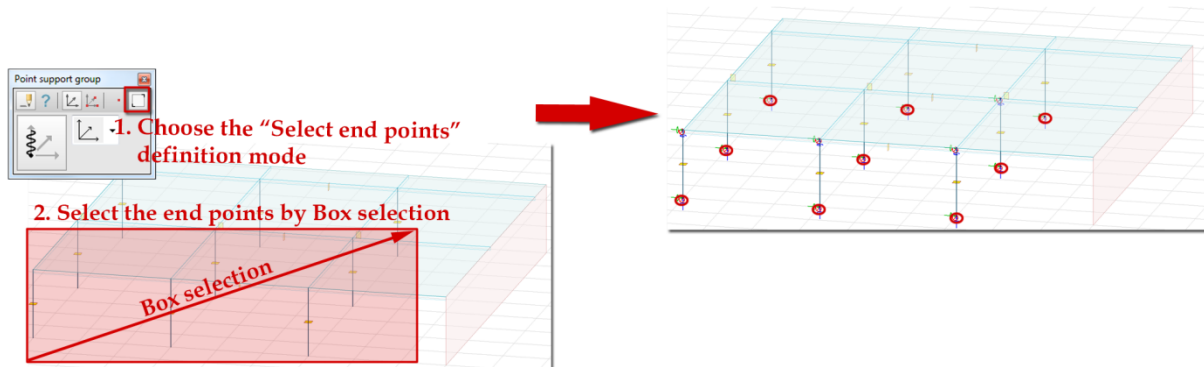
- Reduction for dynamic anal: 1.00

The 'Library' section shows the following materials:

- Concrete
 - C12/15
 - C16/20
 - C20/25
 - C25/30
 - C30/37
 - C35/45
 - C40/50
 - C45/55
 - C50/60
- Steel
 - S 235
 - S 275
 - S 355
 - S 420
 - S 450

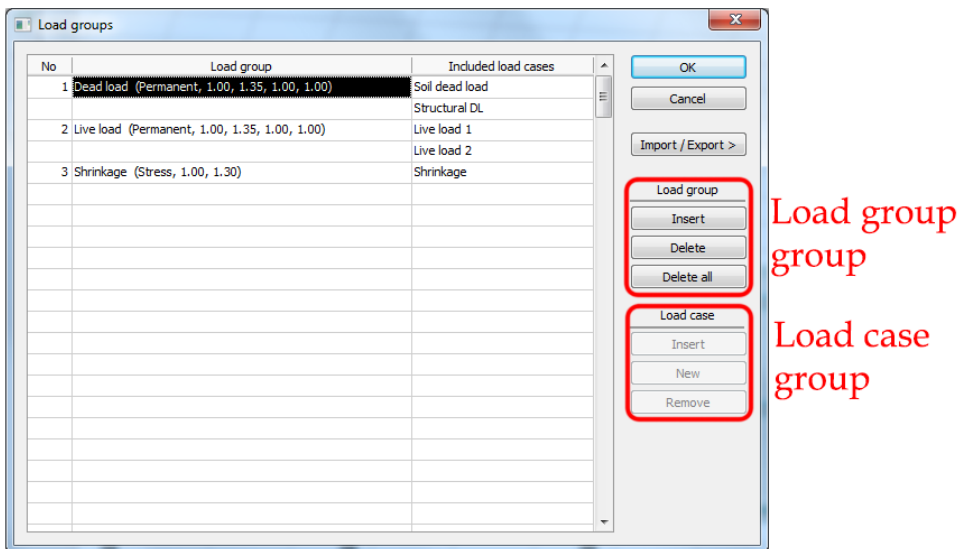
1.3. Support definition

In FEM-Design 13 a new editing function is implemented for Point Supports and Point Support Groups by which multiple supports can be defined in one step instead of placing them one-by-one.



2. LOADS

2.1. Load groups



Load group definition functions (Insert Group..., Delete group, Insert case(s)..., Delete case, Edit cell) are rearranged according to a new conception.

Under Load group group, there are three functions:

- **Insert**

With this function the user can add a new Load group. After clicking on **Insert** button, the parameters can be set in the dialog.

- **Delete**

This function deletes a Load group.

- **Delete all**

This function deletes all defined load groups.

Under Load case group, there are three functions:

- **Insert**

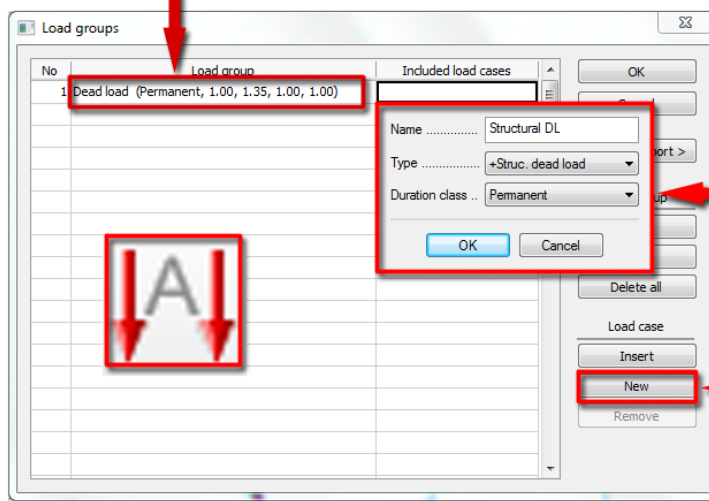
An existing load case can be inserted to a load group.

- **New**

The user has the opportunity to define load cases in Load groups dialog. After defining a load case in this dialog, the new load case will be added to Load case list (the user can see it in Load cases dialog).

1. Define Load group

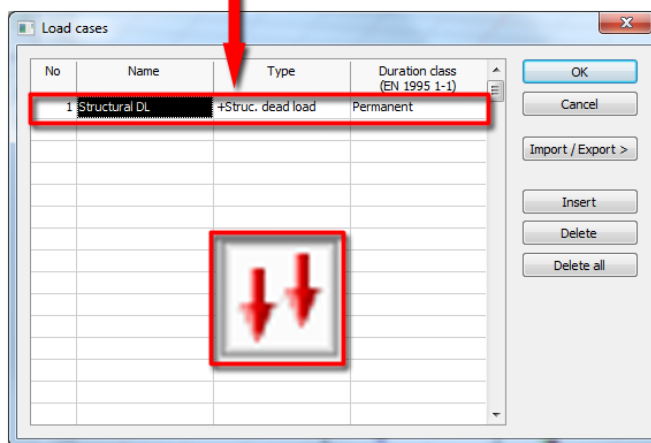
2. Activate the "Included load case" column



4. Set load case properties

3. Click on "New" button

5. The new load case defined!



- **Remove:**

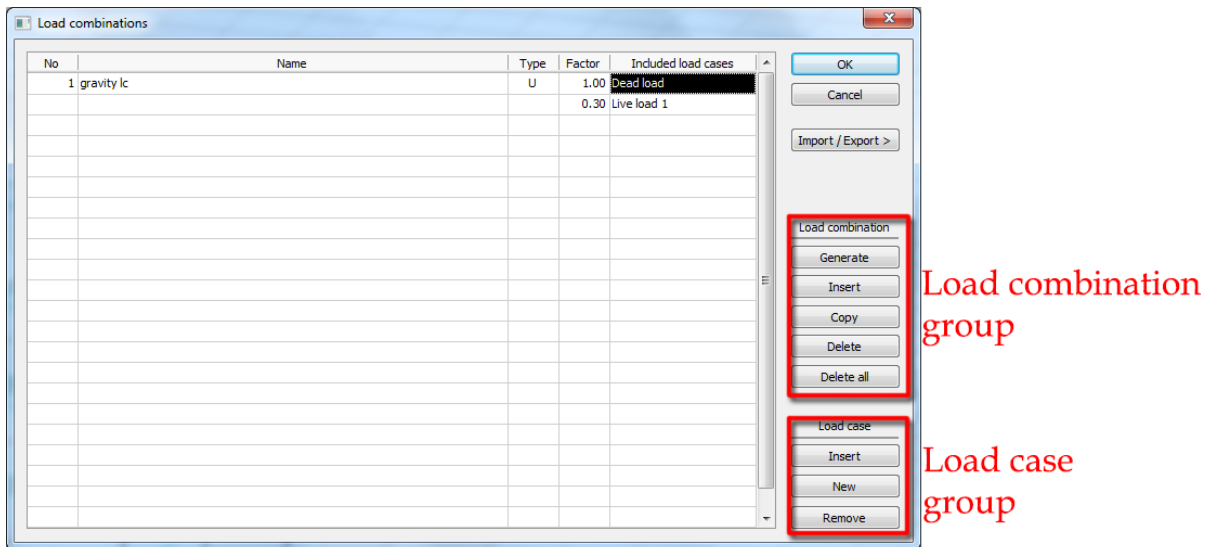
This function removes the selected load case from the current load group.



The older definition mode, by clicking in the cells is still available!

2.2. Load combinations

Load combination definition functions (Insert comb..., Delete comb., Copy comb., Insert case(s)) are also rearranged according to a new concept and a new function so called Generate is implemented



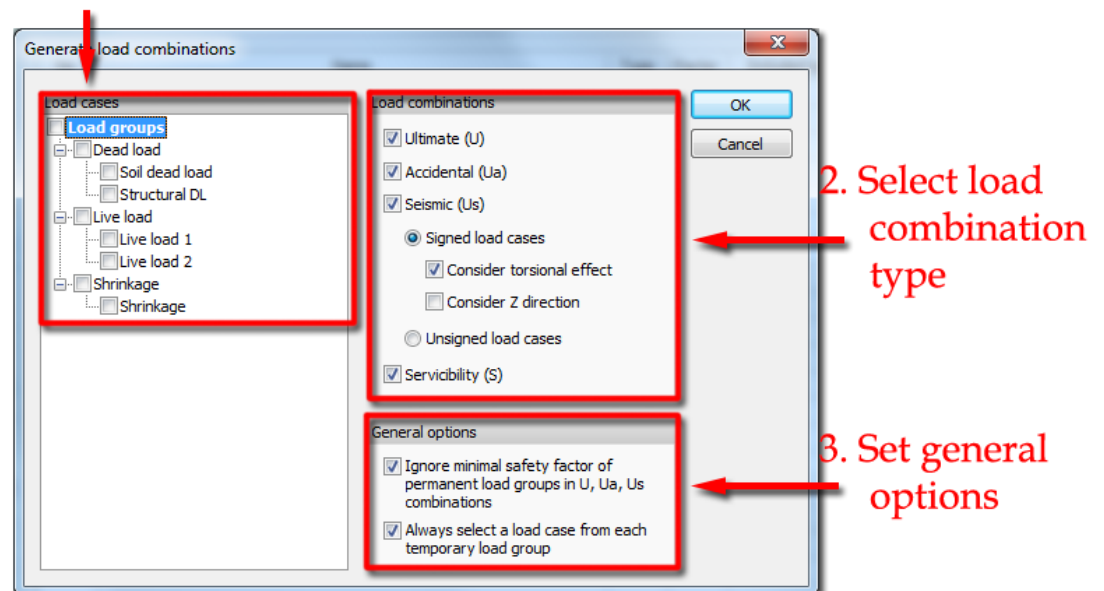
Under Load combination group, there are five functions:

- **Generate:**

FD can automatically generate Load combinations from the Load cases assigned to the Load groups, according to Eurocode 1990 – Chapter 6.4.3.

1. In Load combination dialog click Generate button.

1. Select load cases from load groups



2. Select load cases from the Load groups.
3. Select Load combination type to generate and their properties (if there is any).



The selected Load combinations will be generated if there is at least one load group of that type. E.g. if the user wants to generate Accidental load combination, but there is no Accidental Load group, or in that Load group there is no Load case, the program will not create any accidental Load combination.

4. Set the General options parameters.

- **Insert**

With this function the user can add a new Load combination. After clicking on **Insert** button, the parameters can be set in the dialog.

- **Copy**

With this function the user can copy an existing combination with load cases and factors.

- **Delete**

The user can delete a selected load combination.

- **Delete all**

All the load cases can be deleted with Delete all function.

Under Load case group, there are three functions:

- **Insert:**

An existing load case can be inserted to a load group with **Insert** function.

- **New:**

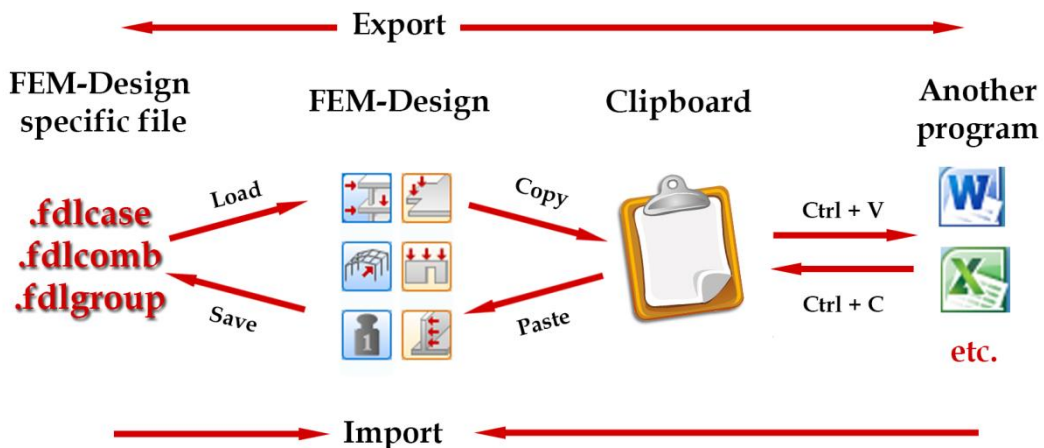
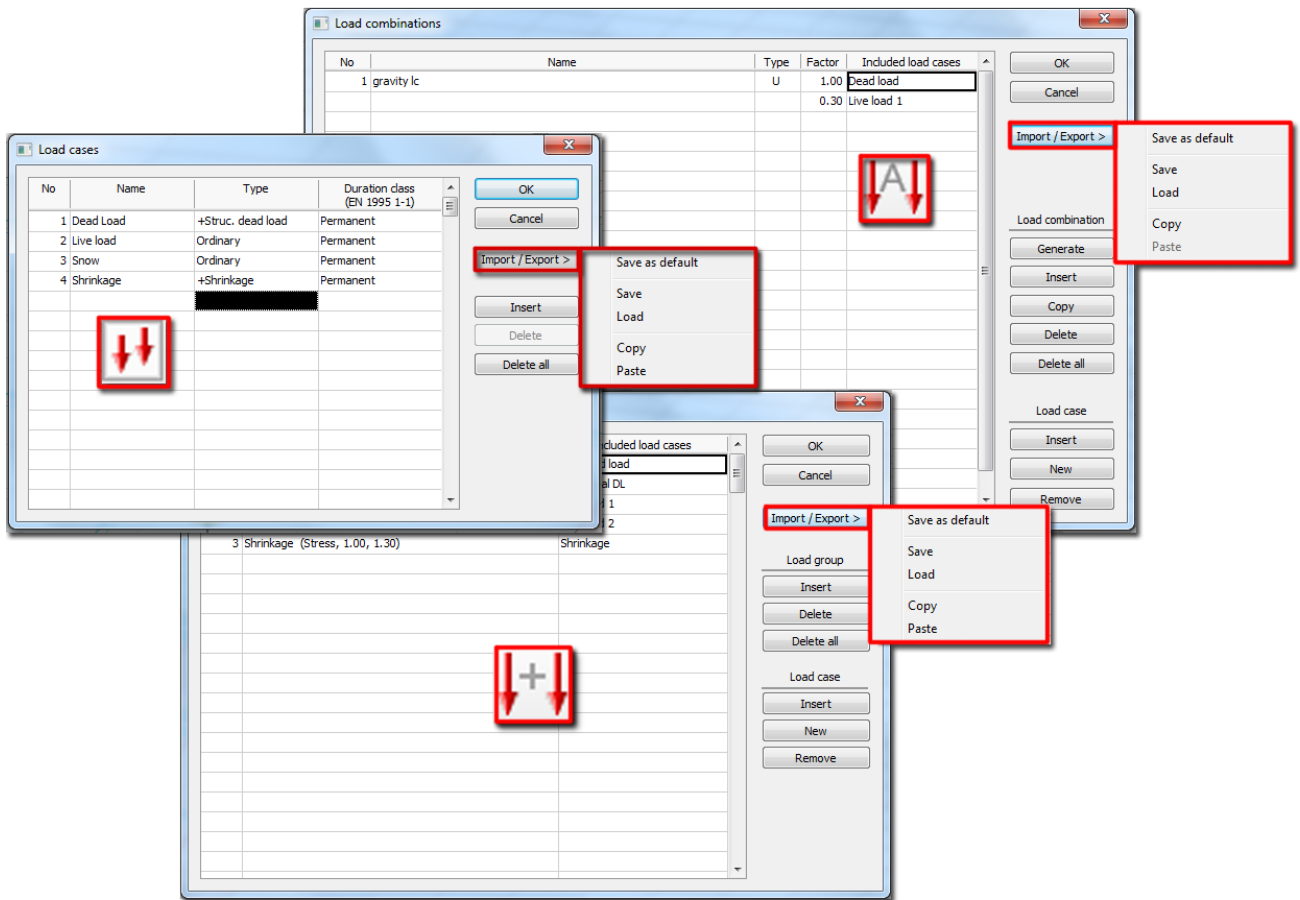
The user has the opportunity to define load cases in Load combinations dialog. After defining a load case in this dialog, the new load case will be added to Load case list (the user can see in Load cases dialog).

- **Remove:**

This function removes the selected load case from the current load group.

2.3. Import/Export of load cases, load groups and load combinations

Load cases, Load groups and Load combination can be imported or exported in native FEM-Design or in text format.



- **Save as default**
FD saves the load cases / groups / combinations and they will be loaded at every new model.
- **Save**
Load cases can be saved to a special .ldcase / .ldgroup / .ldcomb type file.
- **Load**

The user has the possibility to load previously saved .ldcase / .ldgroup / .ldcomb file.

- **Copy**

Load cases / groups / combinations can be copied to Clipboard with Copy command to use them in another program (e.g. MS Excel)

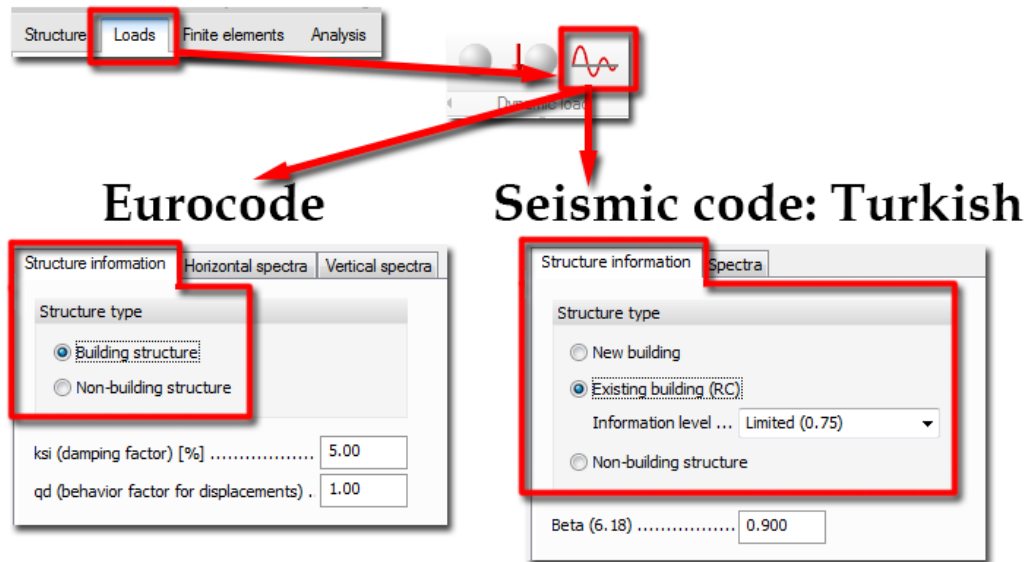
- **Paste**

Load cases / groups / combinations can be imported through Clipboard with Paste command.

2.4. Seismic load

In order to define seismic load in Seismic load dialog the user has to select the structure type under Structure information tab / Structure type:

- Eurocode: **Building structure** and **Non-building structure**.
- Turkish seismic code: **New building**, **Existing building**, **Non-building structure**.



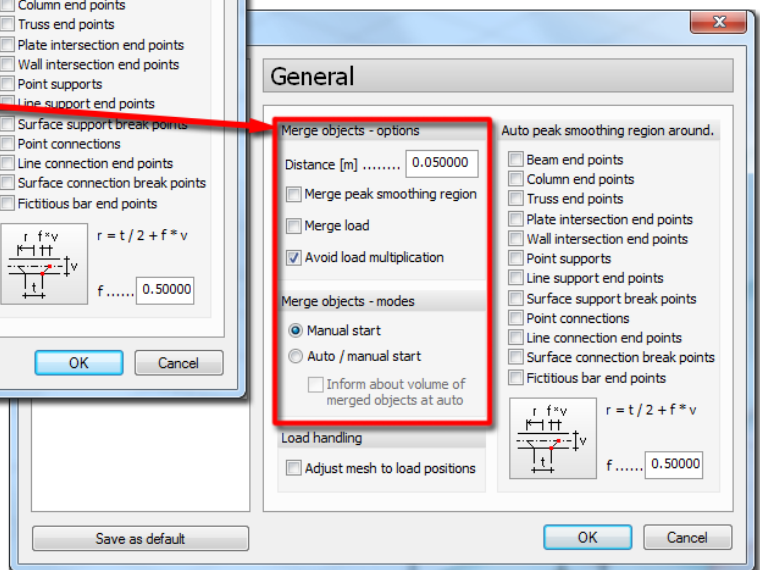
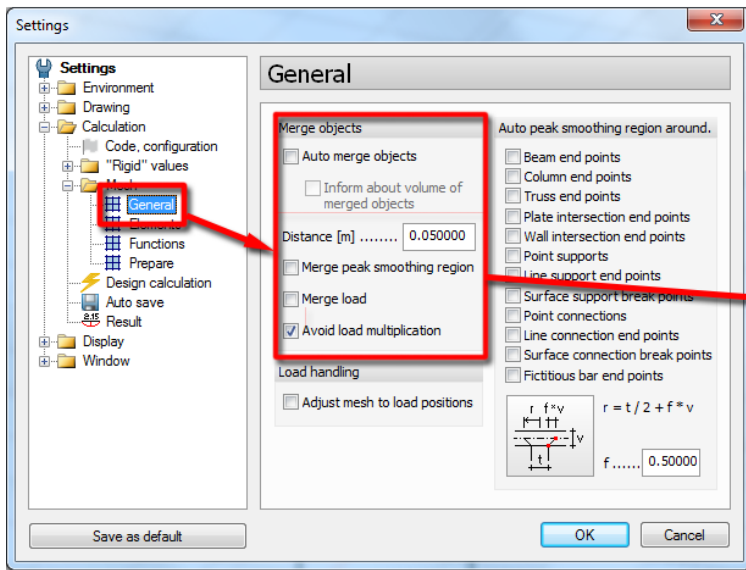
If **Building structure** is chosen, storeys have to be defined in order to get Storey seismic sensitivity (P-delta (θ)) coefficient and Storey shear forces.

If **Non-building structure** is chosen, these results are not available.

3. MESH

3.1. Settings

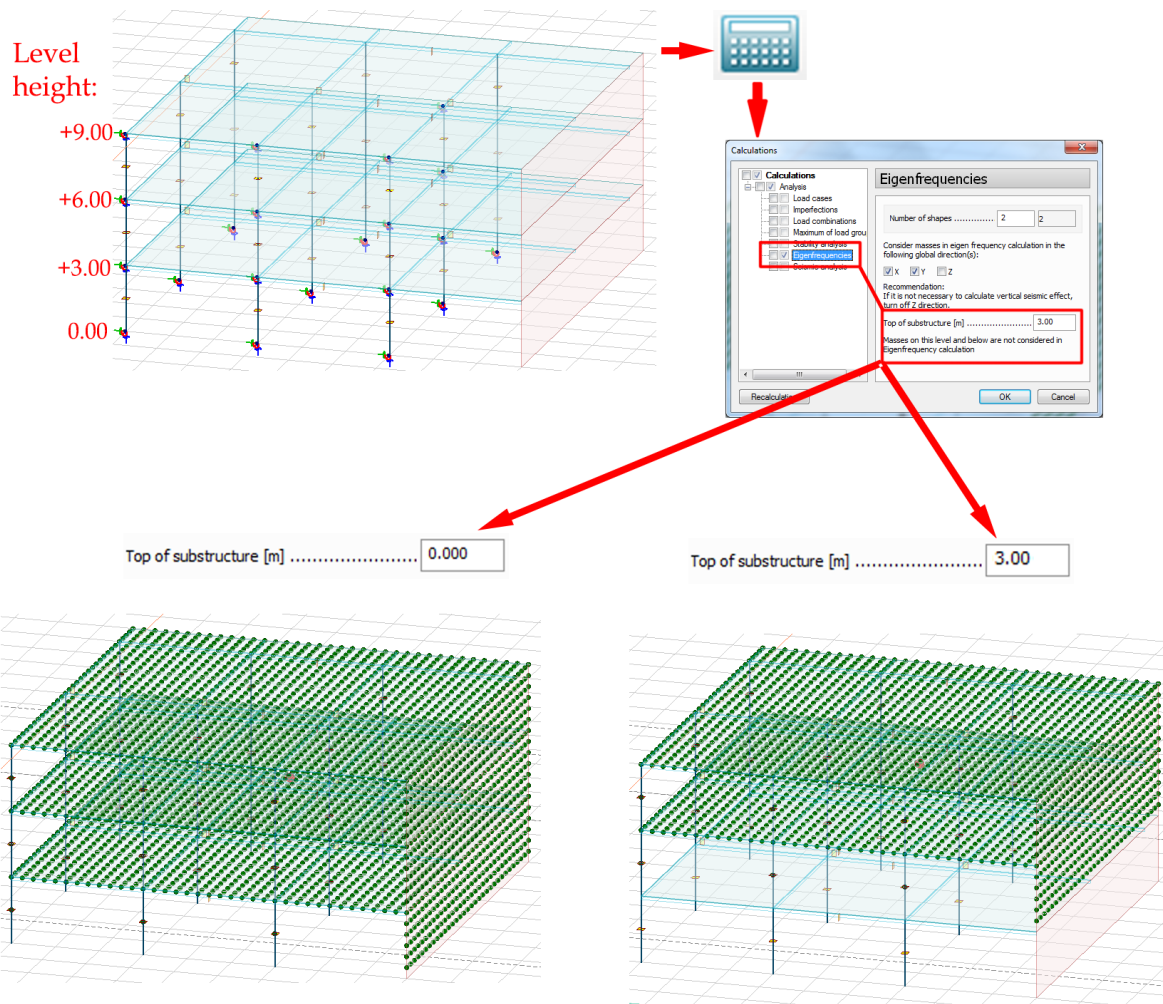
In the Settings/Calculation/Mesh/General dialog some properties are rearranged to make clearer that the options and modes of "Merge object" function are independent from each other.



4. ANALYSIS

4.1. Eigenfrequencies

In Calculation / Eigenfrequencies dialog the user can set the level of top of the substructure. The masses will be neglected at and under this level.



In the mass centre of the masses the total mass is displayed with red circle.



To get the whole structure's mass centre position set the level of the Top of the substructure a bit under the structure.



This function is useful to neglect the foundation mass in the eigenfrequency calculation so the total mass contribution in Modal analysis can reach $\geq 90\%$.

4.2. Seismic analysis

From now the seismic analysis results can be Signed results. This is relevant in Modal analysis (in Turkish seismic code in Mode superposition) where the Equivalent seismic loads are calculated from the Vibration shapes and summarized with SRSS or CQC combination rule, where the sign of the forces are lost. In order to get signed results, the user has to select dominant shapes in the two

perpendicular directions (x' and y') and check the Signed results combo box, so the seismic forces will have the signs of the dominant shape's sign in x' and y' direction.

Eurocode

No	T[s]	mx[%]	my[%]	mz[%]
6	0.021	0.9	0.0	0.0
7	0.020	37.7	0.6	0.0
8	0.019	0.0	0.0	0.0
9	0.017	0.0	0.0	0.0
10	0.017	0.0	0.0	0.0
11	0.016	0.0	0.0	0.0
12	0.016	0.0	1.8	0.0

Mode shapes coloring:
Selected
 Selected and dominant

Newmark summation

$$E = \sqrt{B_x^2 + B_y^2 + B_z^2}$$

$$E = 0.3B_x + 0.3B_y + 0.3B_z$$

Signed results

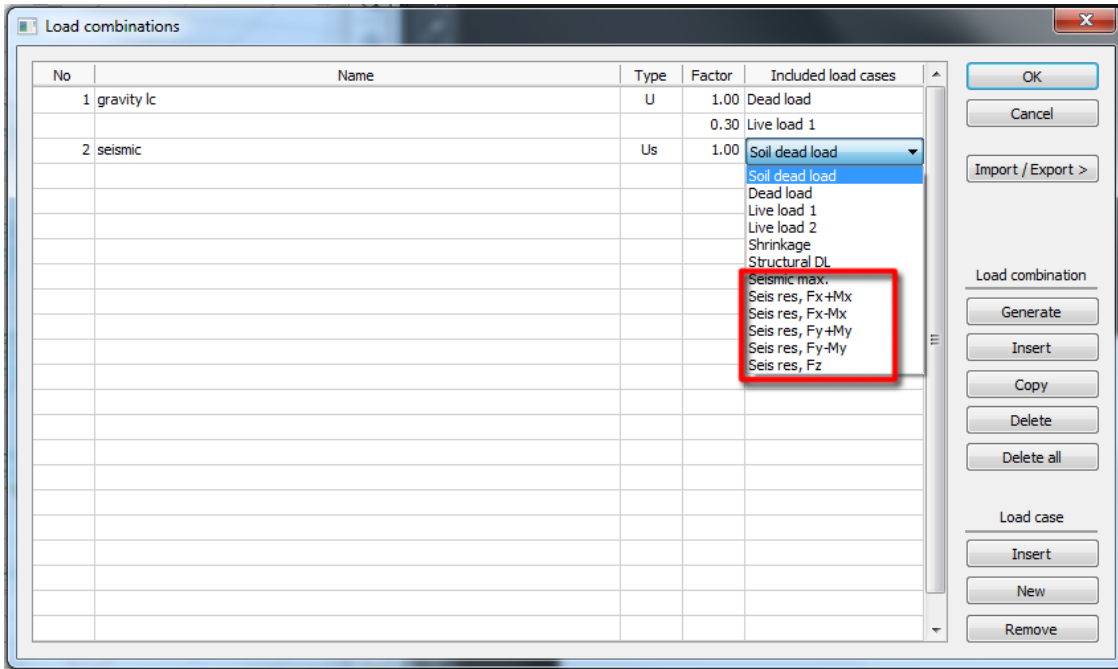
Seismic code: Turkish

No	T[s]	mx[%]	my[%]
1	0.477	0.0	64.4
2	0.435	80.4	0.0
3	0.306	0.0	19.3
4	0.204	0.0	19.3
5	0.197	19.0	0.0
6	0.150	0.0	1.7
7	0.018	0.0	0.0

Mode shapes coloring:
Selected
 Selected and dominant

Signed results

Some changes are made in the way of adding seismic loads to Load combinations and Load groups. According to Eurocode 1998-1-1, 4.3.2 Chapter, the masses are shifted to + 5% and - 5% of the building size perpendicular to earthquake direction, so the seismic analysis results are separated to 5 cases, Fx + Mx, Fx - Mx, Fy + My, Fy - My, Fz. In Load combinations dialog these 5 cases and Seismic max. load cases can be added to Load combinations.

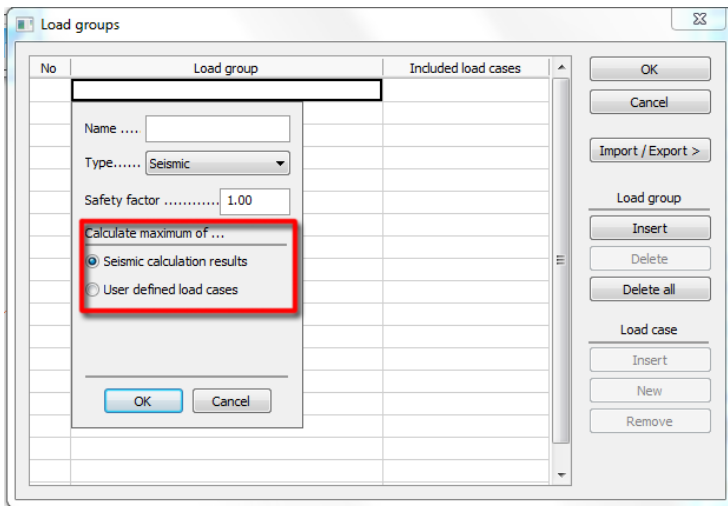


Fx + Mx means that the earthquake force is parallel to x' direction, and the masses are shifted to result positive Mz' moments.

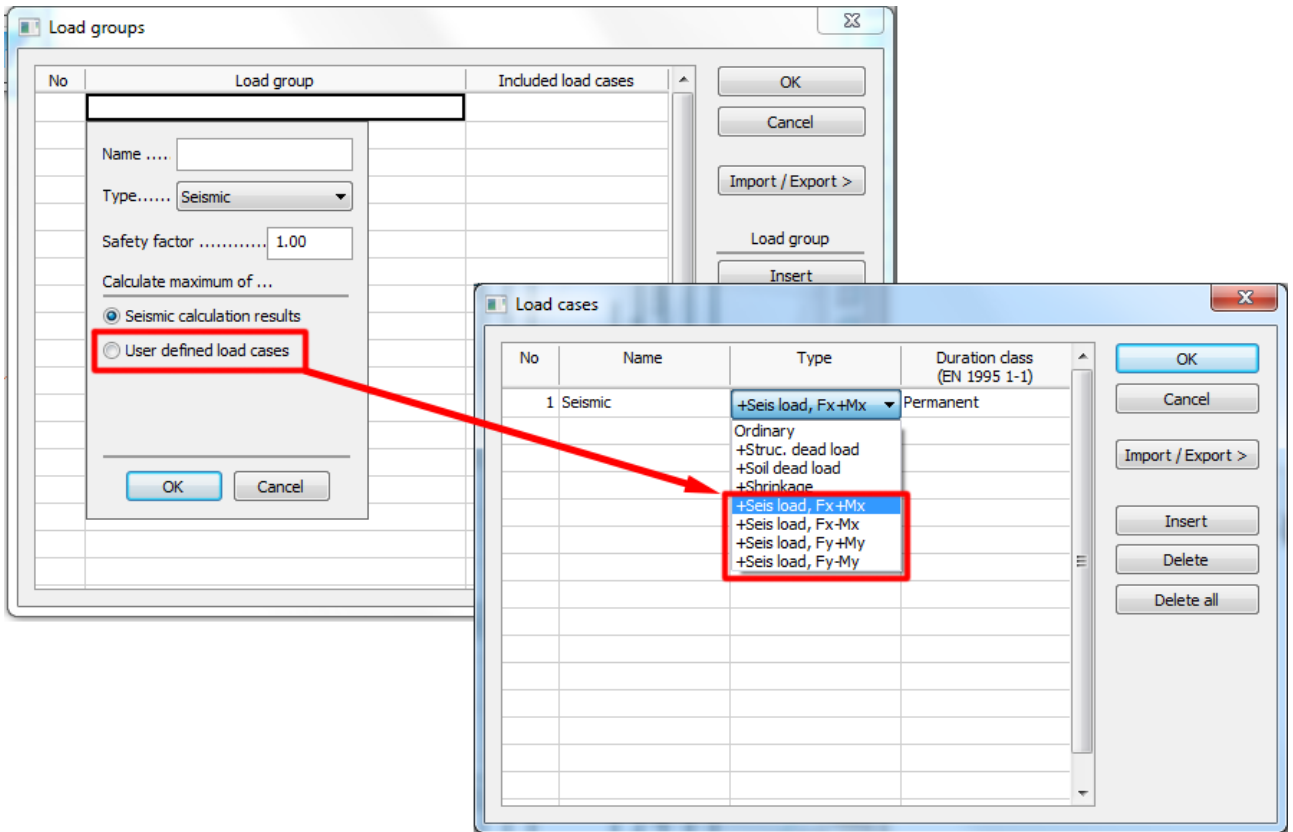


To add **-Fx -Mx, -Fy-My** or **-Fz** earthquake effects to a load combination, the Factor of the Load case (**Fx + Mx, Fy + My, Fz**) has to be negative.

Concept of adding seismic effects to load groups has also changed.



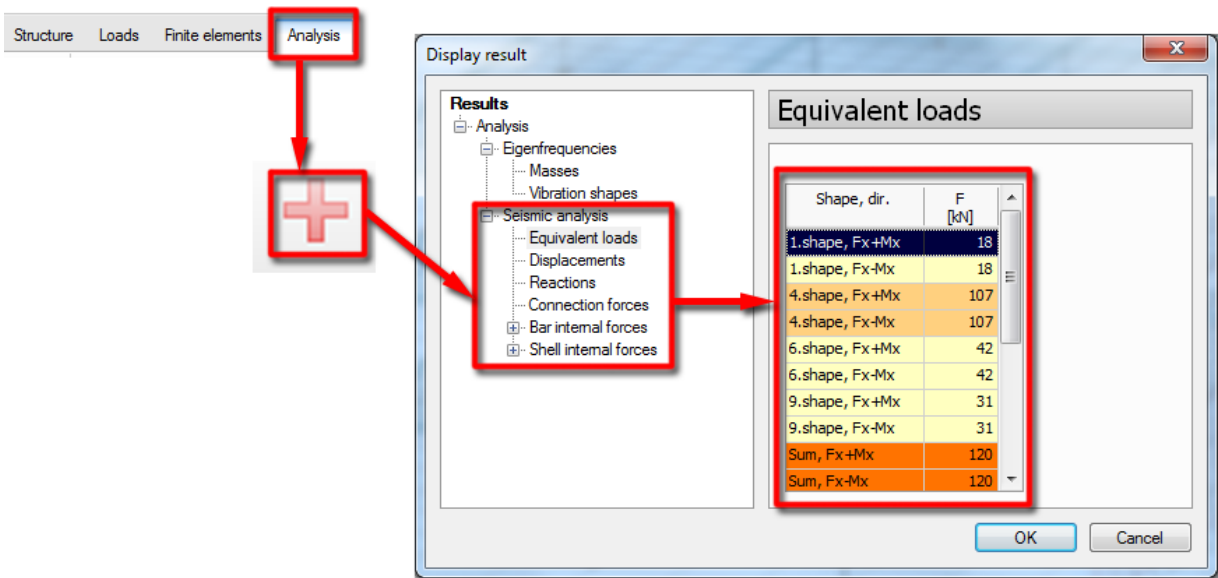
- In case of *Unsigned* seismic analysis:
 - The seismic effects are combined to the static load cases like in the previous versions (all components of the seismic results added to the components of static results with the sign of static component)
- In case of *Signed* seismic analysis:
 - If *Seismic calculation results* option is selected, the seismic results are added to the static results component by component
 - If *User defined load cases* option is selected, the program combines the static "+Seis load..." type load cases with the static loads.

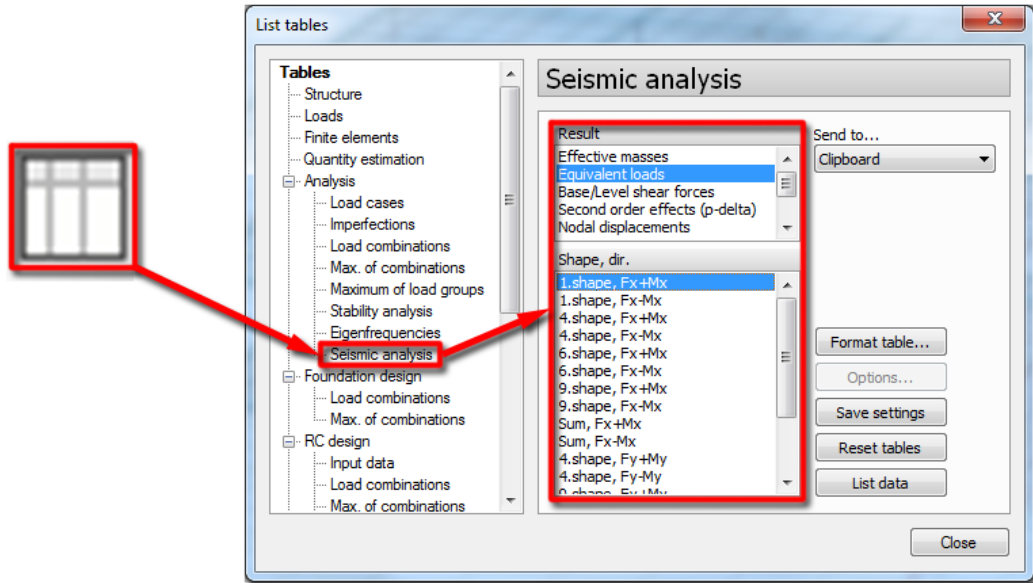


⚠ Seismic results will be added to **Seismic max.** load case, if the seismic loads are Unsinged. The seismic results will be added to the static results with their sign.

⚠ In case of some special structures signed results may be incorrect if the user doesn't choose the dominant shape properly. We suggest using **Seismic max.** results in this case.

Seismic analysis results (Equivalent loads, Displacements, Reactions, Connection forces, Bar internal forces, Shell internal forces) are also rearranged by the 5 cases of earthquake effect.





In case of Eurocode code, if the Structure type is Building (Loads/Seismic load):

- Base/Level shear forces (for levels)
- Second order effects (P-delta) (for storeys)

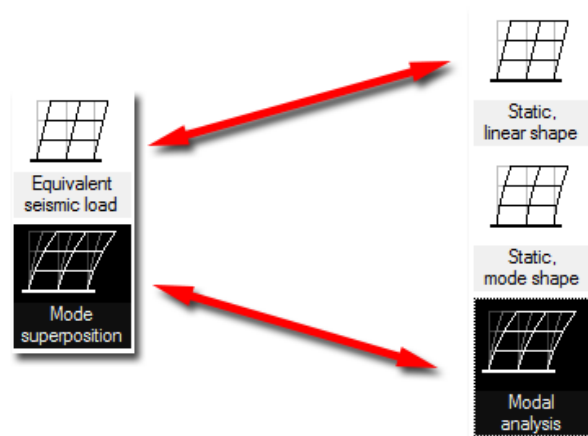
In case of Seismic code: Turkish, if the Structure type is Building (Loads/Seismic load):

- Torsional Irregularity Amplifying factor (for storeys)
- Base/Level shear forces (for levels)
- Second order effects (P-delta) (for storeys)

Two analysis type are available in Turkish seismic code, Equivalent seismic load similar to Lateral force method of Eurocode and Mode superposition similar to Modal analysis of Eurocode method.

Turkish

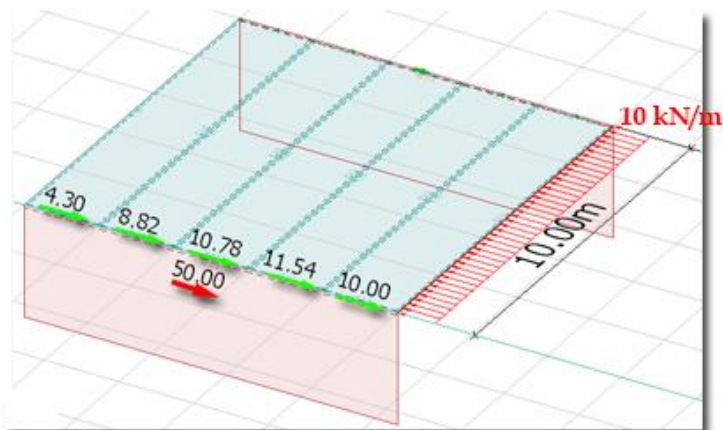
Eurocode



4.3. Resultant

In some special cases, the resultants of the line-line connection were not calculated correctly. From now FD calculates the resultants in an improved way. The following picture shows the resultants and their sum in case of the profiled panels connected to a wall.

FEM-Design 12



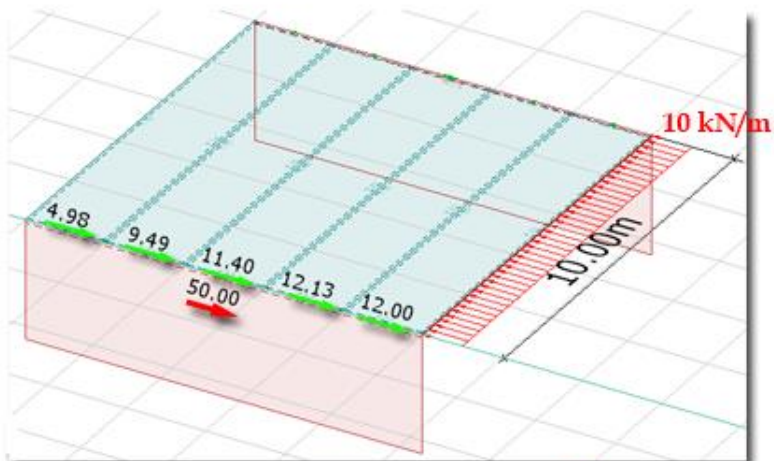
$$4.30+8.82+10.78+11.54+10.00 = 45.44 \neq 50 \quad \text{⚡}$$

Resultant of the profiled panel edges

Sum of resultants

Resultant of the wall edge

FEM-Design 13



$$4.98+9.49+11.40+12.13+12.00 = 50 = 50 \quad \text{✓}$$

Resultant of the profiled panel edges

Sum of resultants

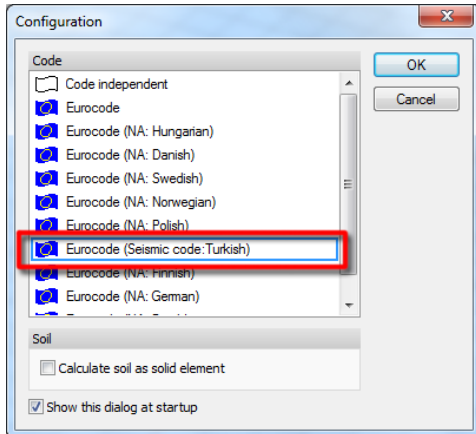
Resultant of the wall edge

5. PERFORMANCE BASED DESIGN

5.1. Model definition and calculation

Performance Based Design is implemented in FD 13 according to Turkish code. This way the performance level of existing RC buildings can be calculated from linear analysis. To calculate Performance level of an existing reinforced concrete building, follow these steps:

1. Set the code to "Eurocode (Seismic code: Turkish)"

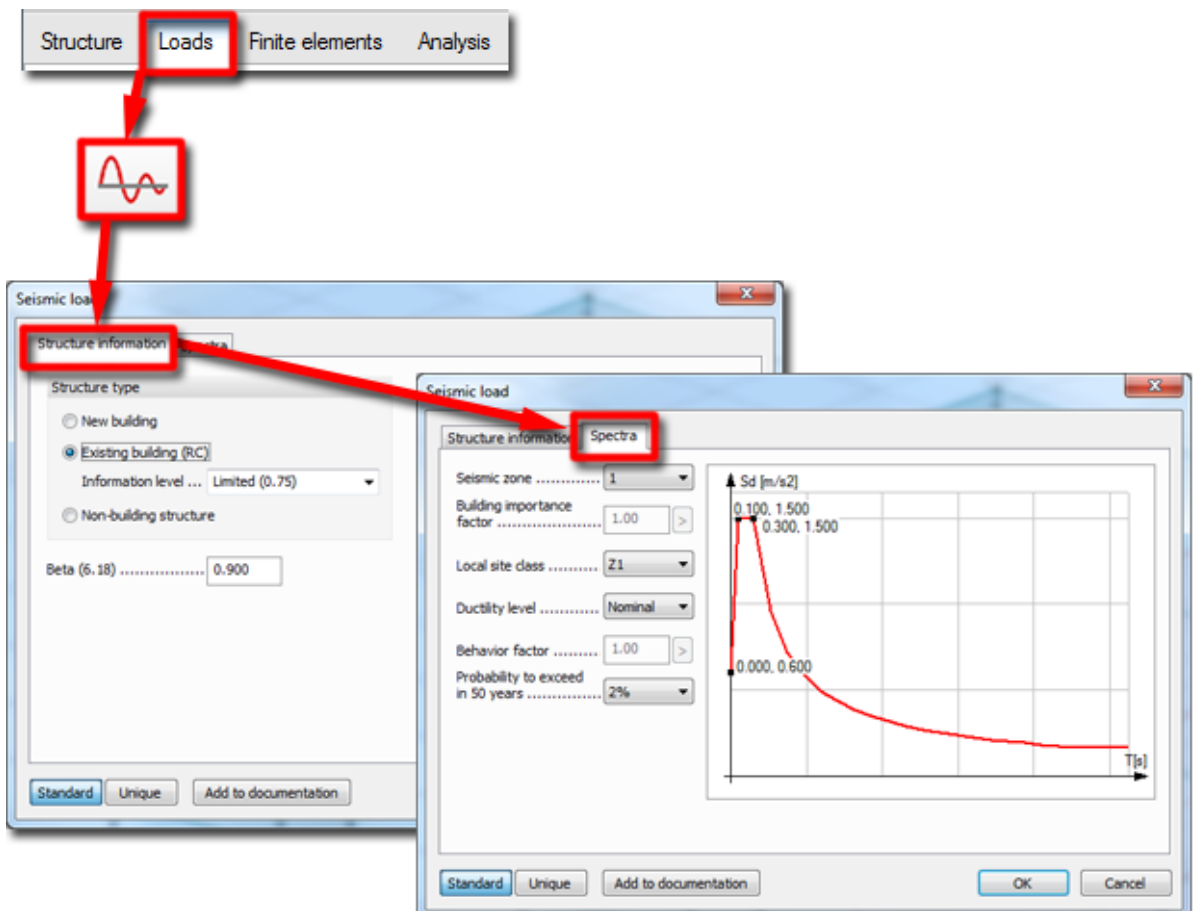


2. Build the model in Structure tab.



The columns and walls has to be defined from storey to storey.

3. Add the loads in Loads tab



- a. Define Gravity load combination.
- b. Define masses from loads in Load case – mass conversion dialog.
- c. Define the Seismic Load properties:
 - i. Set the Structure type to Existing building, and also the Information level in Structure information tab.
 - ii. Define the design spectra in Spectra tab.

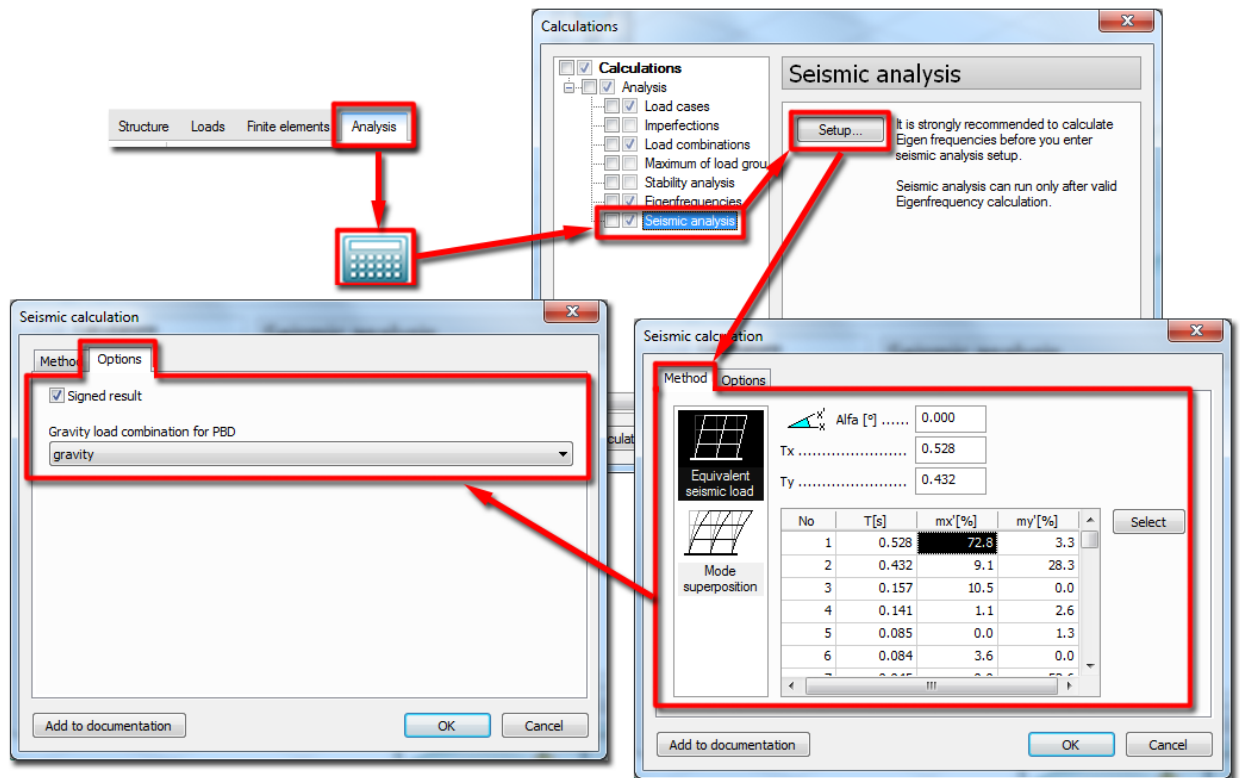


There are Standard and Unique spectra. Standard spectra is calculated according to the current seismic code. In order to define non-standard spectra the use the Unique option.

4. Run analysis in Analysis tab

- a. Calculate Load combinations in Calculation dialog.
- b. Calculate Eigenfrequencies in Calculation dialog.
- c. Calculate Seismic analysis in Calculation dialog.

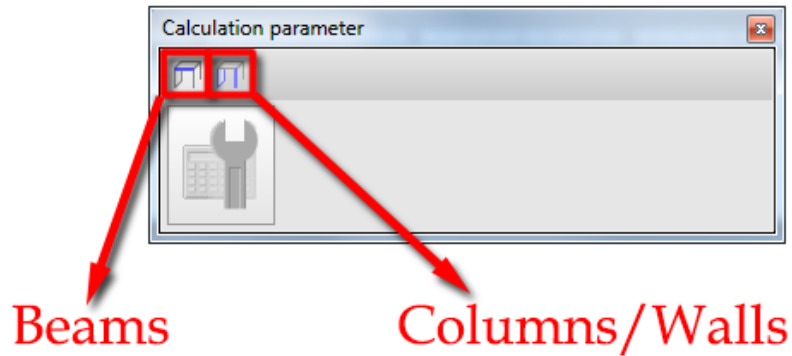
Choose between two analysis type, Equivalent seismic load (similar to Lateral force method of Eurocode) and Mode superposition (similar to Modal analysis of Eurocode) method.



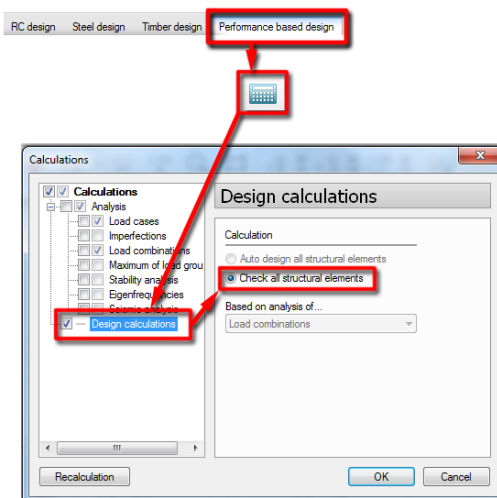
5. Define reinforcement for RC elements in RC Design tab

6. In Performance based design tab

- a. Set the Calculation parameters for beams, columns and walls:



- Beams: both ends are **Confined** and/or **Shear strengthened** and/or **Secondary beams**
 - Columns / Walls: both ends are **Confined** and/or **Shear strengthened**
- b. Start the Performance based design calculation by selecting the Design calculations under Calculation dialog.



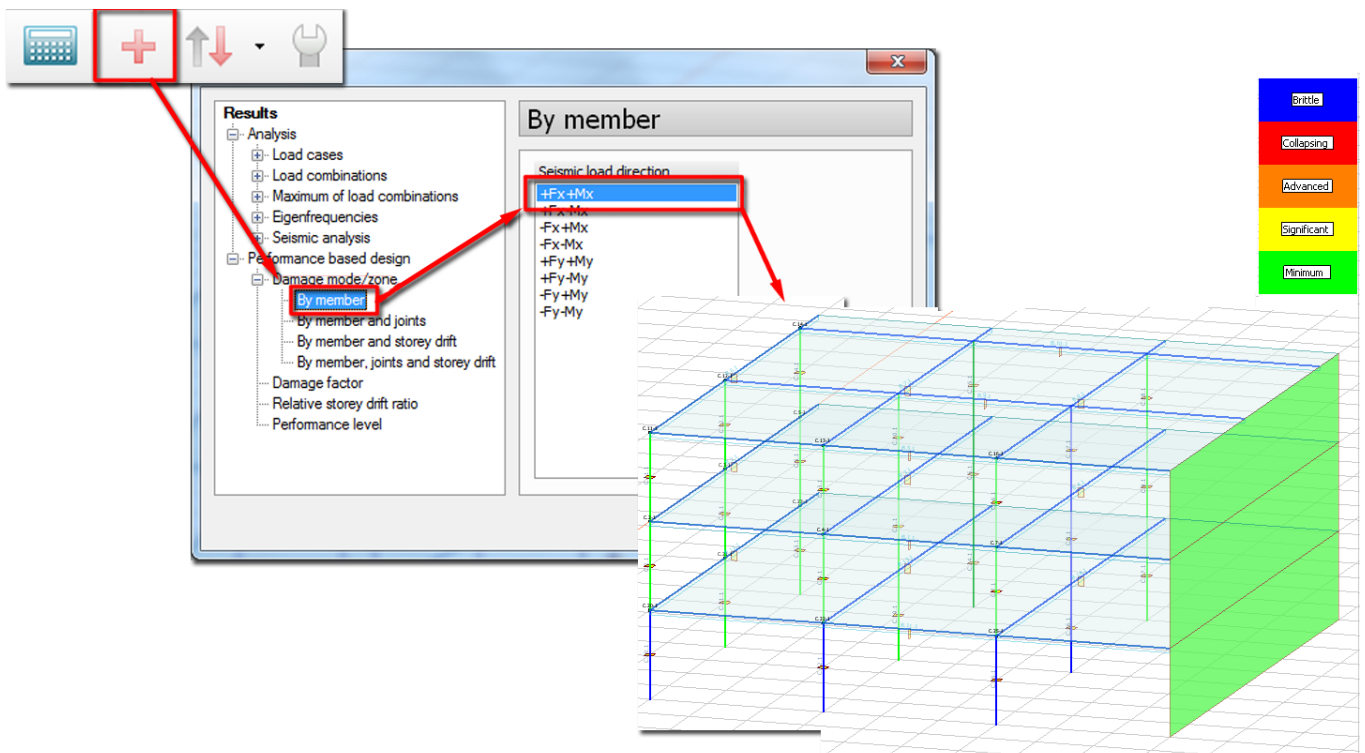
5.2. Results

For Performance based design the following results are available to display and list:

Results for display:

- Damage mode/ zone:
 - By member
 - By member and joints
 - By member and storey drift
 - By all

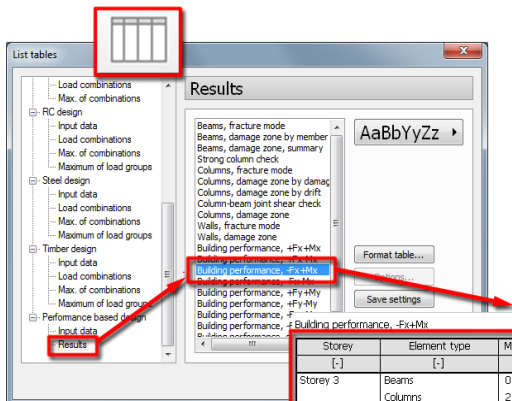
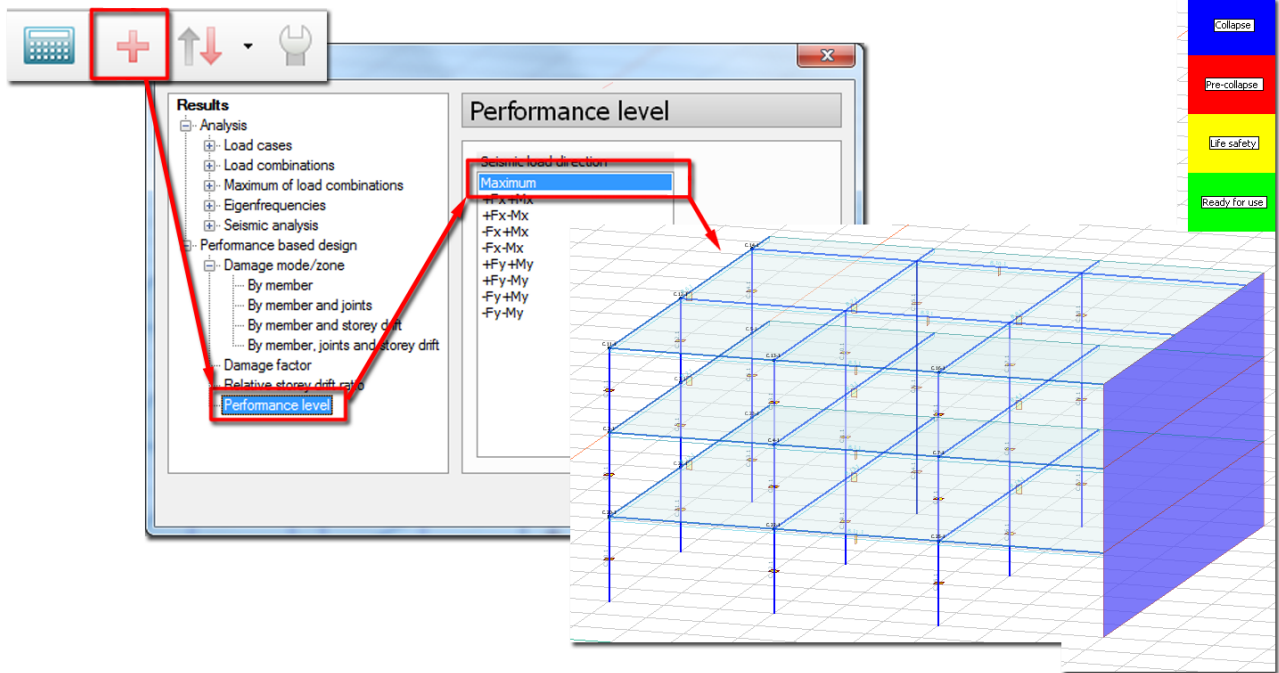
- Damage factor
- Relative storey drift ratio
- Performance level



Results to list:

- Input data
 - Beam longitudinal reinforcement
 - Column longitudinal reinforcement
 - Wall longitudinal reinforcement
 - Beam stirrup reinforcement
 - Column stirrup reinforcement
 - Wall stirrup reinforcement
- Results
 - Beams, fracture mode
 - Beams, damage zone by member
 - Beams, damage zone, summary
 - Strong column check
 - Columns, fracture mode
 - Columns, damage zone by damage factor
 - Columns, damage zone by drift
 - Column-beam joint shear check
 - Columns, damage zone
 - Wall fracture mode

- Walls, damage zone
- Building performance (Fx +Mx, Fx - Mx, -Fx + Mx, -Fx - Mx, Fy+My, Fy-My, -Fy+My, -Fy-My, summary)



Storey	Element type	Minimum damage zone	Significant damage zone	Advanced damage zone	Collapsing damage zone	Brittle damage zone	Performance level
(-)	(-)	(-)	(-)	(-)	(-)	(-)	(-)
Storey 3	Beams	0 (0%)	0 (0%)	0 (0%)	0 (0%)	2 (100%)	Collapse
	Columns	2 (40%)	0 (0%)	0 (0%)	0 (0%)	3 (60%)	
	Walls	1 (100%)	0 (0%)	0 (0%)	0 (0%)	0 (0%)	
	Shear force of columns	67.40 kN (66%)	0.00 kN (0%)	0.00 kN (0%)	0.00 kN (0%)	34.93 kN (34%)	
Storey 2	Beams	0 (0%)	0 (0%)	0 (0%)	0 (0%)	2 (100%)	Collapse
	Columns	3 (60%)	0 (0%)	0 (0%)	0 (0%)	2 (40%)	
	Walls	1 (100%)	0 (0%)	0 (0%)	0 (0%)	0 (0%)	
	Shear force of columns	63.17 kN (69%)	0.00 kN (0%)	0.00 kN (0%)	0.00 kN (0%)	9.84 kN (11%)	
Storey 1	Beams	0 (0%)	0 (0%)	0 (0%)	0 (0%)	2 (100%)	Collapse
	Columns	2 (40%)	0 (0%)	0 (0%)	0 (0%)	3 (60%)	
	Walls	1 (100%)	0 (0%)	0 (0%)	0 (0%)	0 (0%)	
	Shear force of columns	37.97 kN (38%)	0.00 kN (0%)	0.00 kN (0%)	0.00 kN (0%)	61.66 kN (62%)	
Foundation	Beams	0 (0%)	0 (0%)	0 (0%)	0 (0%)	0 (0%)	Collapse
	Columns	0 (0%)	0 (0%)	0 (0%)	0 (0%)	0 (0%)	
	Walls	0 (0%)	0 (0%)	0 (0%)	0 (0%)	0 (0%)	
	Shear force of columns	0.00 kN (0%)	0.00 kN (0%)	0.00 kN (0%)	0.00 kN (0%)	0.00 kN (0%)	
Whole building	Storey evaluation						Ready for Use, pro... Collapse

6. CAD/USER INTERFACE

6.1. Rectangle definition

Rectangle definition is improved to define arbitrary positioned rectangle regions (e.g. plate). The following picture shows how to define two perpendicular leaning rectangles:

x: Left mouse button click

x: Move the cursor to there

