AXISVM

ADVANCED STEP BY STEP TUTORIAL

Edited by: Inter-CAD Kft.

Intentionally blank page

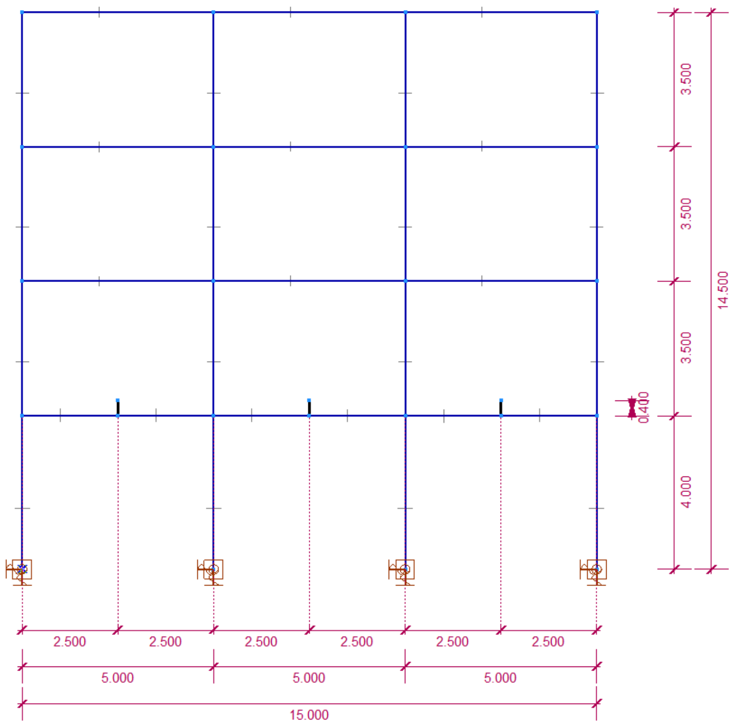
TARTALOM

Page left intentionally blank

1. DYNAMIC ANALYSIS OF A FOUR-STOREY STEEL PLANE FRAME

Objective

The objective of this example is to determine the dynamic response of the following four-storey steel plane frame considering various dynamic loads.



In this example the capabilities of the AxisVM Dynamic (DYN) module are introduced and a complete time history analysis is presented. Different types of dynamic loads are applied: two types of support acceleration (seismic and sine wave acceleration) are used, induced vibration of a rotating (industrial) machine is modelled by dynamic nodal loads, and the effect of short-term shock load is examined.

Linear elastic behaviour is assumed, however in the analysis the 2nd order P-Δ effect will be considered (geometric nonlinearity).

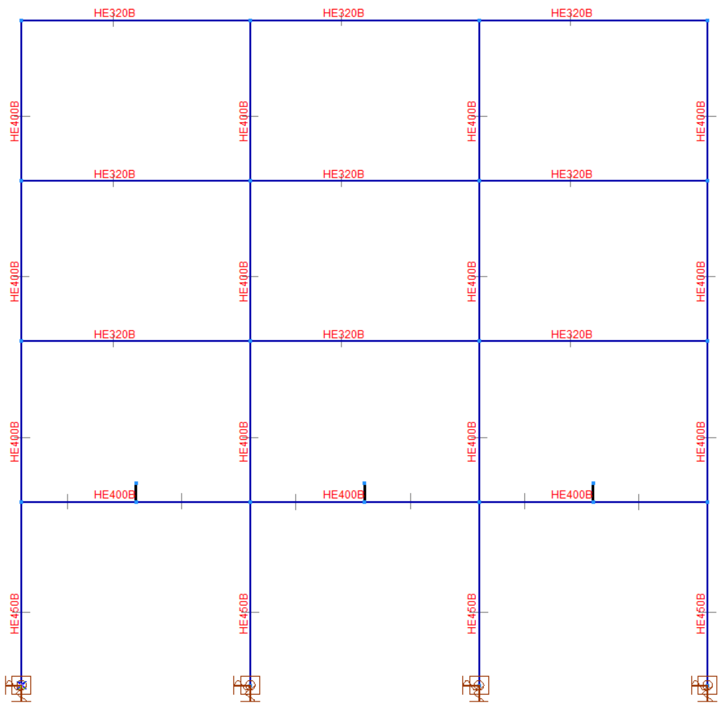
Suggestions and useful tips are provided for the dynamic analysis using AxisVM. The possibilities for evaluating the results are briefly described after calculations.

Description of the finite element model

In the task, the creation of the finite element model will not be described. However, the model can be easily constructed by applying the knowledge of the previous tutorial examples. The starting file (**Four_storey_steel_frame_dynamic_analysis_0.axs**) is available for download and contains the followings

The complete geometry and finite element model of the four-storey steel plane frame. The frame is constructed in the ***X-Z*** work plane.

The columns on the ground floor has a cross-section of **HE450 B**, while the others have a shape of **HE400 B**. The cross-section of the beams above the ground floor is **HE400 B**, the others are **HE320 B**.
The cross-sections of the frame are shown in the next figure:



The frame is analysed according to the ***Eurocode*** standard. The material of the sections is **S275**, and the material is assumed to be linear.

The frame is also assumed to be supported laterally, so only the plane (***x-z***) behaviour is tested. The nodal degrees of freedom are set according to this assumption.

The supports of the ground floor columns are fixed (***Rx=Ry=Rz=1E+7 kN/m***, ***Rxx=Ryy=Rzz=1E+7 kNm/rad***).

The static loads considered in the dynamic analysis have been created and are separated in different load cases (***self-weight***, ***distributed load***, ***concentrated load***) as follows:

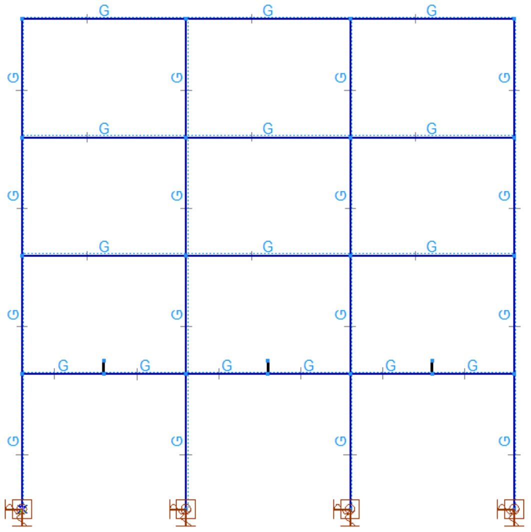
- The structure is loaded by its own ***self-weight***, which is calculated automatically by the program considering the set material and different cross-sections.

- On the lower two beams **70 kN/m** ***distributed loads*** are applied, while on the others **40 kN/m** loads are applied.

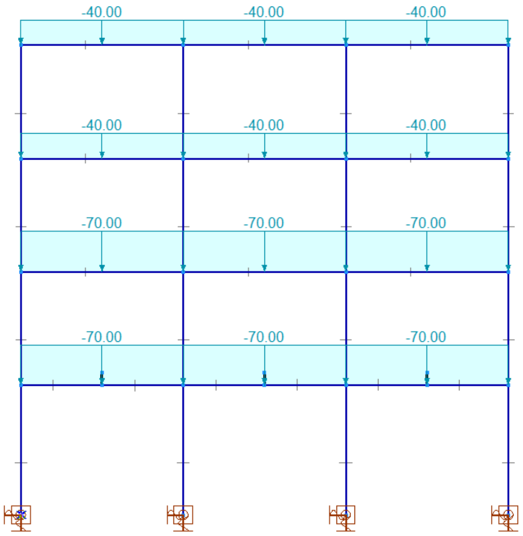
- In the middle of the first-floor beams, ***concentrated loads*** were placed (in three positions, each with a force of **20 kN**) on a rigid body element at a height of 0.4 m measured from the axis of the beams. These represent the weight of the rotating industrial machines in the building. These nodes are connected to the beams by rigid body finite elements.

- In addition, each corner frame node is loaded by ***concentrated load***s (**70** and **100 kN**) according to the arrangement shown in the following figure:

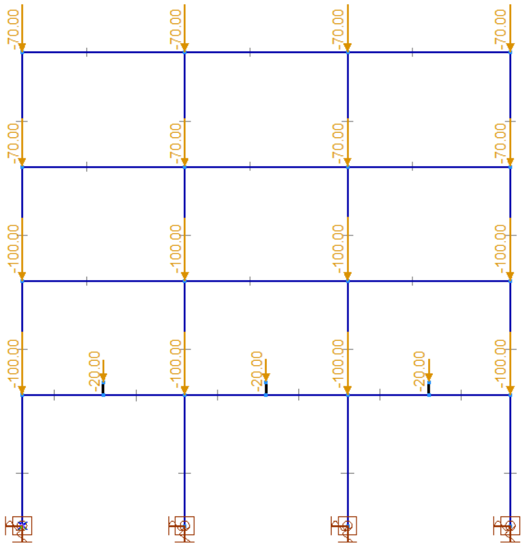
self-weight load case:



distributed load:



concentrated load:

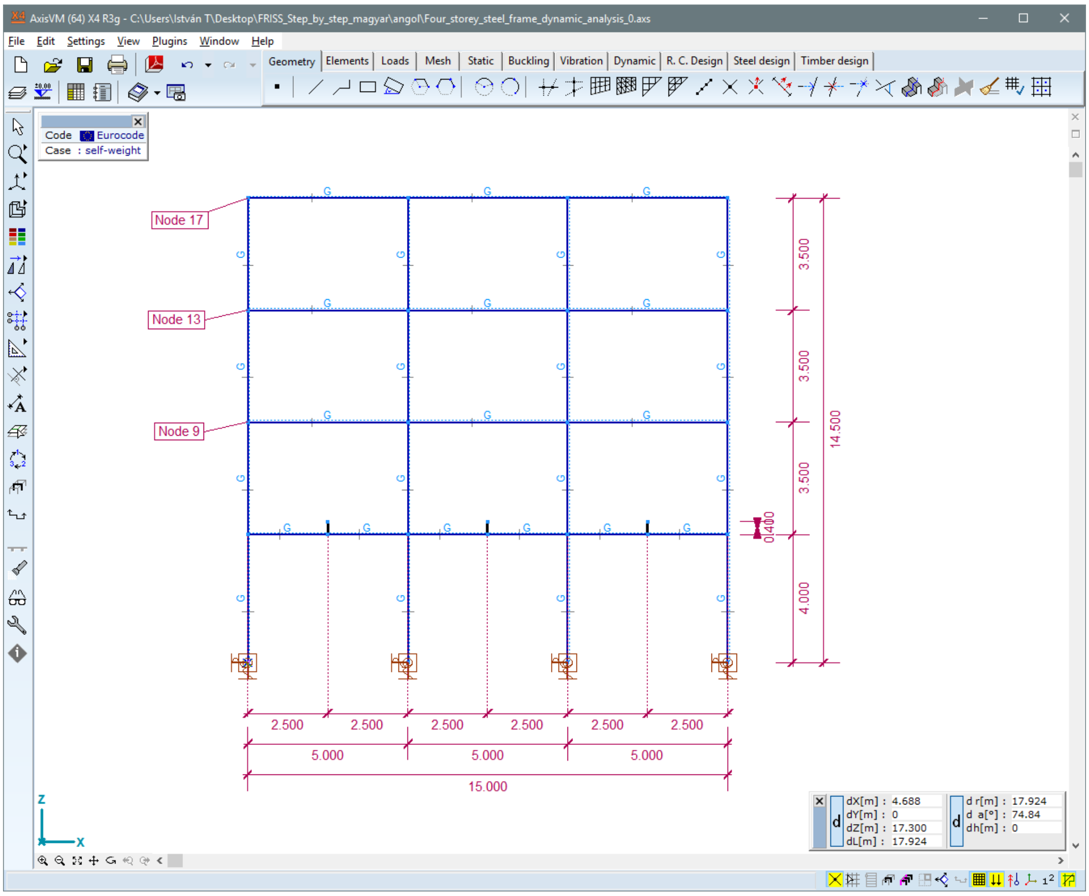


Start

Start ***AxisVMX4*** by double-clicking the ***AxisVMX4*** icon in its installation folder found in the ***Start*** – ***Programs*** menu.

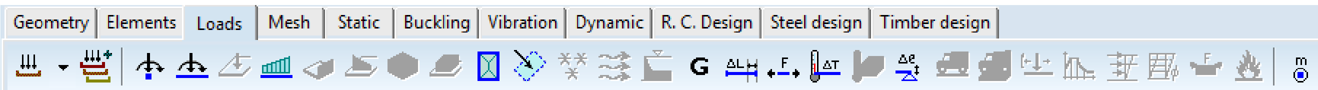
Open

Click on the ***Open*** icon to load the starting file saved on your computer. In the window that appears, choose the directory containing the file and select it. Click on the ***Open*** button to load the model, and the 2D frame structure will appear in the main window:

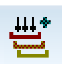


First steps

Before starting the modelling, check the geometry, the finite elements and the settings described in the image above.



Load combinations



Change to the ***Loads*** tab.

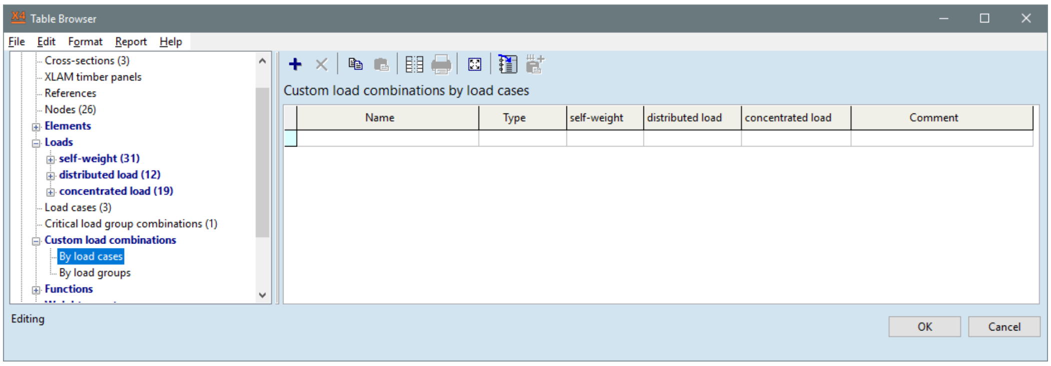
Create load combinations for the vibration and dynamic analysis based on the static loads. In our task, the presented solution is complex, because in the example substantially different dynamic effects are examined in the same model.

The vibration analysis will be executed in two different ways:
- In the case of seismic, sine wave excitation and shock load, it is sufficient to consider only the horizontal (***x***) mass component because the applied dynamic load contains only horizontal components.
- In the case of the machine induced vibration, the vertical component must also be taken into account in addition to the horizontal one.

The same compilation of static loads will be applied for each dynamic load, but due to the setting of considered mass components the vibration analysis will be performed on different combinations (the combination name will be different). The reason for this is that the program binds the results of vibration analysis to the set load case/combination. If a new analysis based on the same load combination is run with different parameters, the

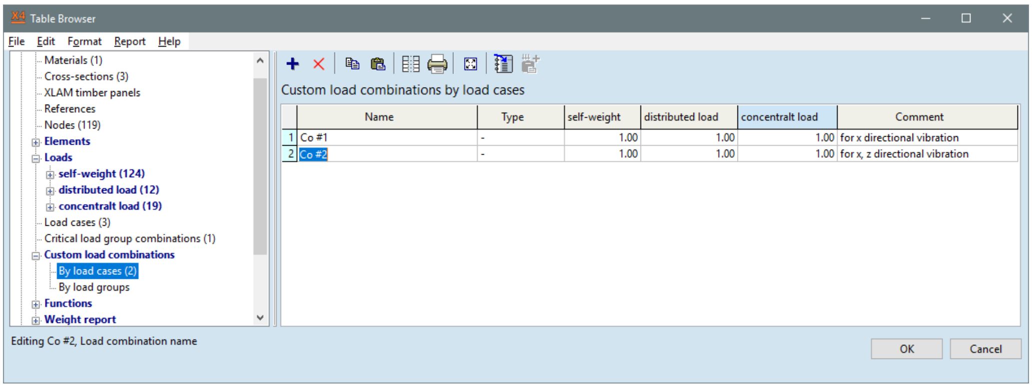
previous results are not stored. Therefore, two different load combinations will be applied to store all the results of different analyses.
Note: this trick can also be applied if the effect of other parameters need to be examined and the results of each analysis have to be saved (e.g. seismic test with different damping parameters).

Click on the ***Load combinations*** icon, and the following window pops up:

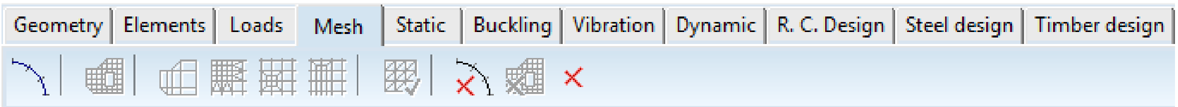


New row

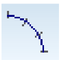
Click on the ***New row*** icon, then create **two** load combinations as follows:
The name of the load combinations should be **Co #1** and **Co #2**. Set the type to "***(user-defined combination)***", apply combination factors of **1.00** for each load case and leave a reminder comment for each combination – "***for x directional comment***" and "***for x, z directional vibration***" as follows:



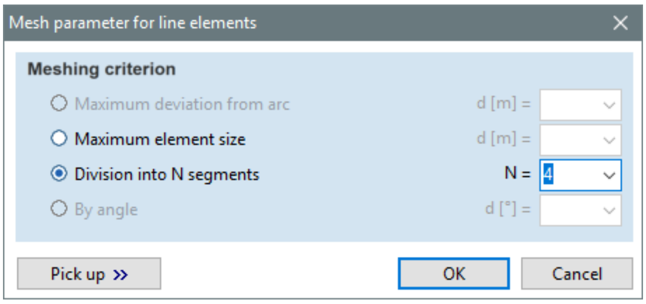
Close the window by clicking ***OK***.



Meshing of line elements

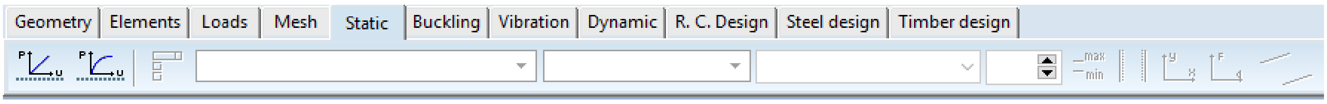


Before the dynamic or nonlinear static analysis, the elements must be meshed to get smooth and correct results. Change to the ***Mesh*** tab and click on the ***Meshing of line elements*** icon. Select all the beam elements and set the desired meshing criteria. Divide each element into **4** (equal) segments.

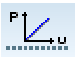


Close the window by clicking on the ***OK*** button.

By clicking on the ***Mesh display on/off*** icon among the ***Speed buttons***, the display of the meshing can be switched on or off. Now, keep the display of the meshing.



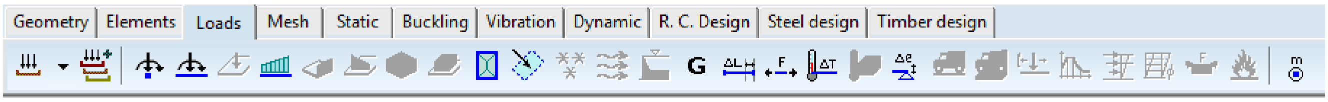
Linear static analysis



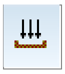
Change to the ***Static*** tab. Perform a linear static analysis to check the load bearing capacity of the frame under static loads.

Now check the main results and make sure that there is no problem with the finite element model (e.g. instability).
Note: Always check the model applying simpler load cases or combinations before running time-consuming nonlinear or dynamic calculations.

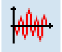
In the next steps, the dynamic loads will be defined.



Load groups and load cases



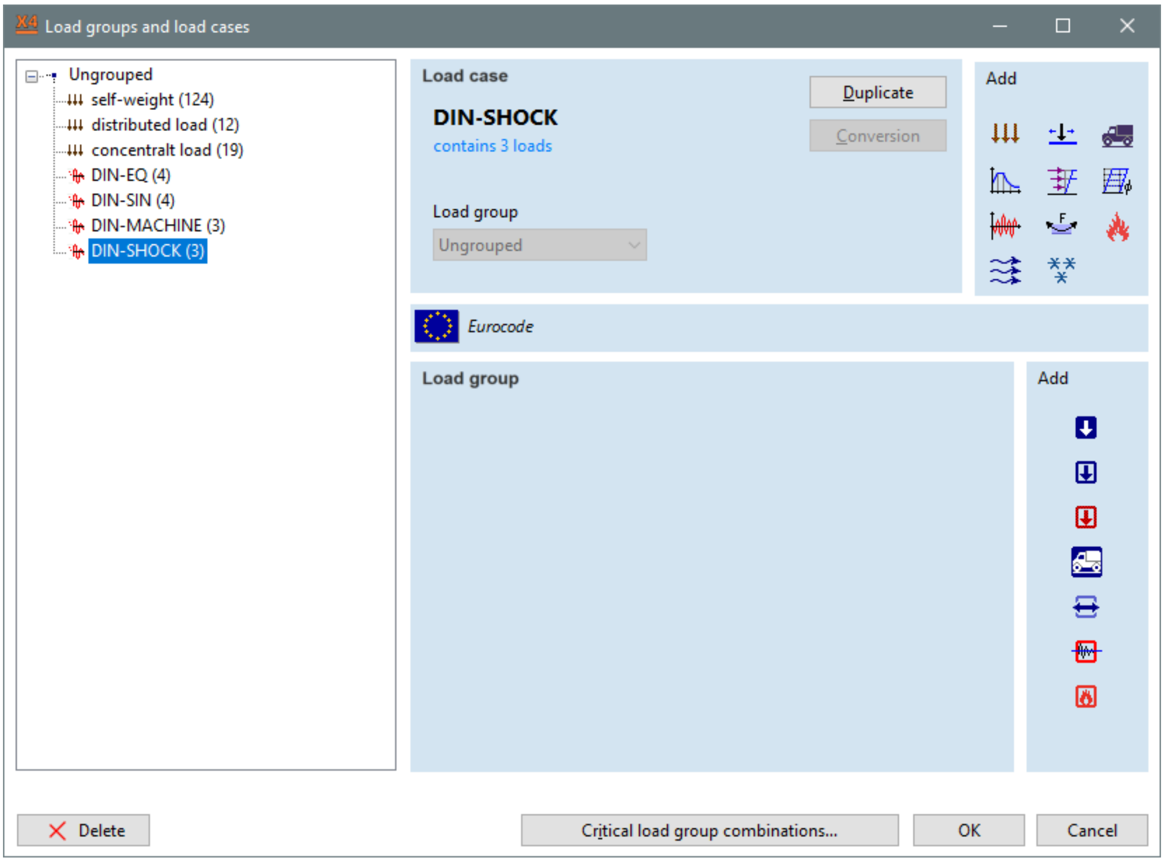
Dynamic load



Change to the ***Loads*** tab. Here, click on the ***Load groups and load cases*** icon. Beside the existing static load cases, create **4 *Dynamic load***s. Click on the ***Dynamic load*** icon and rename the new load cases as follows:

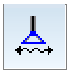
Name of the load case	Type of the load to be defined in the load case
DYN-EQ	seismic acceleration
DYN-SIN	sine wave acceleration
DYN-MACHINE	excitation induced by the rotating machines
DYN-SHOCK	shock load (short-term concentrated load)

After creating the load cases above, the following can be seen in the window:

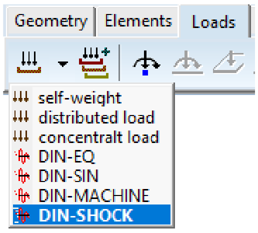


Close the window by clicking ***OK***, then specify the loads in each dynamic load case.

Dynamic support acceleration

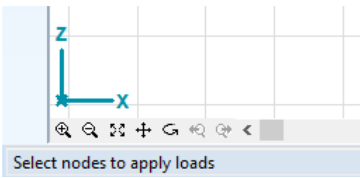


Firstly, a seismic type excitation will be defined. Select the load case named ***DYN-EQ*** (the current load case can be checked in the ***Status*** palette). As a support acceleration, the dataset named **Bucuresti-1986-EW** will be used, which is included in the database of the software.

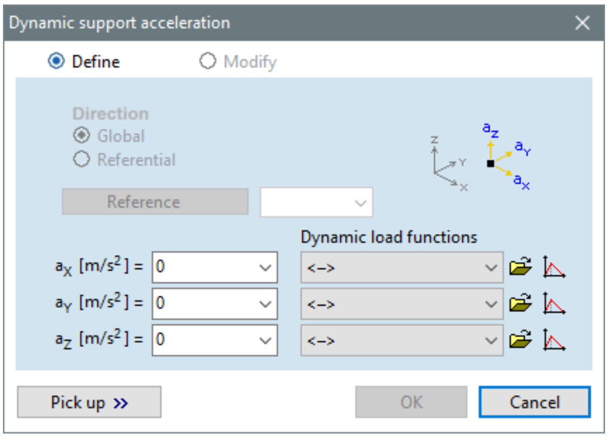


Click on the ***Dynamic support acceleration*** icon and select the nodal supports of the ground floor columns.

Note: if a command is active, the information about the expected current data can be read in the lower left blue-grey strip of the main window.

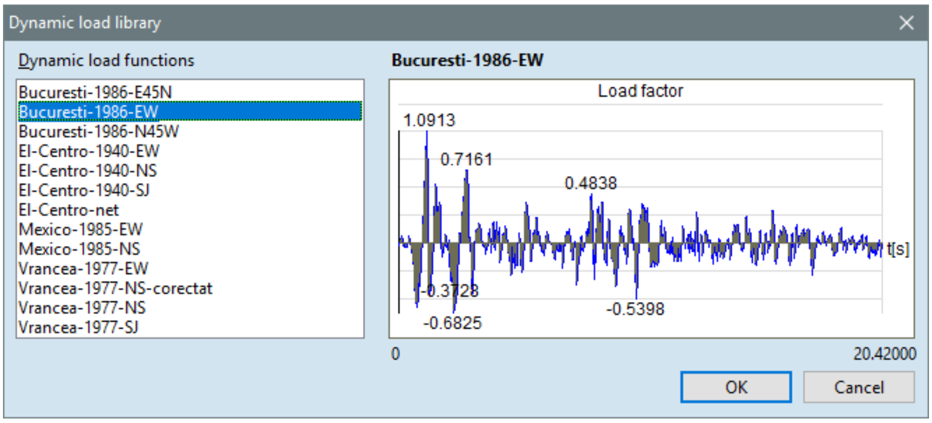


Confirm selection with ***OK***, and the ***Dynamic support acceleration*** window will appear.



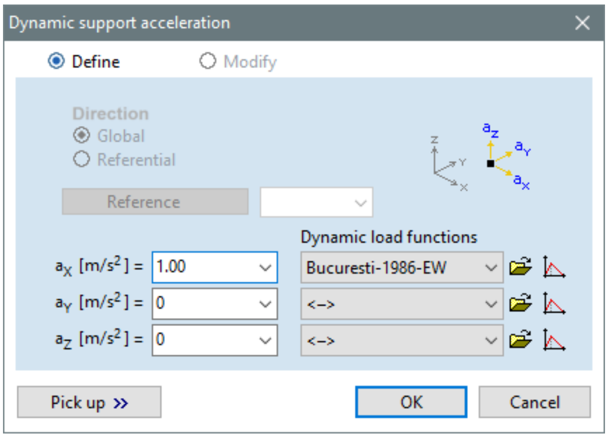
Dynamic load library

Define the global ***x-directional*** seismic acceleration. For the ***ax [m/s2]*** component, click on the ***Dynamic load library*** icon to load the specific function from the database. In the library, select the function named **Bucuresti-1986-EW**, and the earthquake diagram will be displayed on the right window. The presented numeric values are the main minimum and maximum values of the series.

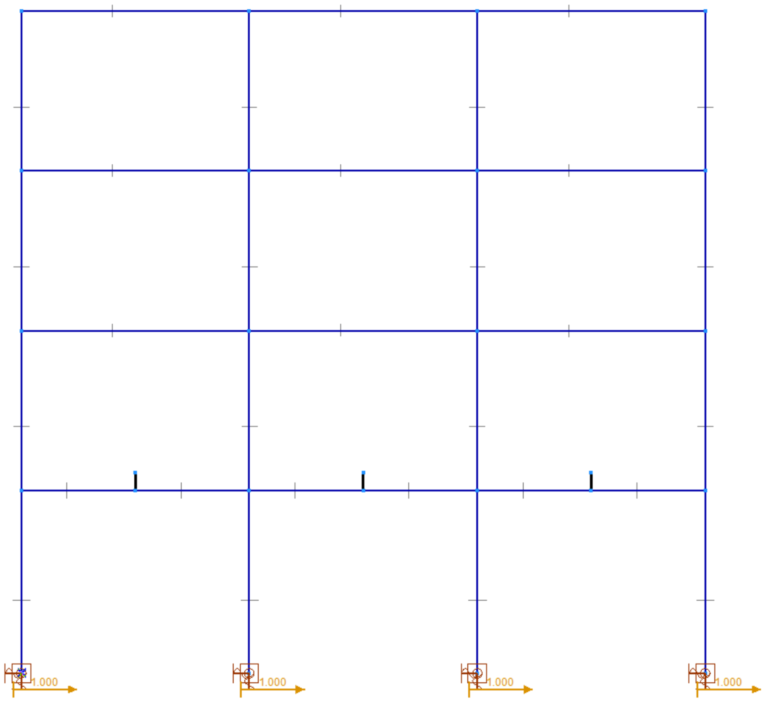


This dataset is the east-west earthquake data series of the Bucharest Earthquake in 1986. It contains data in **0.02 s** time steps, and the total duration of the function is **20.42 s**. The maximum value is **1.0913**. The set is unitless, it is a so-called load factor function. The support acceleration to be considered in the calculation is obtained by multiplying this function with a (constant) acceleration value in the given direction. To recover the values recorded at the the site of earthquake, apply an acceleration value of **1 m/s²**. This will be specified in the next step:

Close the window with ***OK***. In the text box, Set the ***ax [m/s²]*** acceleration constant to **1.00** (setting this constant, allows the load factor function to be scaled without modifying the original dataset).

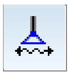


Close the window with ***OK***. In our example, only the global ***x-directional*** acceleration is applied (our model is tested only in the plane of ***x-z***) and the vertical component is neglected. The applied dynamic loads are indicated by the arrows under the supports, and the values above them are the set constant acceleration values.

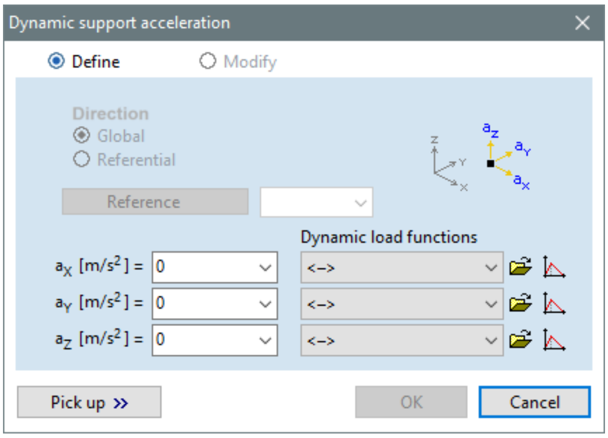


Note: the user can also specify custom functions using the ***Function editor***. The functions can also be saved to the database and can be loaded in other models. The use of the editor will be described later.

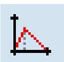
Dynamic support acceleration



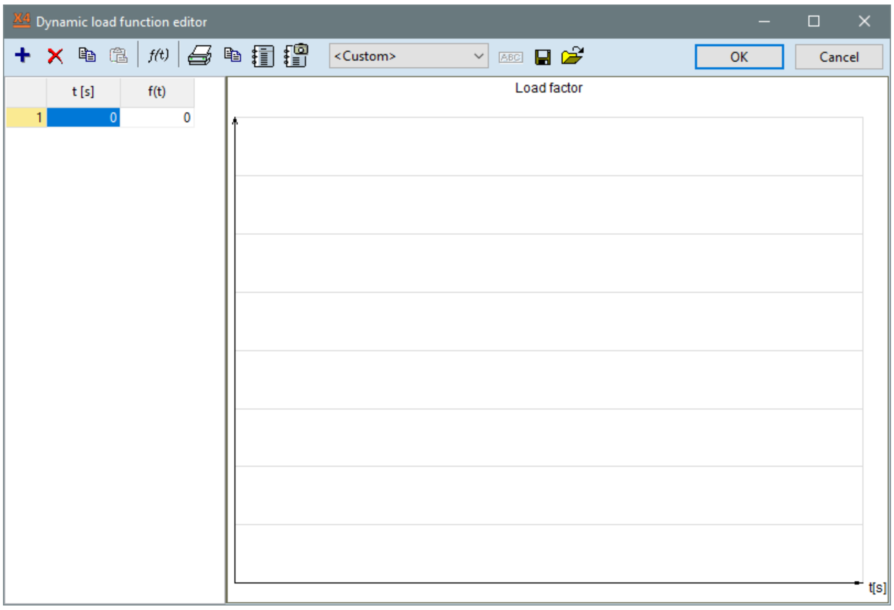
In the second step, define the ***SINE*** wave-like support acceleration. Change the load case to **DYN-SIN** and click on the ***Dynamic support acceleration*** icon. Select all the nodal supports again.



Function editor…

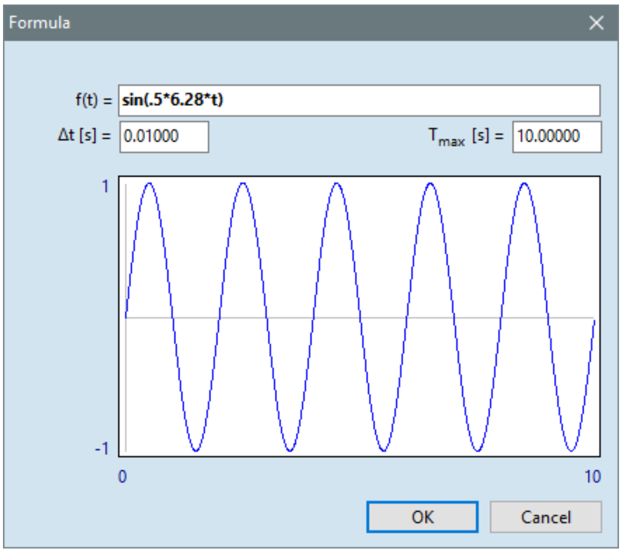


Simple functions (sin, exp, sqrt, etc...) can be defined easily in the ***Function editor*** using formulas. Considering the ***ax [m/s²]*** component, click on the ***Function editor…*** icon.



Formula

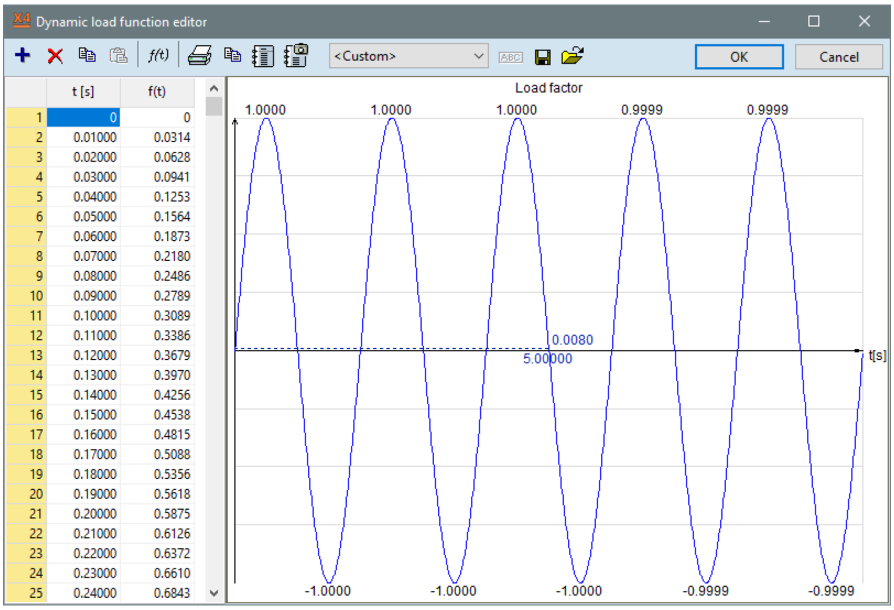
In the ***Dynamic load function editor*** window click the ***Formula*** icon. In the window that appears, define a sine wave for the acceleration.



The formula must be given as a function of "***t***" (time).

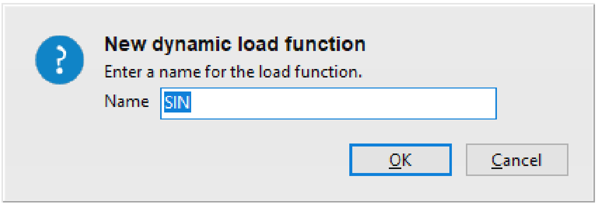
Enter the following formula in the input field of ***f(t)***: **sin(.5*6.28*t).** (Based on the basic formula sin(2*π/T*t) the period time (***T***) of the harmonic function will be **2 s**, and its frequency is **0.5 Hz**.)

The list of the applicable operators can be found in chapter ***4.10.28*** of the User's Manual. Using the formula, the software generates the discrete series considering the given time step (**Δt=0.01 s**) and the duration of the excitation (**Tmax=10 s**). Close the window by pressing ***OK***, and the ***Dynamic load function editor*** will appear again:

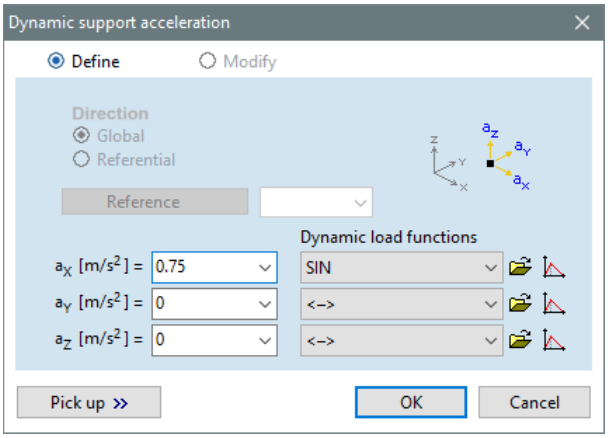


On the right side of the window, the specified function can be seen. The table on the left contains the full discrete data series of the curve considering the given time step. The whole data can be looked through. If necessary, each data in the set can be edited manually.

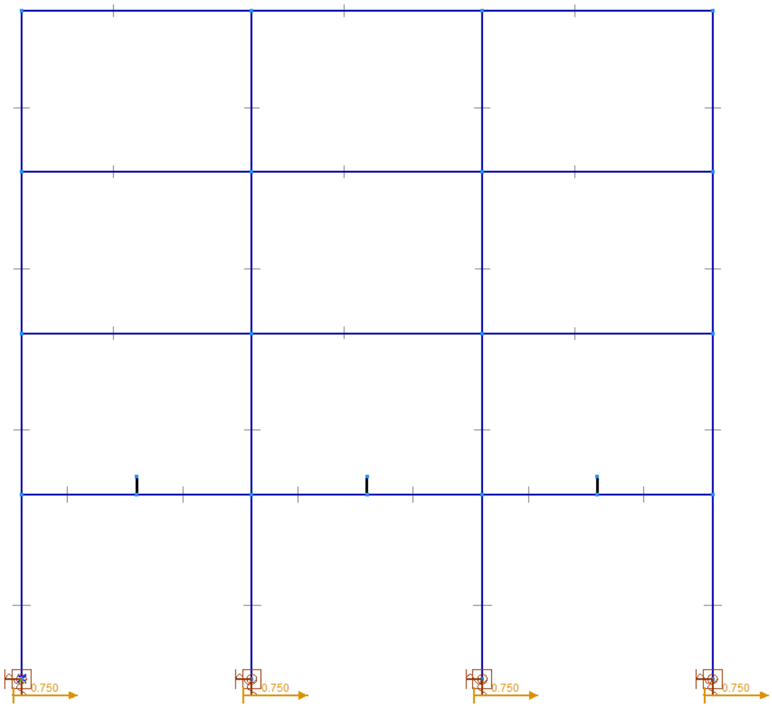
Close the editor, and a new window pops up to save the set load function. Save the specified function with name **SIN**.



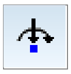
In the ***Dynamic support acceleration*** window, specify the acceleration constant value for the load function, which should be **0.75 m/s²** in this example.



Close the window with ***OK***. The applied dynamic loads are indicated by the arrows under the supports, the values above them are the set constant acceleration values.

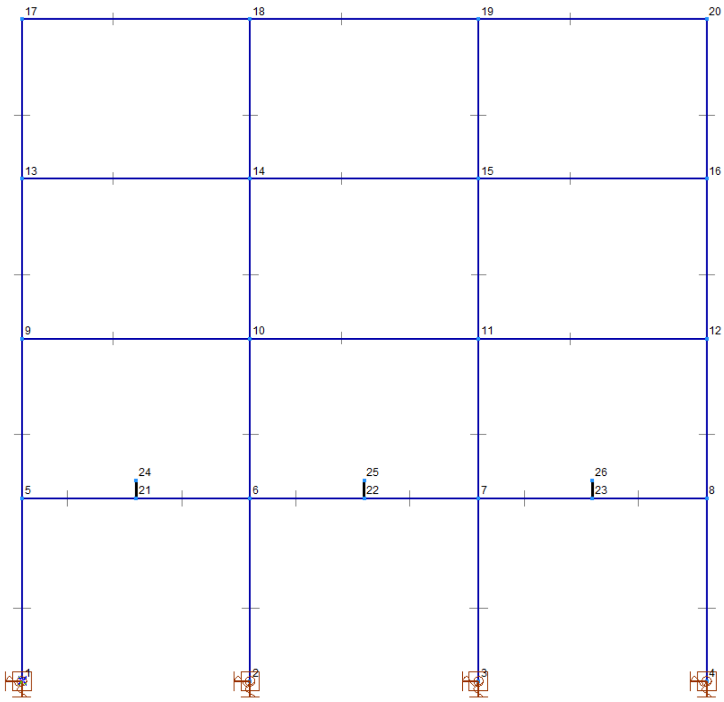


Nodal load

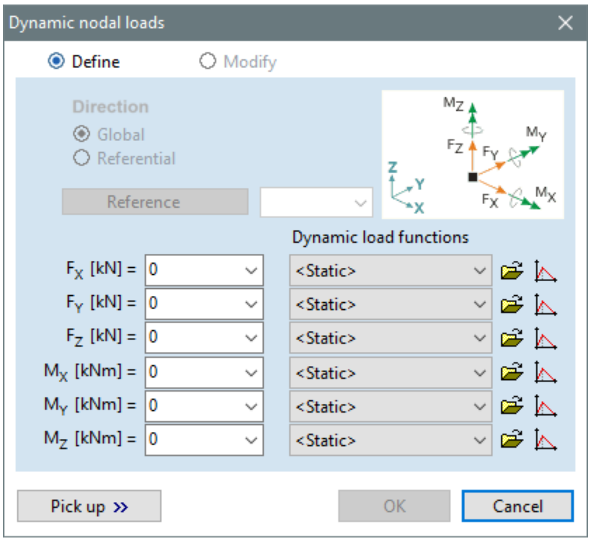


Next, let's model the dynamic effect of the rotating industrial machines. We will use a time-varying concentrated load to simulate the effect of the rotating machines (rotating around the global ***y*** axis). In the example, it is supposed that the machines have simultaneous phase motion.

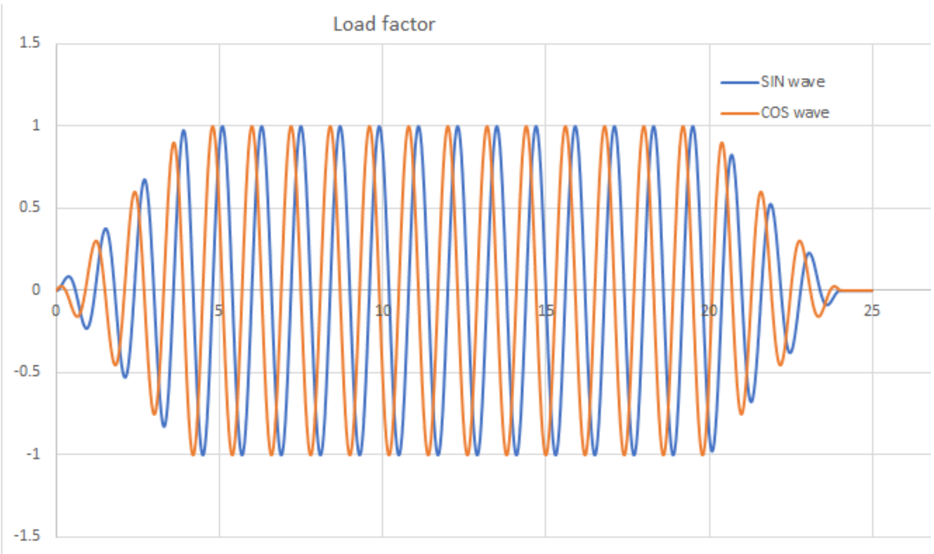
Assign the following dynamic loads to nodes **24**, **25** and **26**.



Change load case to **DYN-MACHINE** and click on the ***Nodal load*** icon and select the nodes mentioned above. The following window pops up after selection:



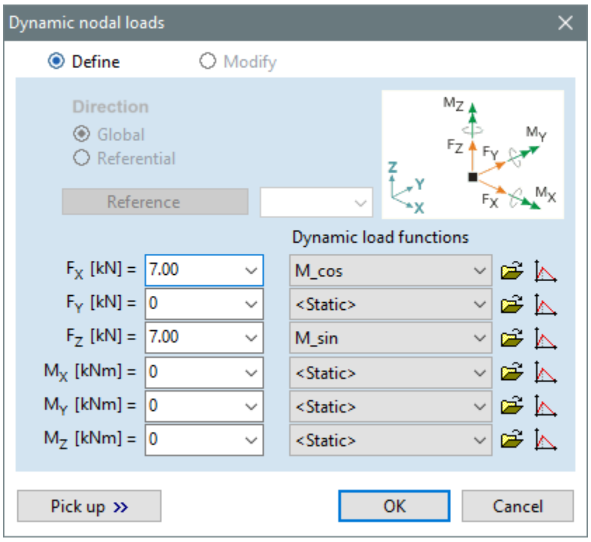
The time-varying resultant force is modelled by its global ***x*** and ***z***-directional components. For this purpose, two dynamic load functions should be applied to describe the circular motion. For the ***x*** directional force, the **cosine** function is used, and in the ***z*** direction the **sine** function is applied.
The time series to be applied starts with a monotonic, linearly increasing part, followed by a pure harmonic waveform, and ends with a monotonous decreasing part. The maximum amplitude of the **sine** and **cosine** functions is **1.00,** the period time is **1.2 s**. The complete series was prepared in a ***Microsoft Excel*** spreadsheet because of its complexity. It was generated with a timestep of **0.01 s** and the total time is **25 s**. This data is in the enclosed ***Excel*** file and the diagrams described above are the following:



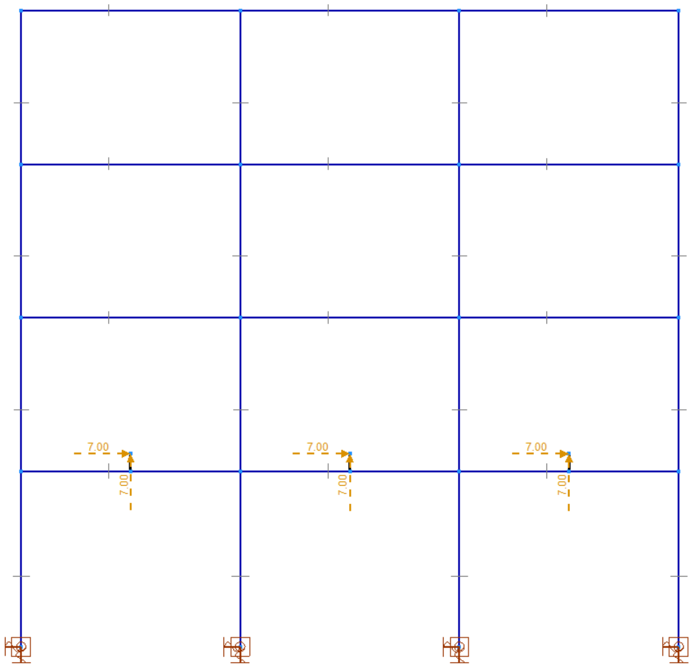
Specify the load factor functions of the ***Fx*** and ***Fy*** nodal forces one after another.
Copy the artificially generated data (time and value columns) into the cells of the ***Function editor***.
Highlight and copy the whole data set starting from the second row. Paste the data into the table in **AxisVM** starting in the first cell (**t=0 s**). Take care to not copy the row of **t=0 s**, because the first row cannot be

overwritten in this way. After completing this procedure, the diagram will immediately appear in the right-hand window. Compare the result with the curve shown in the ***Excel*** table.

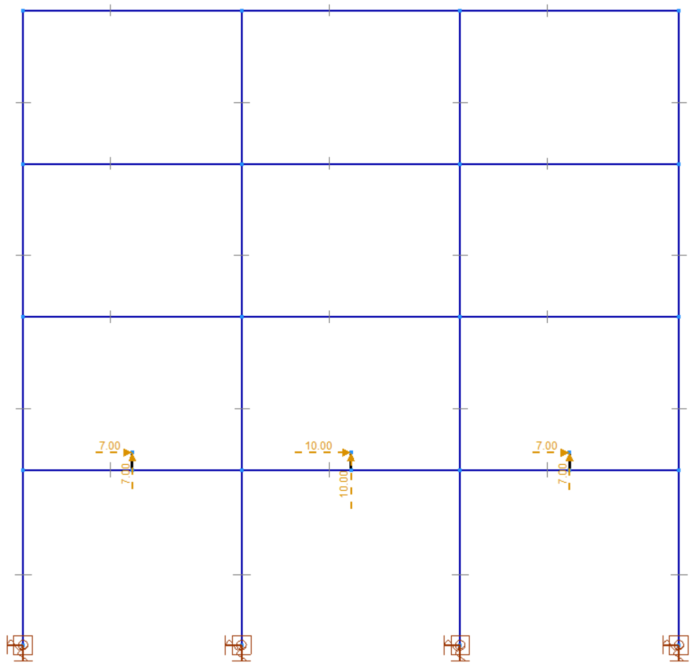
Save the new functions with names **M_cos** (***x*** direction) and **M_sin** (***z*** direction). Close the editor, set the constant load to **7 kN** for both components (the load factor function will be multiplied with this constant value).



Close the window by pressing ***OK***. The applied dynamic nodal loads are indicated by the arrows (in the directions of ***x*** and ***z***) under the supports, the values above them are the set constant load values.

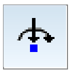


Increase the magnitude of the load in the middle. Click on the sign of the load and change the value of the ***Fx*** and ***Fz*** components to **10 kN**. After closing the window, the next result can be seen:

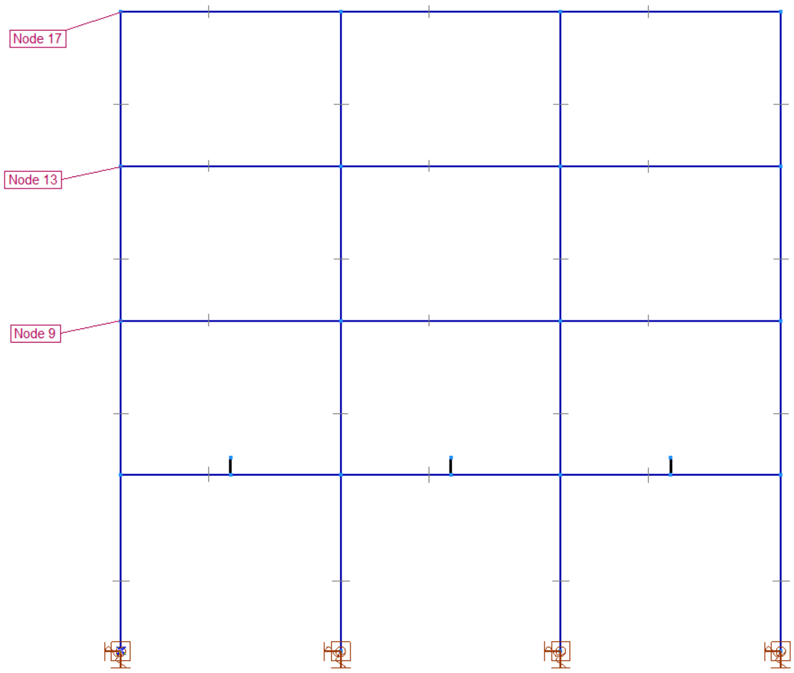


Note: a possible modelling method has been presented above. For correct modelling of the effect of a rotating machine, always ask for detailed information from the manufacturer.

Nodal load

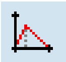


Finally, apply a short-term dynamic nodal load as a shock load in nodes **9**, **13** and **17** in the **DYN-SHOCK** load case.



Change the load case to ***DYN-SHOCK***, and click on the ***Nodal load*** icon. Select the nodes marked in the figure above and confirm selection with the ***OK***. The steps of the data input are similar to before: firstly, the load factor function should be defined, then the value of the constant load should be set.

Function editor…



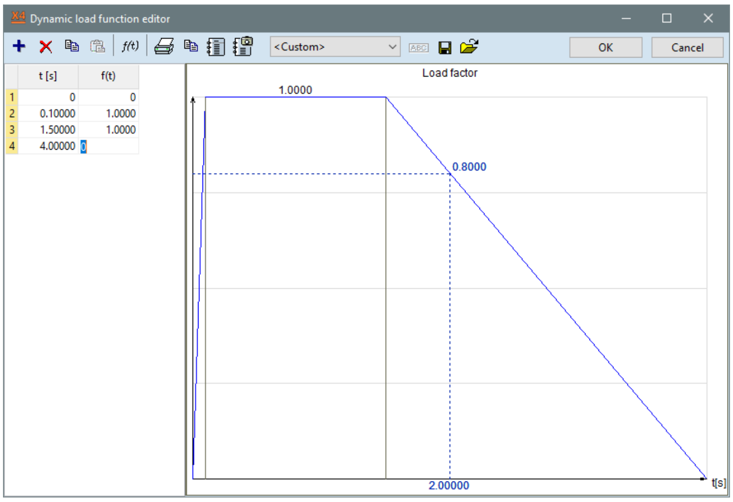
In the line of the ***Fx*** component click on the ***Function editor…*** icon and define the load factor function according to the following series of data:

	t [s]	f(t)
1	0	0.00
2	0.1	1.00
3	1.5	1.00
4	4.0	0.00

The function is defined by the breakpoints. Intermediate values are generated by linear interpolation based on the time increment set in the dynamic analysis parameters (see later).

Adds a new row

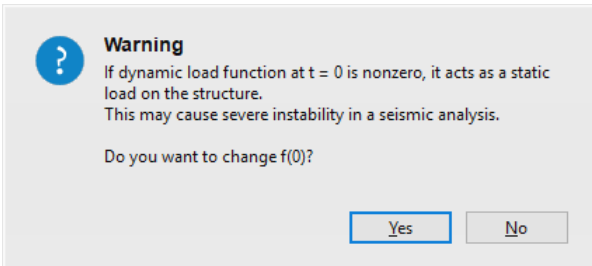
The data has to be entered step by step by inserting new data rows and entering the corresponding values. After entering, we get the next curve as a result:



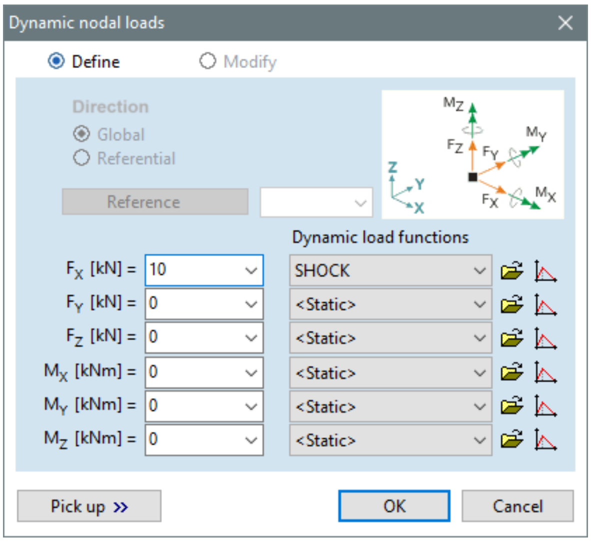
The maximum is **1.00**, and the total load will be scaled by adjusting the ***Fx*** constant value. After entering the data, close the window with ***OK*** and save the custom function with name **SHOCK**.

Additional remark:

If the dynamic load function at the first step (**t=0 s**) is nonzero, then it acts as a static load on the structure. In this case the software warns the user before saving the load series:

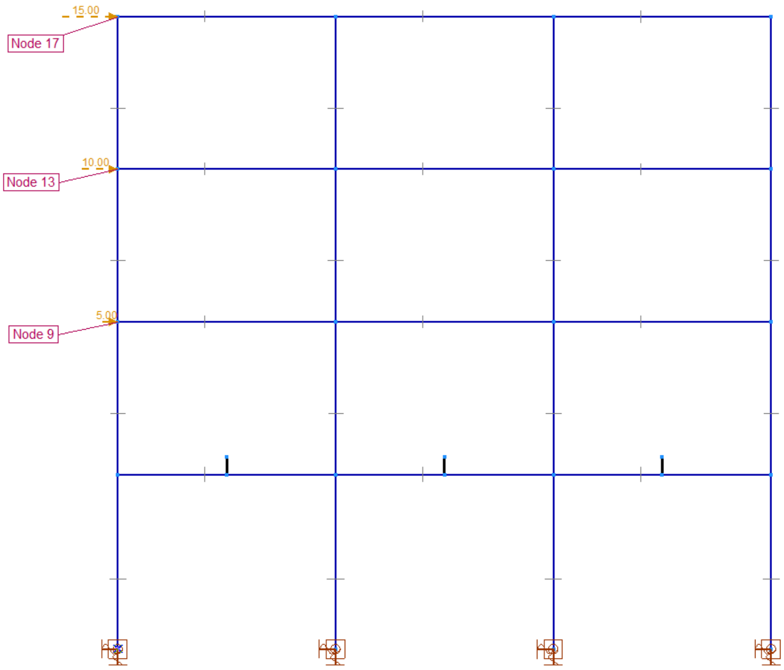


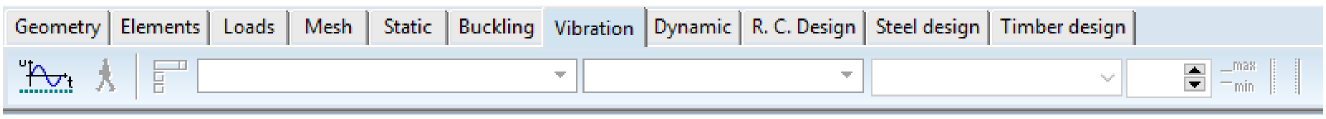
The ***Dynamic nodal loads*** window pops up after saving the function.



Set the ***Fx*** constant load to **10 kN** and confirm the changes with ***OK***. The given loads are indicated by the arrows, the value above them are the set constant load values.
Change the value of ***Fx*** in nodes **17** and **9**. Increase load in node **17**. Click on the load that represents the dynamic load and in the pop-up window, change ***Fx*** to **15 kN**.
Similarly, reduce the load in node **9**, by changing the value of ***Fx*** to **5 kN**.

The following figure shows the result:



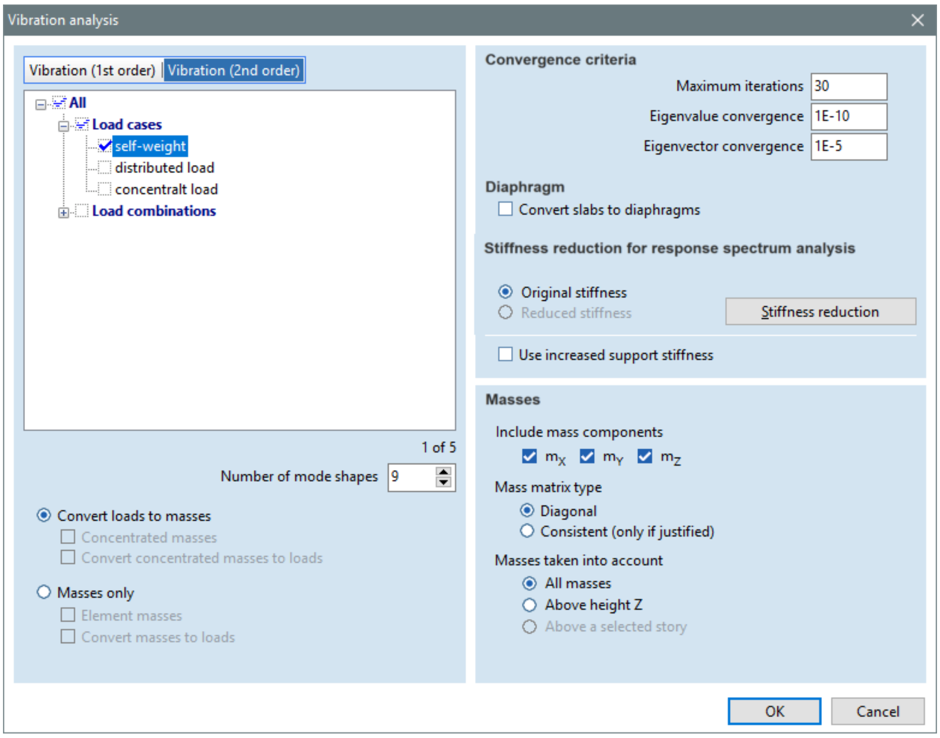


Vibration analysis

Change to the ***Vibration*** tab and perform a vibration analysis to determine the lowest natural frequencies and mode shapes. This analysis has more roles. In order to avoid a resonance effect, the natural frequencies must be compared with that of different excitations (sine wave, machine induced vibration).

On the other hand, the Rayleigh damping constants in the dynamic analysis will be tuned considering the dominant vibration shapes in our example.

Click on the ***Vibration analysis*** icon, and the next window pops up:



Firstly, the type of vibration analysis has to be selected. The default is the ***1st order analysis***, but now let us use a ***2nd order analysis***.
Note: Second order analysis means that the solution includes the effect of axial forces of truss/beam elements on the system stiffness. Tension axial forces have a stiffening effect, while the compression axial forces have a softening effect. These effects influence the free vibrations of the structure.

In the vibration analysis the masses can be converted by the loads defined in a load case/combination (only the load components parallel with gravity can be converted to masses). As the basis of this procedure, select the correspondent load case or combination. In the list of the available cases, the software automatically selects the first static load case **self-weight**. Uncheck this and mark the specific load combination. Two different analysis will be executed based on the mass components to be considered, see later. Select **Co #1** for the first analysis and **Co #2** for the second one

Our model does not contain mass (created on the ***Loads*** tab), so the program offers the option ***Convert loads to masses*** automatically.

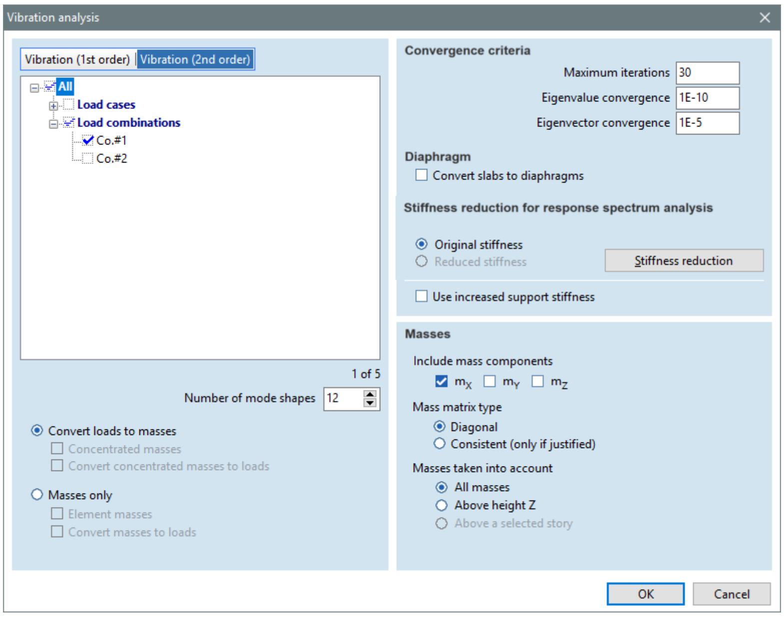
Set the number of mode shapes to **12** in the first analysis and to **30** in the second one. (Considering vertical mass component, more shapes have to be determined. Beside the main shapes, several shapes with lower modal mass factor will appears – see the results later.)

Now set the number of ***Maximum iterations*** to **30.**

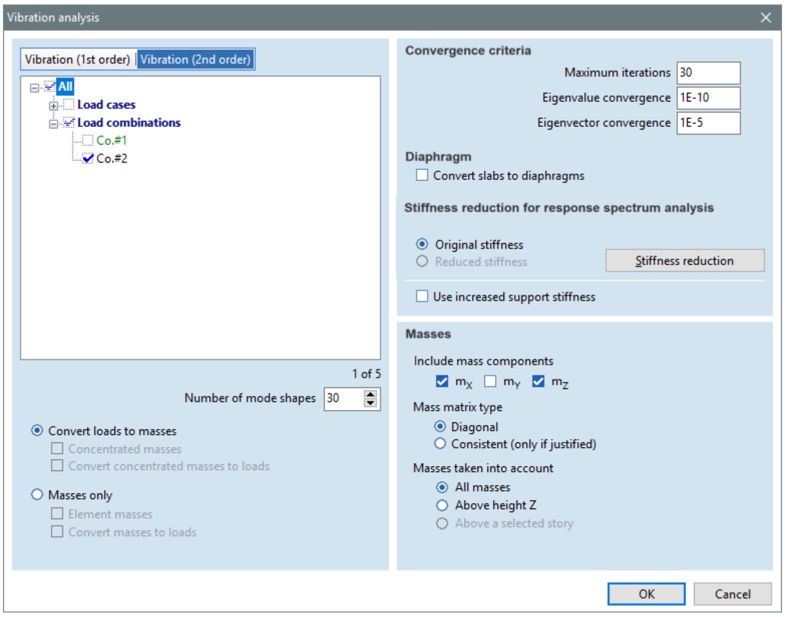
As described before, in the first analysis, only the ***x***-directional mass components are needed, the others have to be switched off. In the second case select also ***m_z*** component.

Set the parameters as follows, then click ***OK*** to start the calculation. After completing the first analysis, modify the settings according to the following and perform a new analysis.

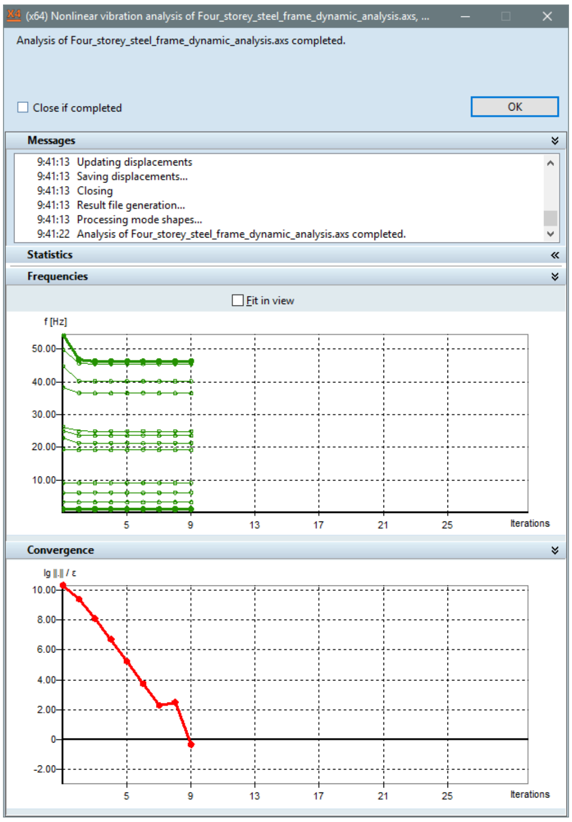
Settings for the **first analysis**:



Settings for the **second analysis**:



During the analysis, the user is given information about the progress of the calculation. The ***Frequencies*** panel displays how the frequencies converge step by step. ***Convergence*** shows the convergence process, but it only shows the convergence of the slowest converging mode shape.



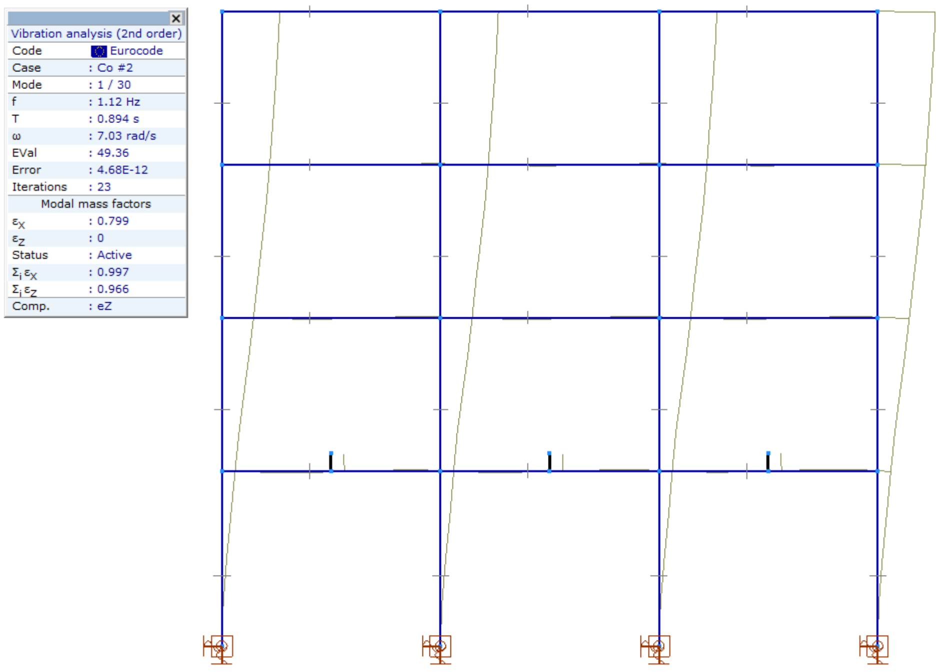
If the calculation is completed, close the window with ***OK***.

Let's have a look at the determined modes shapes in each case (**Co #1** and **Co #2**). The results of the analysis are nearly identical considering the x-directional shapes, the main shapes containing vertical displacement appear only in the higher modes (5-30).

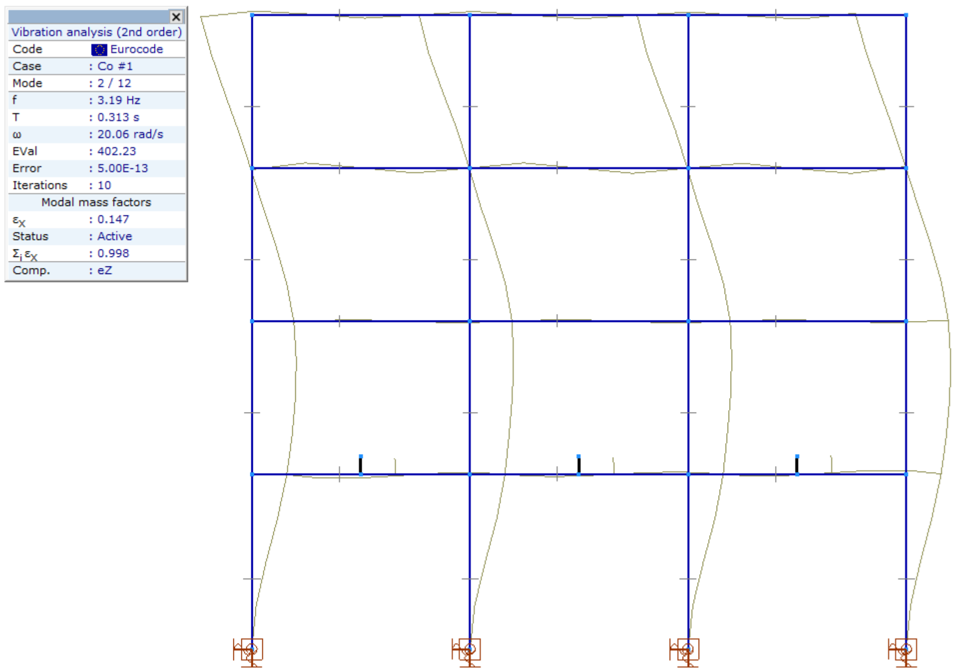
The main data of a displayed shape is also shown in the ***Status palette***.

In our model, only the first five shapes will be dominant based on the two calculations.

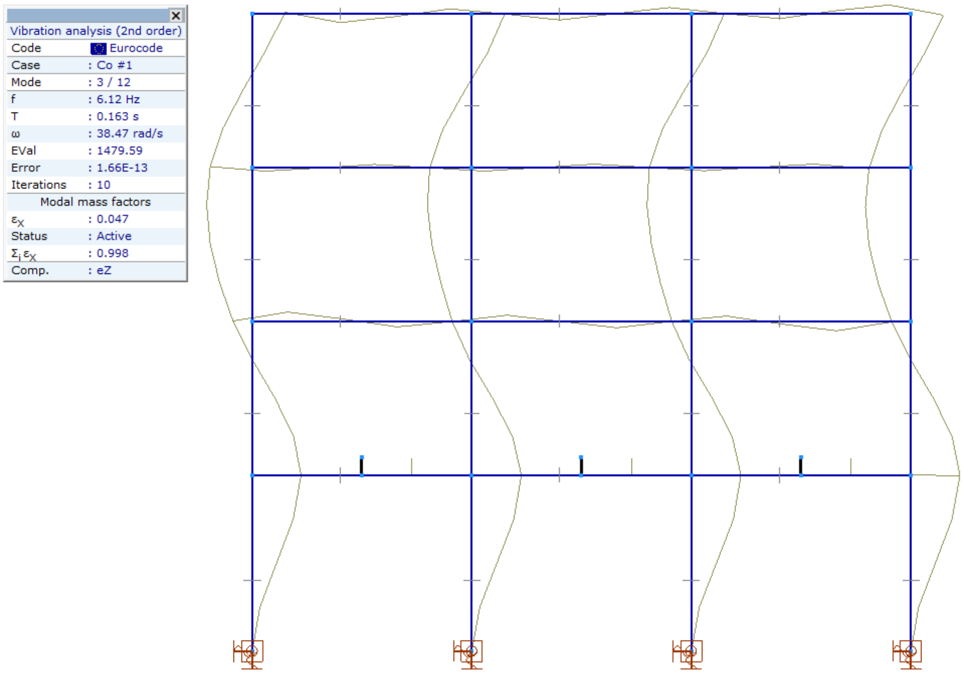
1st mode shape (load combination **Co #1**):



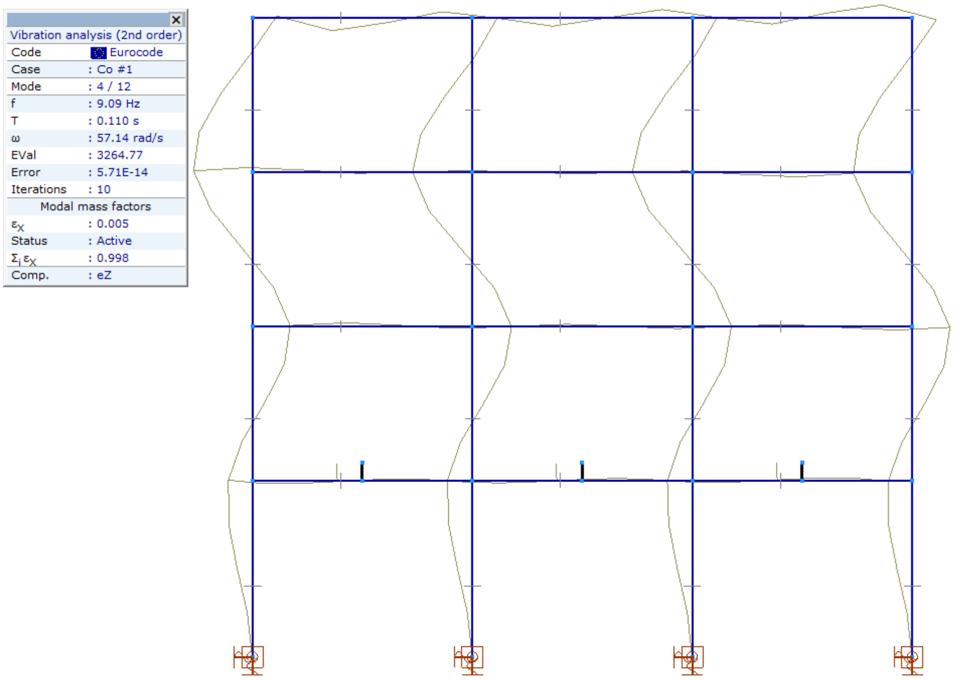
2nd mode shape (load combination **Co #1**):



3rd mode shape (load combination **Co #1**):



4th mode shape (load combination **Co #1**):



5th mode shape (load combination **Co #2**):

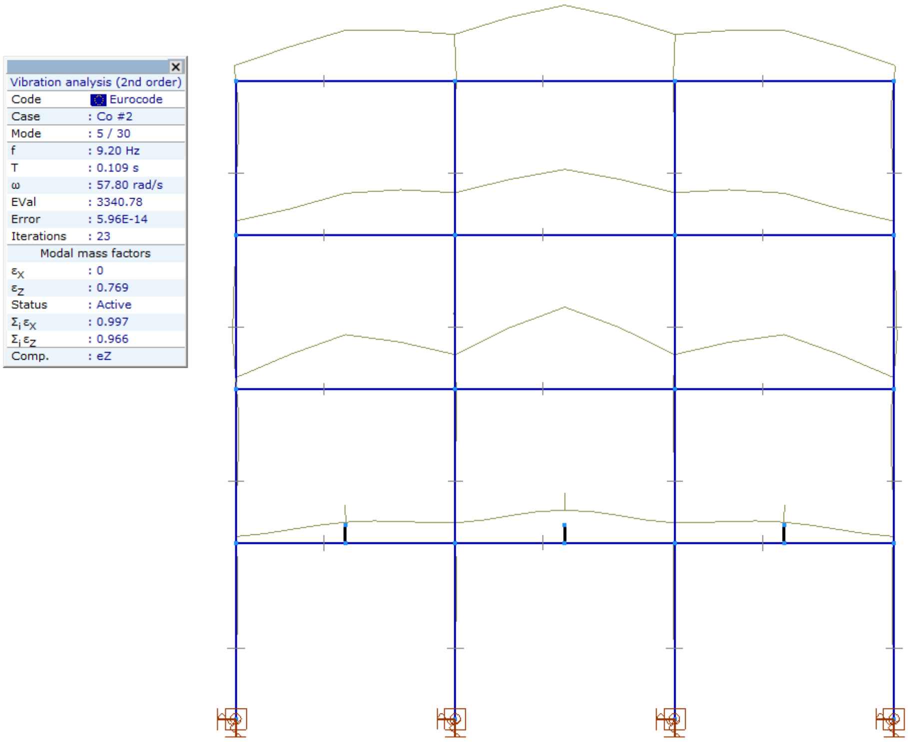
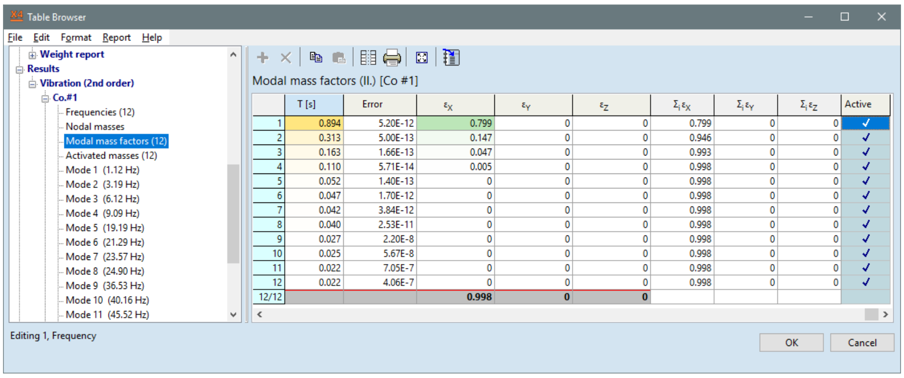


Table browser

Open the ***Table browser*** to see the summarized data of the mode shapes and their modal mass factors:

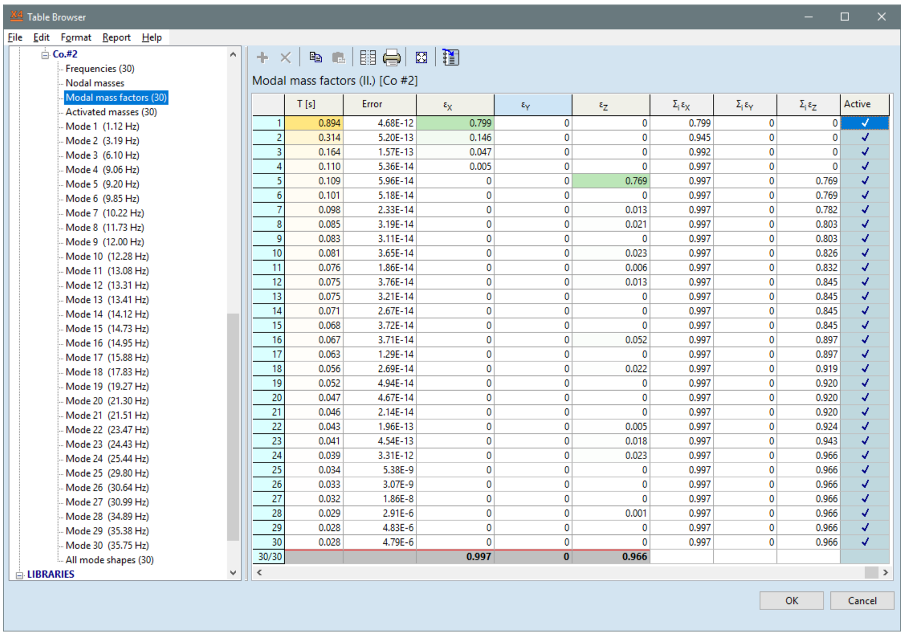
Based on the parameters set in the **Co #1** combination:



Modal mass factors (II.) [Co #1]

	T [s]	Error	ε_X	ε_Y	ε_Z	$\Sigma_i \varepsilon_X$	$\Sigma_i \varepsilon_Y$	$\Sigma_i \varepsilon_Z$	Active
1	0.894	5.20E-12	0.799	0	0	0.799	0	0	✓
2	0.313	5.00E-13	0.147	0	0	0.946	0	0	✓
3	0.163	1.66E-13	0.047	0	0	0.993	0	0	✓
4	0.110	5.71E-14	0.005	0	0	0.998	0	0	✓
5	0.052	1.40E-13	0	0	0	0.998	0	0	✓
6	0.047	1.70E-12	0	0	0	0.998	0	0	✓
7	0.042	3.84E-12	0	0	0	0.998	0	0	✓
8	0.040	2.53E-11	0	0	0	0.998	0	0	✓
9	0.027	2.20E-8	0	0	0	0.998	0	0	✓
10	0.025	5.67E-8	0	0	0	0.998	0	0	✓
11	0.022	7.05E-7	0	0	0	0.998	0	0	✓
12	0.022	4.06E-7	0	0	0	0.998	0	0	✓
12/12			**0.998**	**0**	**0**				

In the **Co #2** combination:

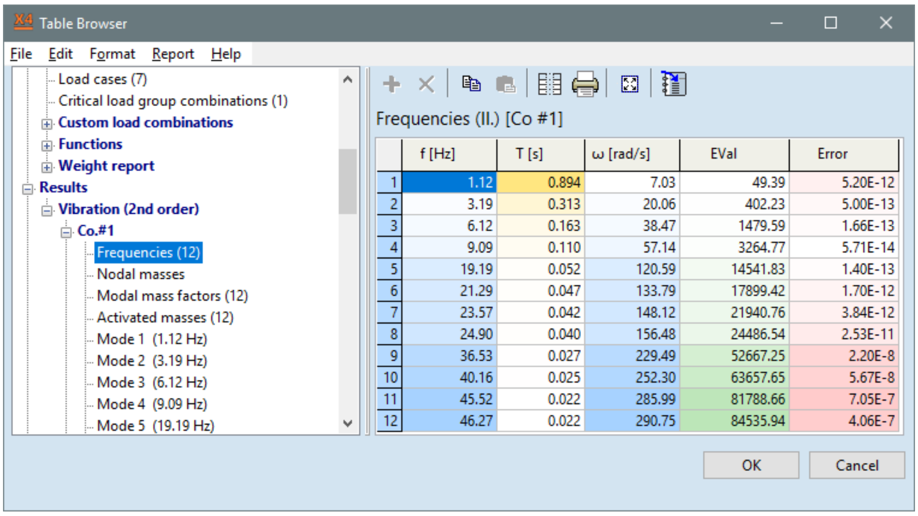


Modal mass factors (II.) [Co #2]

	T [s]	Error	ε_X	ε_Y	ε_Z	$\Sigma_i \varepsilon_X$	$\Sigma_i \varepsilon_Y$	$\Sigma_i \varepsilon_Z$	Active
1	0.894	4.68E-12	0.799	0	0	0.799	0	0	✓
2	0.314	5.20E-13	0.146	0	0	0.945	0	0	✓
3	0.164	1.57E-13	0.047	0	0	0.992	0	0	✓
4	0.110	5.36E-14	0.005	0	0	0.997	0	0	✓
5	0.109	5.96E-14	0	0	0.769	0.997	0	0.769	✓
6	0.101	5.18E-14	0	0	0	0.997	0	0.769	✓
7	0.098	2.33E-14	0	0	0.013	0.997	0	0.782	✓
8	0.085	3.19E-14	0	0	0.021	0.997	0	0.803	✓
9	0.083	3.11E-14	0	0	0	0.997	0	0.803	✓
10	0.081	3.65E-14	0	0	0.023	0.997	0	0.826	✓
11	0.076	1.86E-14	0	0	0.006	0.997	0	0.832	✓
12	0.075	3.76E-14	0	0	0.013	0.997	0	0.845	✓
13	0.075	3.21E-14	0	0	0	0.997	0	0.845	✓
14	0.071	2.67E-14	0	0	0	0.997	0	0.845	✓
15	0.068	3.72E-14	0	0	0	0.997	0	0.845	✓
16	0.067	3.71E-14	0	0	0.052	0.997	0	0.897	✓
17	0.063	1.29E-14	0	0	0	0.997	0	0.897	✓
18	0.056	2.69E-14	0	0	0.022	0.997	0	0.919	✓
19	0.052	4.94E-14	0	0	0	0.997	0	0.920	✓
20	0.047	4.67E-14	0	0	0	0.997	0	0.920	✓
21	0.046	2.14E-14	0	0	0	0.997	0	0.920	✓
22	0.043	1.96E-13	0	0	0.005	0.997	0	0.924	✓
23	0.041	4.54E-13	0	0	0.018	0.997	0	0.943	✓
24	0.039	3.31E-12	0	0	0.023	0.997	0	0.966	✓
25	0.034	5.38E-9	0	0	0	0.997	0	0.966	✓
26	0.033	3.07E-9	0	0	0	0.997	0	0.966	✓
27	0.032	1.86E-8	0	0	0	0.997	0	0.966	✓
28	0.029	2.91E-6	0	0	0.001	0.997	0	0.966	✓
29	0.028	4.83E-6	0	0	0	0.997	0	0.966	✓
30	0.028	4.79E-6	0	0	0	0.997	0	0.966	✓
30/30			0.997	0	0.966				

In the list on the left side, select the ***Frequencies*** option to display another type of summary table which contains the frequency, period, angular frequency and relative error of each eigenvalue.

Based on the parameters set in the **Co #1** combination:

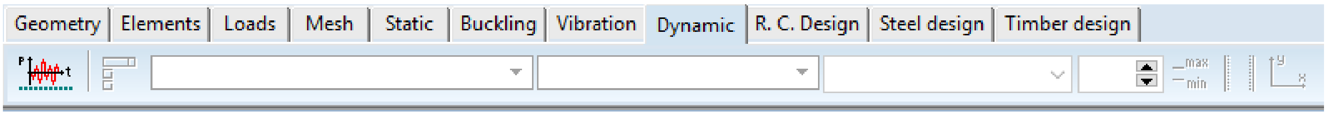


Frequencies (II.) [Co #1]

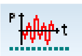
	f [Hz]	T [s]	ω [rad/s]	EVal	Error
1	1.12	0.894	7.03	49.39	5.20E-12
2	3.19	0.313	20.06	402.23	5.00E-13
3	6.12	0.163	38.47	1479.59	1.66E-13
4	9.09	0.110	57.14	3264.77	5.71E-14
5	19.19	0.052	120.59	14541.83	1.40E-13
6	21.29	0.047	133.79	17899.42	1.70E-12
7	23.57	0.042	148.12	21940.76	3.84E-12
8	24.90	0.040	156.48	24486.54	2.53E-11
9	36.53	0.027	229.49	52667.25	2.20E-8
10	40.16	0.025	252.30	63657.65	5.67E-8
11	45.52	0.022	285.99	81788.66	7.05E-7
12	46.27	0.022	290.75	84535.94	4.06E-7

In a time history analysis, the factors for Rayleigh damping have to be set based on the main, dominant frequencies. The constants will be tuned by considering a range of frequencies, the boundary values are the following in our cases:

- Based on the first analysis (**Co #1**), the lowest frequency is **1.12Hz**, the maximum is **9.09Hz**,
- In the second analysis (**Co #2**), the lowest frequency is the same (**1.12Hz**), and the maximum is **9.20Hz**.



Dynamic analysis



After the vibration analysis, change to the ***Dynamic*** tab.Click on the ***Dynamic analysis*** icon and perform the necessary calculations for the four dynamic load cases considering the following settings:

Load cases:

Simultaneously with the dynamic effect, apply a static load on the frame as well. The first static load case (**self-weight**) is automatically selected as default. Change this to the load combination **Co #1** for each run. (**Co #2** contains the same loads, it has to be used only in vibration analysis to store the results of different analysis.)

Note: if no static load case or combination is selected, only the structural mass can be considered or the given concentrated masses can be converted to loads in the analysis.

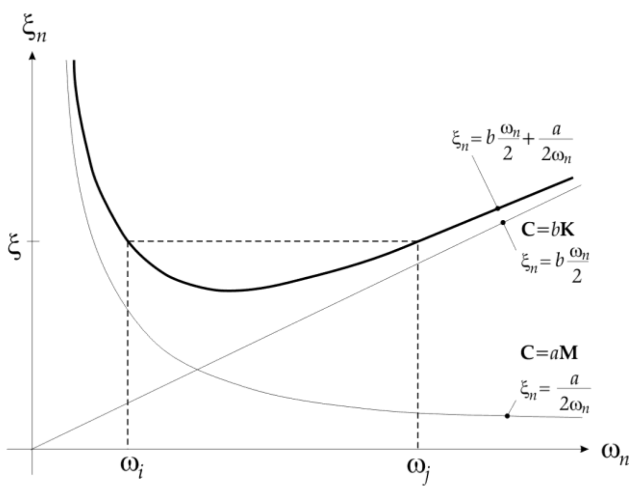
Under this option, the specific ***Dynamic load case*** has to be selected.

Solution control:
When setting the time increment, the dynamic properties of the excitation and the response of the structure must be considered. It should be set depending on the highest frequency of the excitation. **Note that increasing the time increment affects the quality and accuracy of the results, the numerical error may increase.**
The time increment may differ from the time step specified in the load function, the software uses linear interpolation to determine intermediate values.

To check the response of the structure, select a typical node in the structure where (greater) displacement is expected under the excitation. During analysis, its displacement, acceleration and velocity can be tracked. Set the ***Direction x*** component (since this is the main direction of the expected motion) and select the top left node of the frame – node **17**.

Linear or nonlinear equilibrium equations are solved with the Newmark-beta method. **C** matrices are calculated from the Rayleigh damping constants: $\mathbf{C} = a\mathbf{M} + b\mathbf{K}$, where **M** are the mass matrices and **K** are the stiffness matrices. Constants **a** and **b** should be calculated from the damped frequency range (between $\mathbf{f_i}$ and $\mathbf{f_j}$) and the damping ratio.

The damping function is explained in the next figure and the following formulas are to be used to determine the necessary constant values.



$$a = \xi \frac{2\omega_i \cdot \omega_j}{\omega_i + \omega_j}$$
$$b = \xi \frac{2}{\omega_i + \omega_j}$$

where ξ is the damping ratio, ω_i and ω_j are the angular frequencies related to f_i and f_j

$$\omega_i = 2\pi \cdot f_i$$
$$\omega_j = 2\pi \cdot f_j.$$

For detailed instructions please see chapter ***5.3*** of the ***User's Manual***.

Adjusting these parameters strongly depends on the relevant frequencies of the model, the type and frequency of the excitation and the response of the structure. Due to the nature of the complex curve, for frequencies greater than $\mathbf{f_j}$, or smaller than $\mathbf{f_i}$, significantly higher damping can be expected.

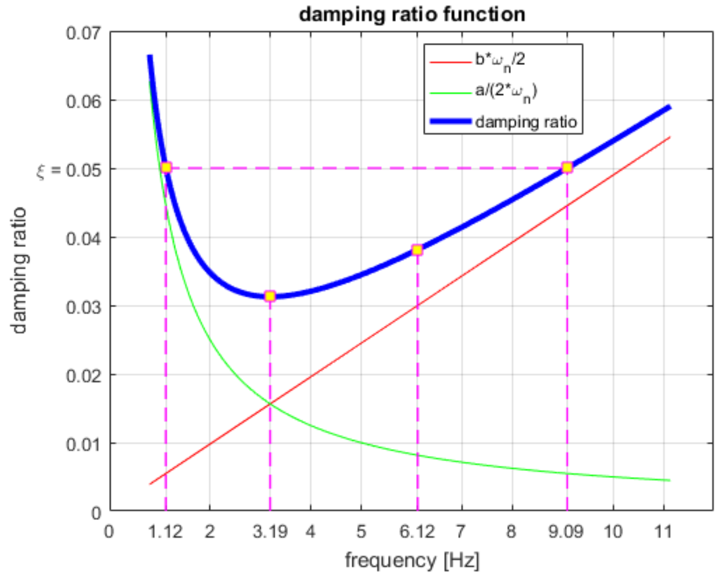
Therefore, its assignment is primary in the calculation and it requires experience in the field of structural dynamics.

Without further analysis, apply the next Rayleigh constants in the analysis:

In case of seismic, sine wave and shock load, set frequencies $\mathbf{f_i}$ and $\mathbf{f_j}$ related to the **1st** and **4th** frequency of the structure equal to **f_i =1.12 Hz** and **f_j=9.12 Hz**. The damping ratio should be **5%** (**x=0.05**). As a result of the above formulas, set constant as follows: **a=0,6265 [1/s]** and **b=0.0016 s**.

Considering machine induced vibration, set frequencies $\mathbf{f_i}$ and $\mathbf{f_j}$ related to the **1st** and **5th** frequency of the structure equal to **f_i =1.12 Hz** and **f_j=9.20 Hz**). The damping ratio is the same - **5%** (**x=0.05**). Set the constants **a=0.6273 [1/s]** and **b=0.0015 s**.

As an explanation, the function of the damping ratio can be seen in the next figure considering the results of the 1st vibration analysis:



In the figure, the damping ratios for the first four modes have also been marked. For the **1st** and **4th** frequency it is equal to the given ratio, but in case of the **2nd** and the **3rd** frequencies, lower values are expected (**3-4%**).

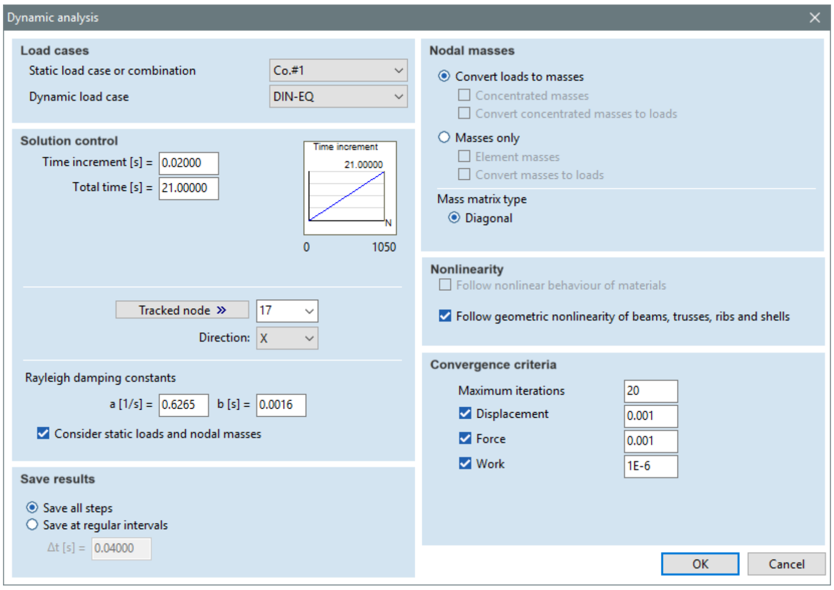
To ***Consider static loads and nodal*** masses in damping, the related function has to be switched on.

Saving results can be controlled: the user can save all the results in each step (***Save all steps***), or only in custom intervals (***Save at regular intervals***).
In preliminary testing of a large or nonlinear model, a greater interval can be specified to save considerable amounts of time. If the model is in the final stage, it can be refined.
Note: the interval of saving does not affect the numeric error. If the interval is increased, less 'insight' is expected into the calculated values, so only certain points of the function can be seen.

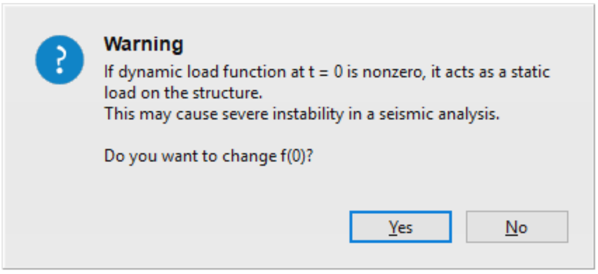
Check the ***Convert loads to masses*** and ***Follow geometric nonlinearity of beams, trusses, ribs and shells*** options.

Set the ***Maximum*** number of iterations to **20**, the other convergence criteria do not have to be changed.

Perform the analysis according to the above instructions. As an example, consider the following:



The ***Total time [s]*** should be set for each ***Dynamic load case***. This may exceed the duration of the excitation (this may be useful if the vibration of the structure must to be tested as well). Exceeding the total time, the software warns the user before the analysis starts, but it can be ignored:



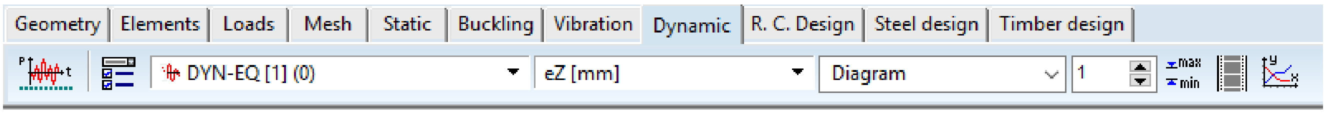
Our analyses have been completed considering the following time settings:

	Load function of the excitation		**Dynamic analysis**		
Dynamic load	**Δt [s]**	**total time [s]**	**time increment [s]**	**total time [s]**	**save results**
DYN-EQ	0.02	20.42	0.02	21.00	save all steps
DYN-SIN	0.01	10.00	0.01	10.00	save all steps
DYN-MACHINE	0.01	24.99	0.05	25.00	save all steps
DYN-SHOCK	defined by break-points	4.00	0.02	20.00	save all steps

In this demonstration example, if the capacity of your computer is limited, the total time can be limited or the time increment and saving intervals can be controlled. Note that increasing the time increment can have an influence on the accuracy of the results, the numeric error can increase.

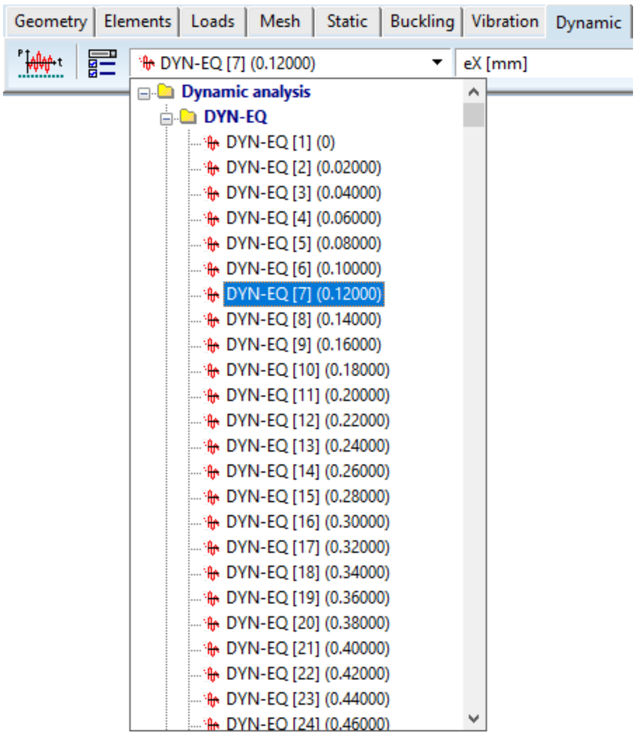
After the required settings, the calculation can be launched by clicking on the ***OK*** button. During the analysis, the software gives information about the current steps of analysis, the results of the tracked node (***aX***, ***vX*** and ***eX***) and the convergence are displayed on diagrams.

In the following, some examples are presented to show alternative methods for the evaluation of the results.



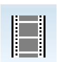
Results

During the analysis the results are calculated considering the set time interval, but the results that can be queried depend on the time interval of saving (see before setting the analysis). The results of each time step are put in separated cases – for example: **DYN-EQ [358] (7.14),** which contains the result of the **358th** time step (**7.14 s**) in the **DYN-EQ** load case.
The results, the internal forces of the model, can be queried in usual way, so the use of basic functions for result query are not presented in this example. In dynamic tests, the result components are supplemented with speed and acceleration data of the nodes.
A specific time step can be selected from the drop-down list by clicking on one of them or the keyboard arrows can be used to change results step by step. In the last case the displayed results are refreshed in the background.
Note: each dynamic time step contains the results of combined effect of the set static (here it is the **Co #1**) and the dynamic load cases.

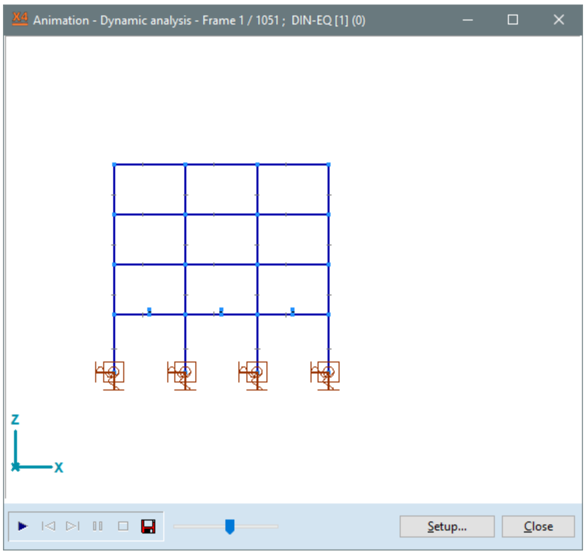


In this example, the ordinary methods to inquire results are not described, only a few typical functions are presented which can be efficiently used in dynamic analysis.

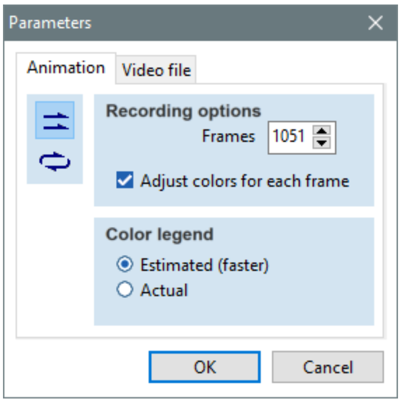
Amination



An animation can be made for spectacular presentation of dynamic results.
Select the load case **DYN-EQ** and the result component ***ex***. Set the scale of the ***Diagram*** to **4**, and align the model to the centre. Maximize its scale to fit the window, because the animation only shows the view that appears in the main window. Click on the ***Animation*** icon.



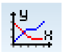
The video parameters can be set by clicking on the ***Setups*** icon in the lower right corner (e.g. playback type: ***unidirectional play*** / ***bi-directional play***, number of ***Frames***, ***Frames duration***, etc...)



To maximize the resolution of the animation, maximize the size of the ***Animation*** window. Generation of the phases can be started by clicking on the ***Play*** icon in the bottom left corner. If the video frames have been completed, the animation can be saved in **AVI** video or animated **GIF** file format. (A presentation video based on the results of load case **DYN-EQ** is available in the pack of the example.)
With the slider next to the ***Video file*** (save) icon, the speed of the animation can be adjusted.

If the result display is also switched on, results are also shown on the animated frames.

Diagram display



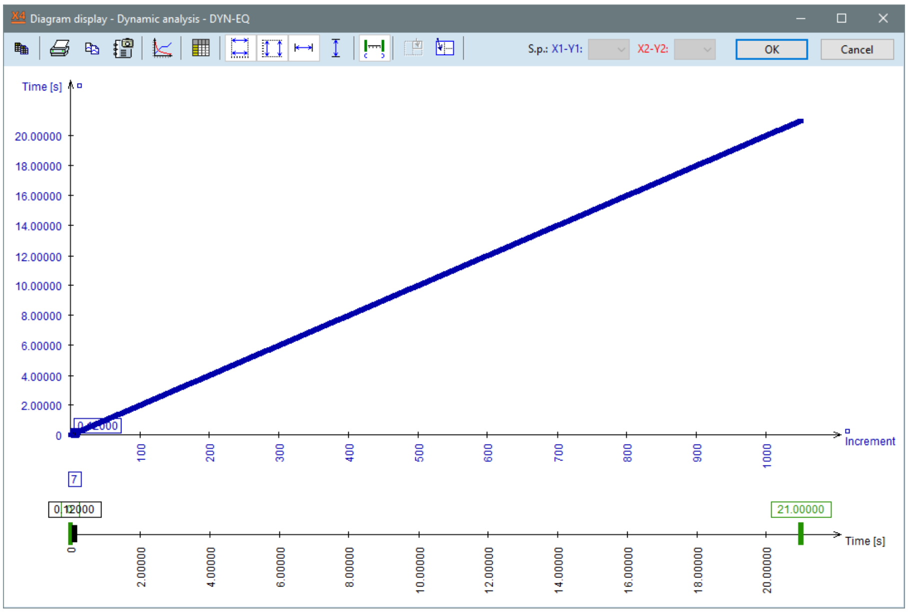
Creating diagrams can be helpful in evaluating dynamic results. The results can be displayed as a function of multiple component (result components, time, increment, iteration steps, etc...), and two types of graph can be presented in a graph at the same time.

Let us look at some examples of using this feature.

Firstly, check the acceleration of the supports as a function of time in the earthquake type load case. The result must match the set acceleration function.

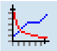
From the drop-down list, select load case **DYN-EQ [1] (0)** (although this is the first load step in the series, the whole time history diagram will be shown on the diagram).

Click on the ***Diagram Display*** icon, and the following window pops up:

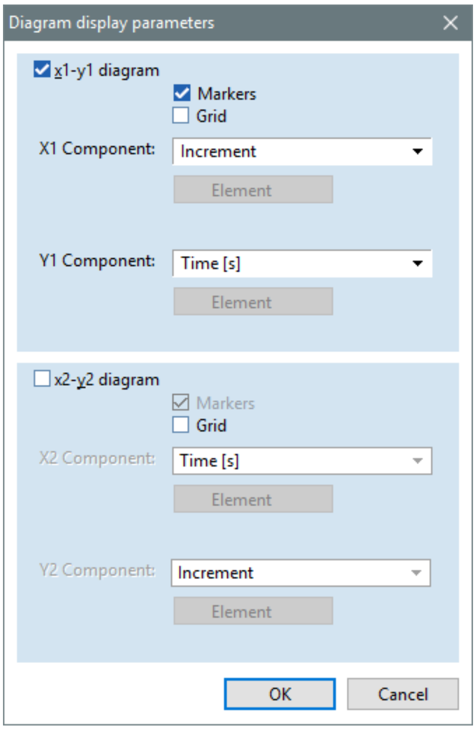


By default, the ***Increment-Time [s]*** curve can be seen in the diagram.

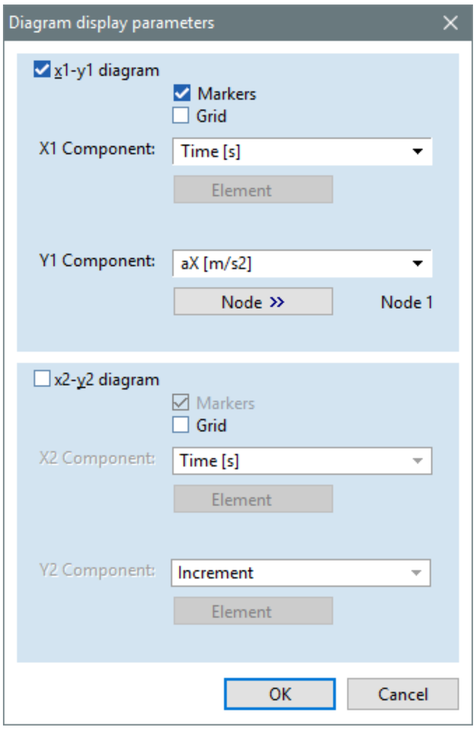
Diagram parameters



Next, set the desired type of function to be displayed. Click on the ***Diagram parameters*** icon, and the following window will appear:

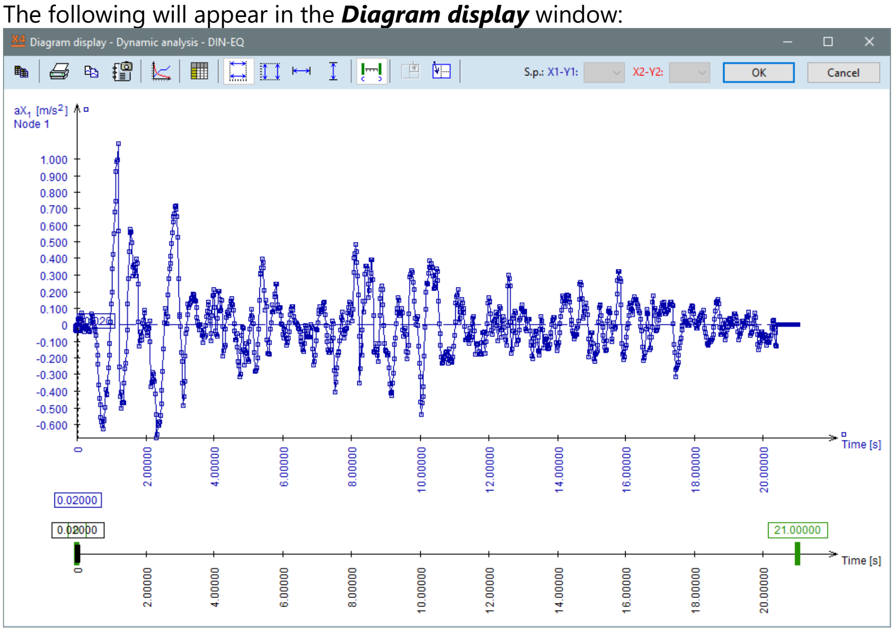


The user can specify two different diagrams to be shown in one graph (the result components on the ***x*** and ***y*** axes can be also different).
First, let's display a single curve, the acceleration curve of node **1** (support) as a function of time. Take the time on the horizontal axis, select ***Time [s]*** from the drop-down list of ***Component X1***. At the list of the ***Y1 component***, look for ***aX [m/s2]***, which is the horizontal, global ***X-directional*** acceleration. Click on the ***Node*** button, and in the main window select node ***1***.

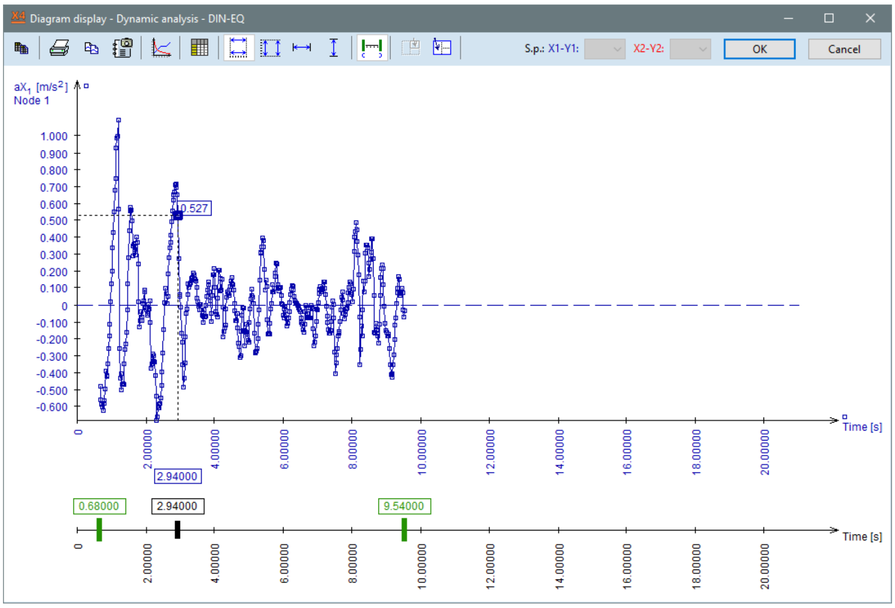


Confirm changes by clicking on ***OK***.

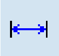
The following will appear in the ***Diagram display*** window:



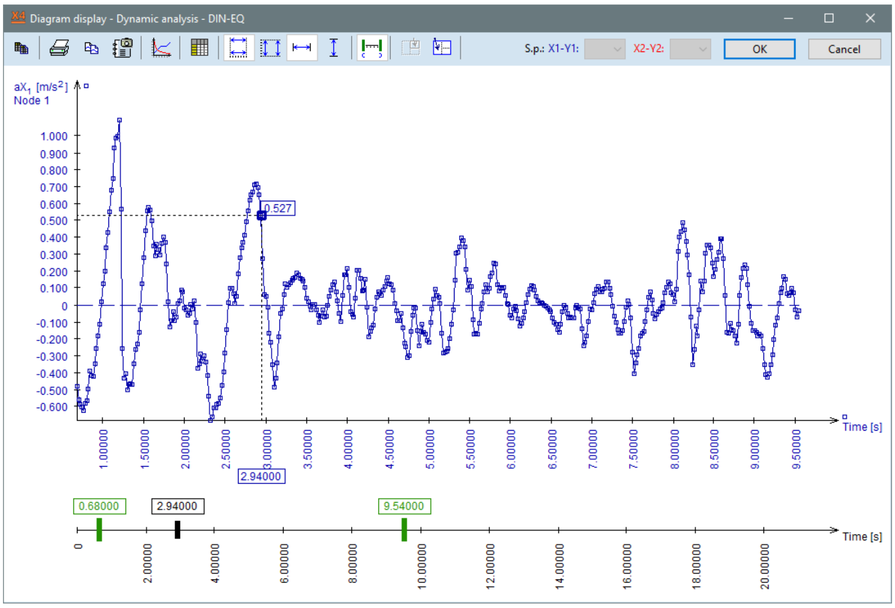
Under the graph, there is a scale which represents the ***Time*** axis and it contains green and black markers at the ends. Moving the black marker along the axis, the current result value will be shown on the diagram according to its position. Moving the green ones allows the range of the displayed function to be set. The next figure shows the effect of moving the markers.



Fit in view in X direction

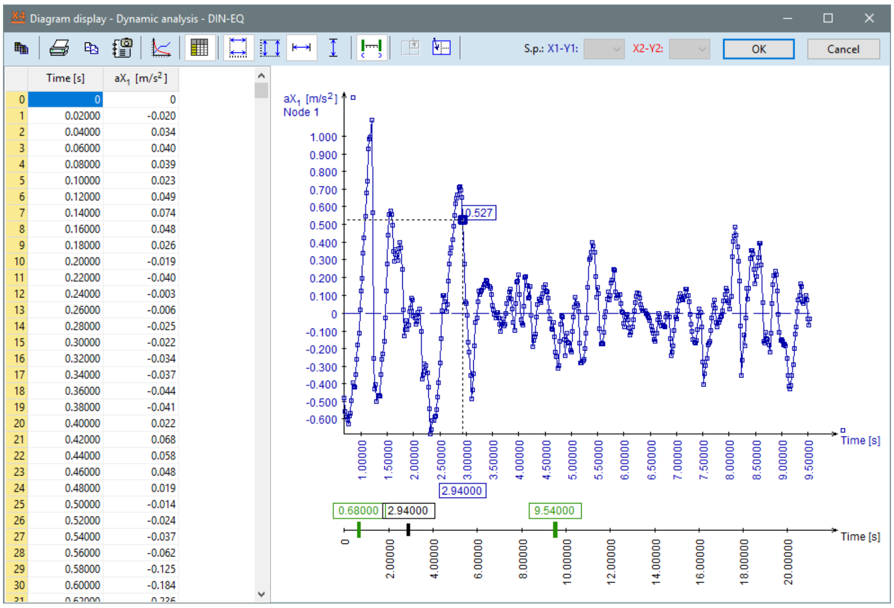


If the range of the diagram has been limited by the green markers, then the graph can be stretched by clicking on the ***Fit in view in X direction*** icon.



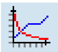
Table

The results of the calculated time steps can be presented in a table. Click on the ***Table*** icon, and a table appears next to the graph on the left side, which contains the whole data. If it is necessary, these cells can be copied to the clipboard for further processing (using Excel, or saving the data file, etc...).

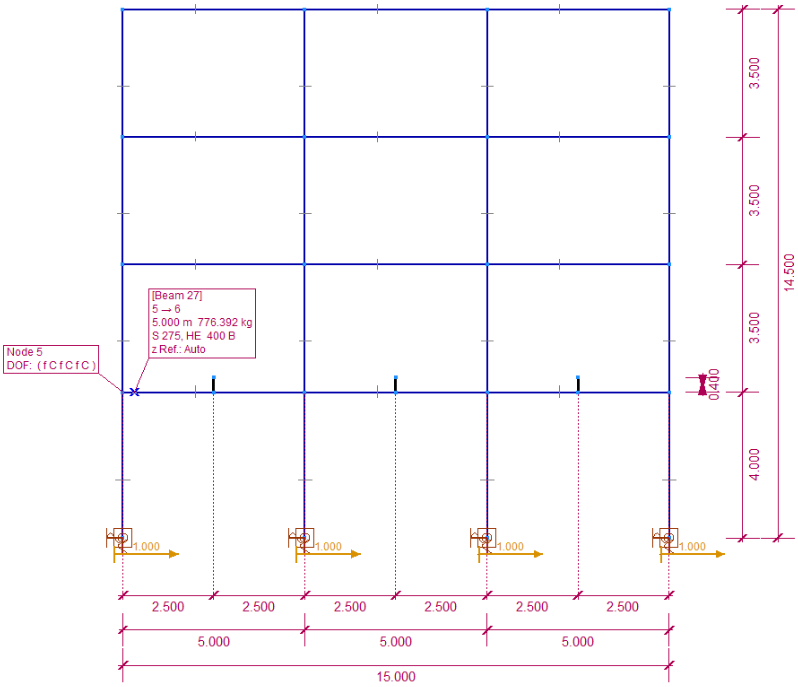


By clicking on a cell in the table, the result label jumps to the selected time step, then the black marker on the bottom axis turns grey (it will be active again when moving this marker to a specific point).

Diagram parameters

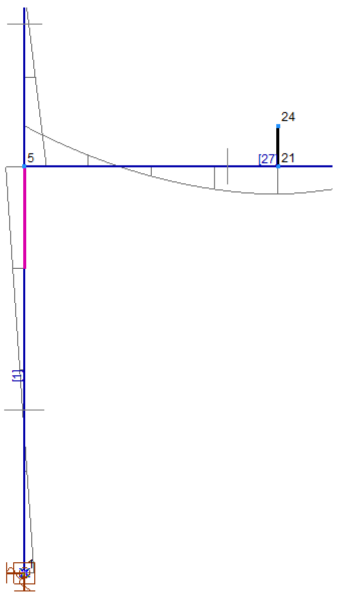


Now, let's show the ***My*** flexural moment of beam no. **27** in node **5** as a function of time. For reference, the specific beam and its end node are marked in the next figure.

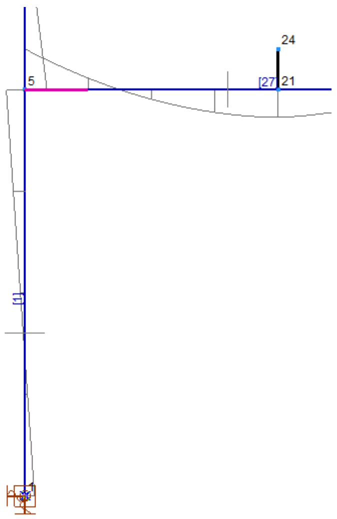


Click on the ***Diagram parameters*** icon.

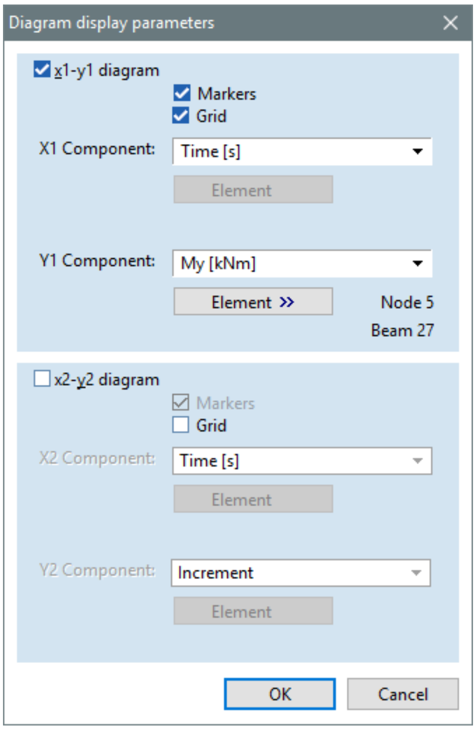
On the ***x1-y1 diagram***, select the ***Time [s]*** for the ***X1 component***, and select the ***My [kNm]*** result component for ***Y1***. Next, select the correspondent node (**5**) and beam (**27**) by following the instructions below:
Click on the ***Element*** button, then the main window becomes active.
With one click select node **5**, then one of the beams which belongs to the selected node will be highlighted automatically in purple. The following status will be shown on the screen:



The software has automatically selected beam **1** (the beam is divided into finite elements, the program highlighted only the upper finite element), however we need to select beam **27**. By pressing the ***Tab*** key on the keyboard, one can switch between the beams (that have the same end node). Press the ***Tab*** key repeatedly to select the correct element (in our case just press it once to highlight beam **27**). As a result, the following selection is obtained:



Click on node **5** again to close the selection, and the ***Diagram display parameters*** window will appear again:



Click ***OK*** to accept the changes, and the required curve is presented in the ***Diagram display*** window. Drag the green markers to the ends (0-21 s) to display the whole diagram.

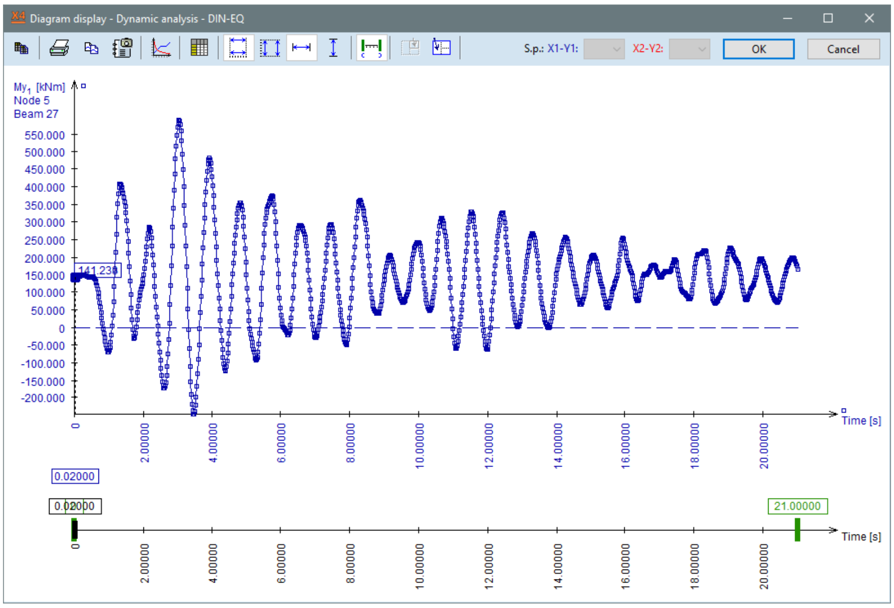
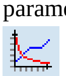
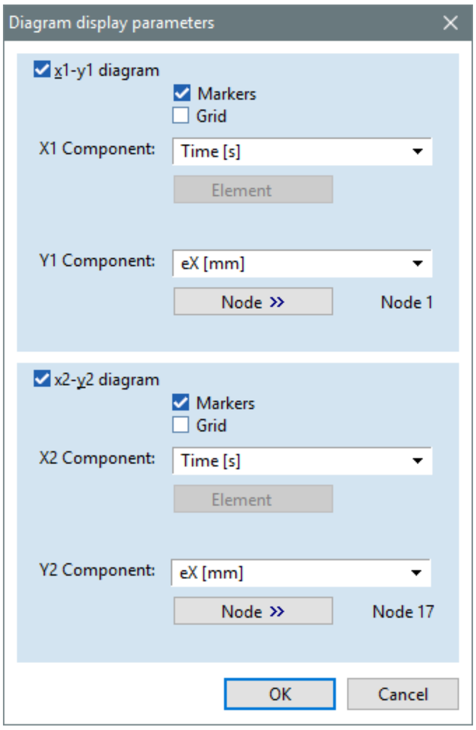


Diagram parameters

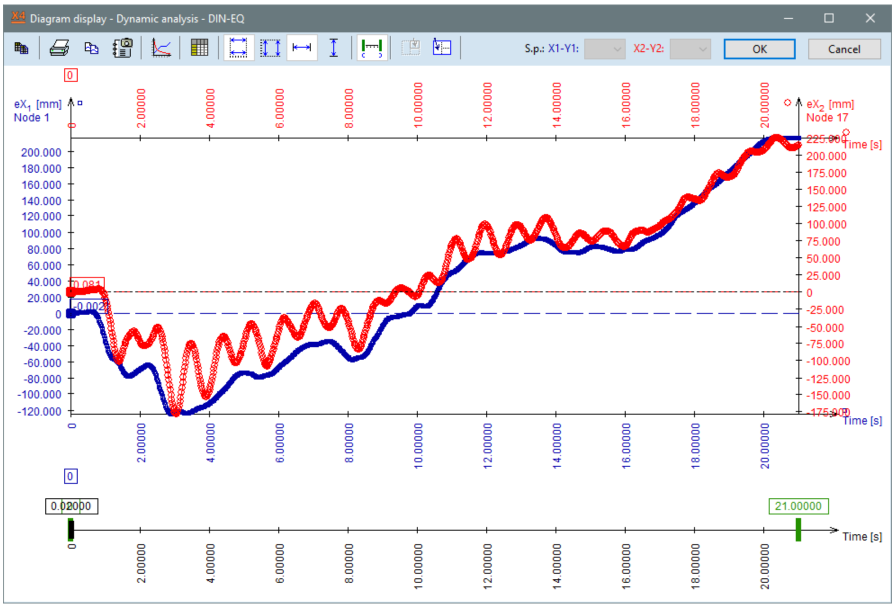


Show the displacement of node **1** (lower, support node) and node **17** (top level) as a function of time on the same diagram to see the relative displacement between nodes (or levels).
Click on the ***Diagram parameters*** icon.
On the ***x1-y1 diagram*** select the ***Time [s]*** for the ***X1 component***, and for ***Y1*** select the ***eX [mm]*** result component and select node **1**.
Activate the ***x2-y2 diagram*** with its checkbox. At the ***X2 component*** the default setting ***Time [s]*** is suitable for our purposes, but for the ***Y2 component*** select the ***eX [mm]*** component. Finally, select node **17**.
Finally, the next result can be seen in the ***Diagram display parameters*** window:



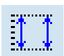
Click on ***OK*** to close the window.

In the ***Diagram display*** window turn off the ***Table***, if necessary drag the green markers to the end positions (if only a range of the diagram is visible) to see the whole diagram:

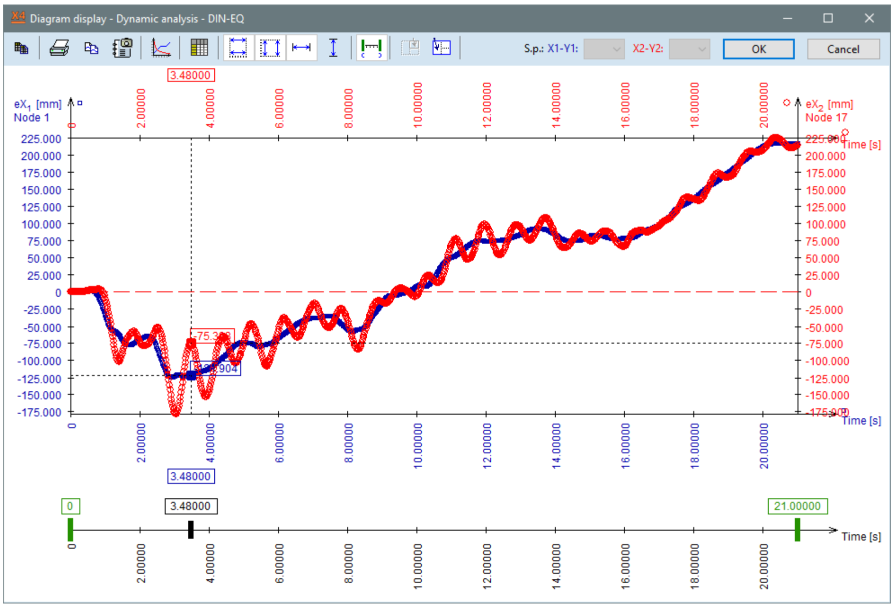


The blue curve shows the displacement of node **1**, the red curve belongs to node **17**. The colour of the curves and their labels give information about the type of the curve.

Same range on the two Y axes



It can be seen on the figure that the program used different scales on the vertical axis (see the blue and red vertical axis). The same scale can be set by clicking on the ***Same range on the two Y axes*** icon.



On the bottom axis, move the black marker to a desired position to show the displacements values of the nodes. The relative displacement can be calculated by taking the difference between the results.
If the ***Table*** function is opened, then the whole data of the diagrams can be easily read and compared. If it is necessary, the data can be copied and evaluated in any other spreadsheet program (e.g. ***Excel***).
For more details about the using of the special features of the current window, please see the ***User's Manual***. Close the ***Diagram display*** window by clicking on the ***OK*** button.

Diagram display

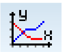
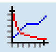
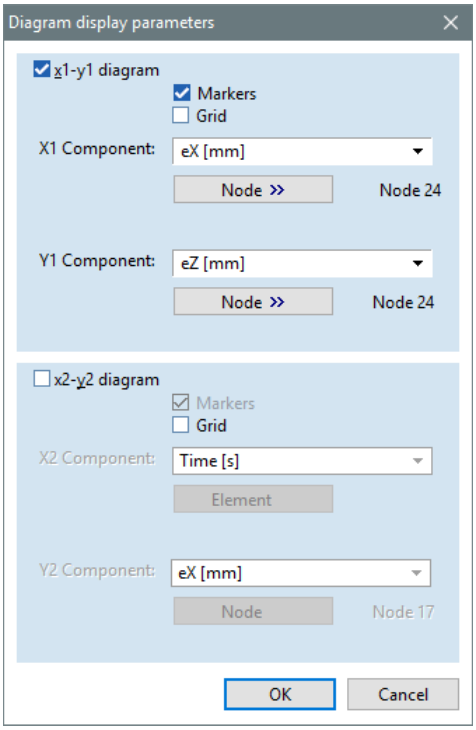


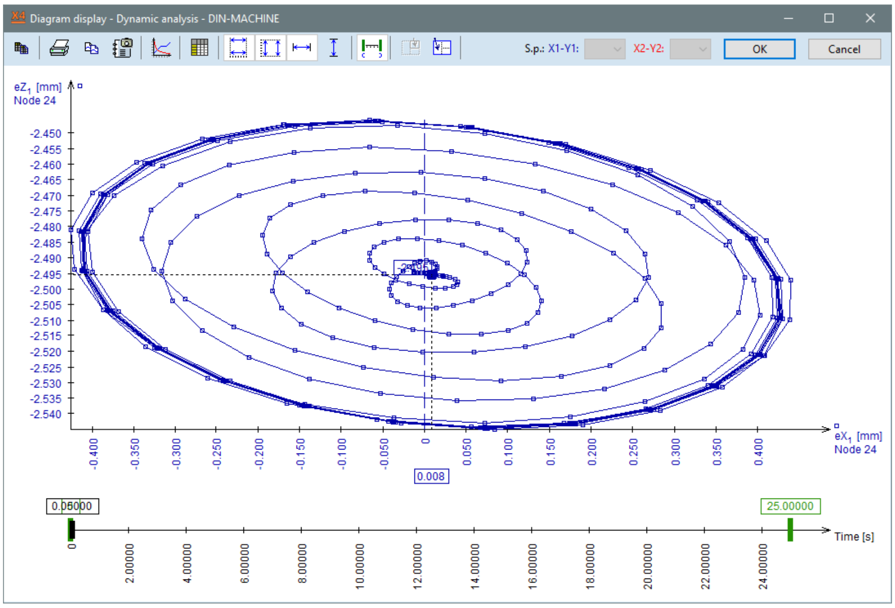
Diagram parameters



Next, follow the motion of one of the excited nodes in the **DYN-MACHINE** load case. In the main window change the load case to the ***DYN-MACHINE [1] (0)***.
Click on the ***Diagram display*** icon, and in the window that appears click on the ***Diagram parameters*** icon.
Only the ***x1-y1 diagram*** is needed, therefore uncheck ***x2-y2 diagram*** (and their parameters will become disabled). By finishing the settings, the following can be seen in the parameters window:

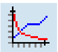


Click ***OK*** to see the result diagram:



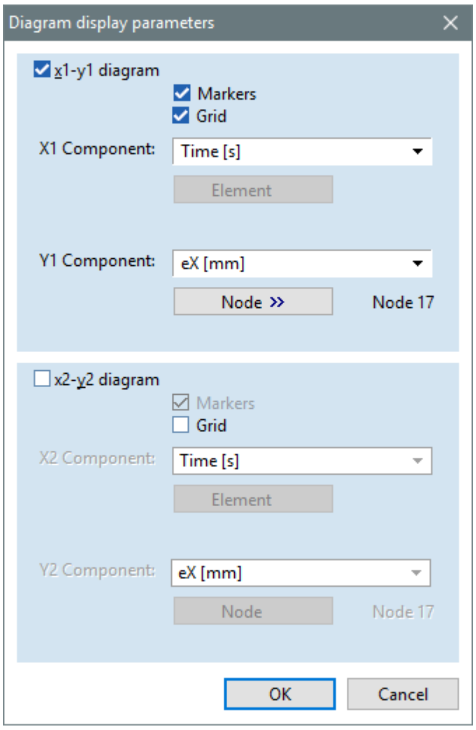
Due to the excitation and movement of the structure, a quasi-steady movement similar to an ellipse is formed. Close the ***Diagram display*** window by clicking ***OK***.

Diagram parameters



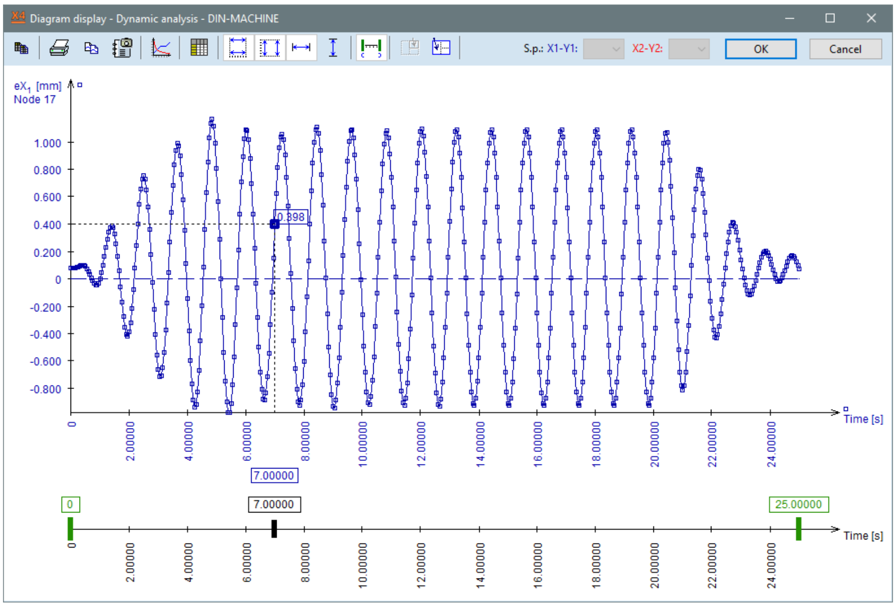
Analyse the displacement of node **17** in a diagram.

Click on the ***Diagram display parameters*** icon and set the following:

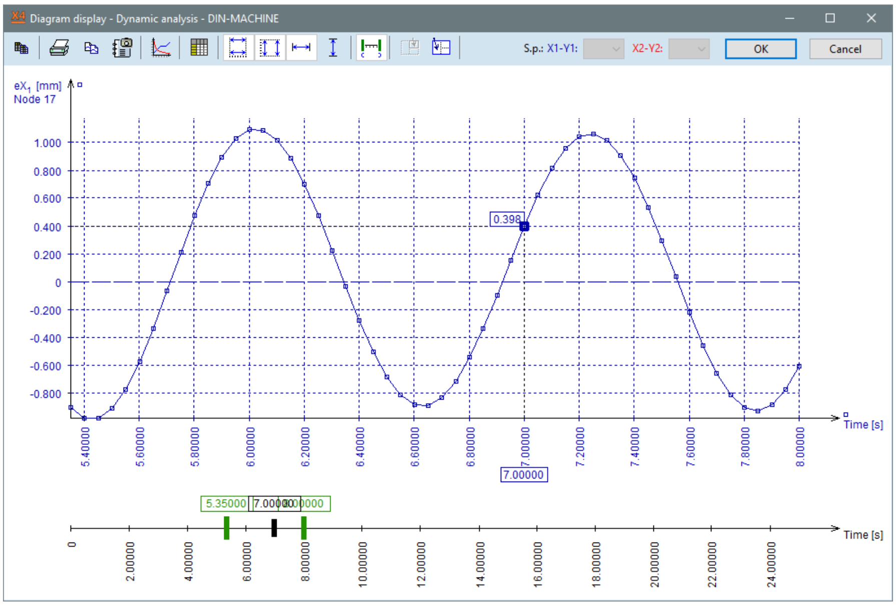


Close the window with ***OK***.

The following result can be seen displaying the whole range of the diagram:



Limit the range of the presented curve to see two internal waves as shown below. Adjust the markers to the following limits: **5.35 - 8.00 s**.



In this example, a quasi-harmonic motion is induced by the excitation. If the time increment or the density of the saved steps is insufficient, then the curve will be 'piecewise', and the accuracy of solution will not be adequate.

Take a look back, the load function was defined in 0.01 s time steps, but the calculation was performed and results were saved considering 0.05 s time steps. The appropriateness of the applied time step can be checked by analysing the result curves by counting the intermediate points of a 'wave'. (It is our experience that a clear harmonic wave - e.g. sine / cosine - must contain at least 10 intermediate points to obtain acceptable results. For more complex / precipitous curves/excitations, it is worth testing the effect of different time intervals to find an adequate setting).

Below is a result of a deliberately wrong adjustment. The calculation was performed with a time increment of **0.2 s**. There is significant error in the minimum and maximum values, and the shape of the curves clearly indicates it.

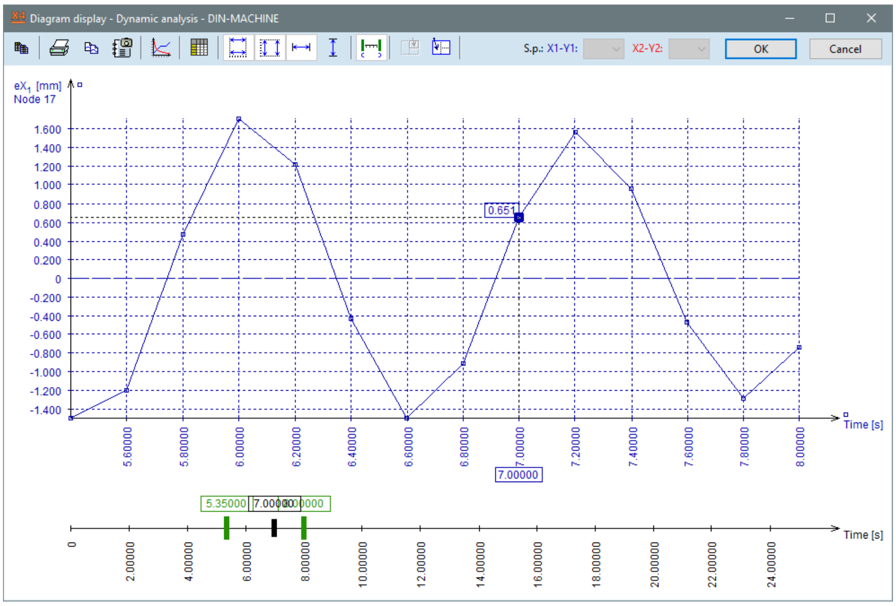
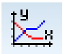
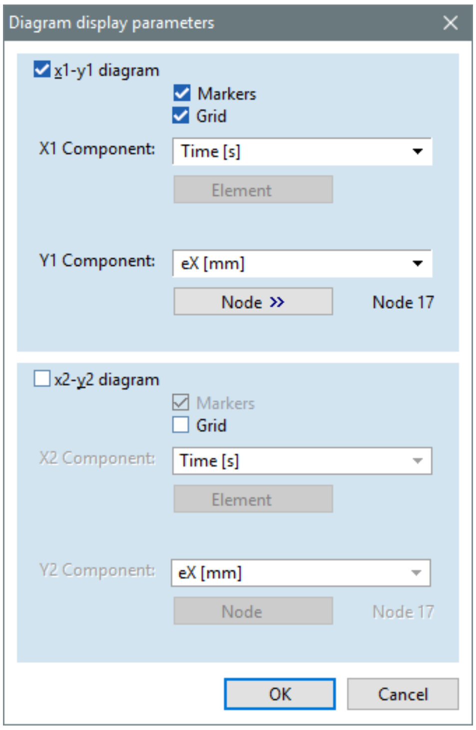


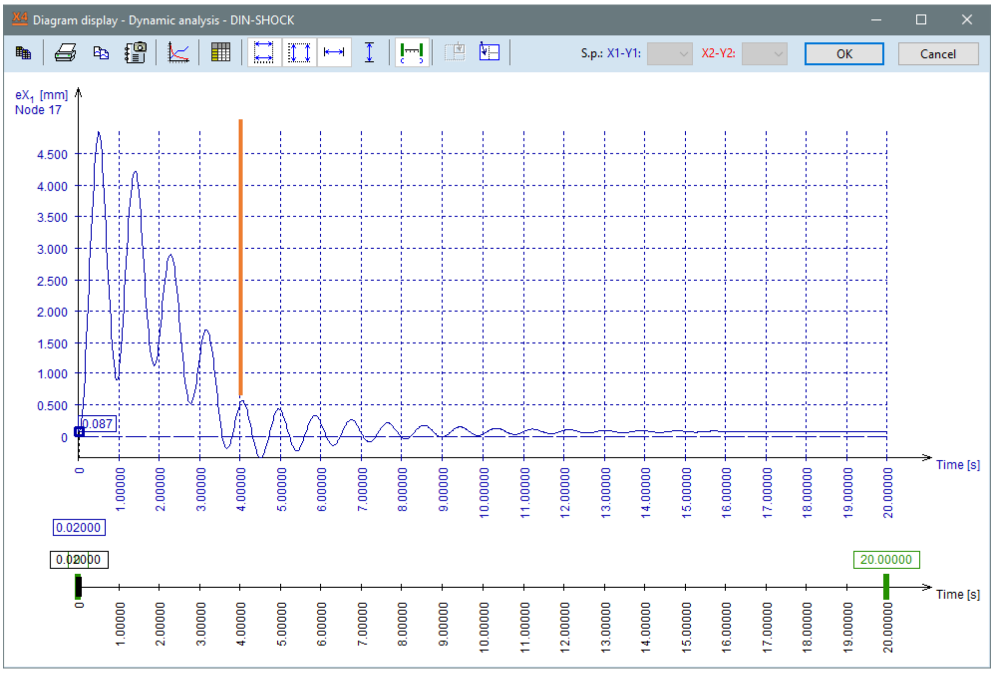
Diagram display



Finally, let us analyse the damping effect of the frame examining in the **DYN-SHOCK** load case (the duration of the shock effect is **4 s**, but the total time of the performed analysis is **20 s**). Select the **DYN-SHOCK [1] (0)** load case and return to the ***Diagram display parameters*** window. Only the ***x1-y1 diagram*** has to be used, the other has to be switched off. Set the displacement of node **17** to display as a function of time as follow:



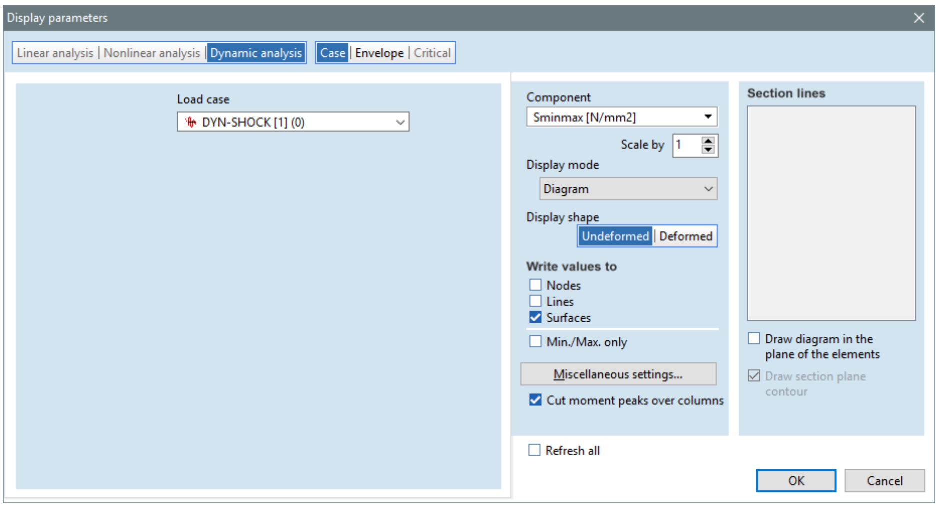
Uncheck the ***Markers*** option, but the ***Grid*** should be displayed. Close the window with ***OK***, and the following will be the result diagram:



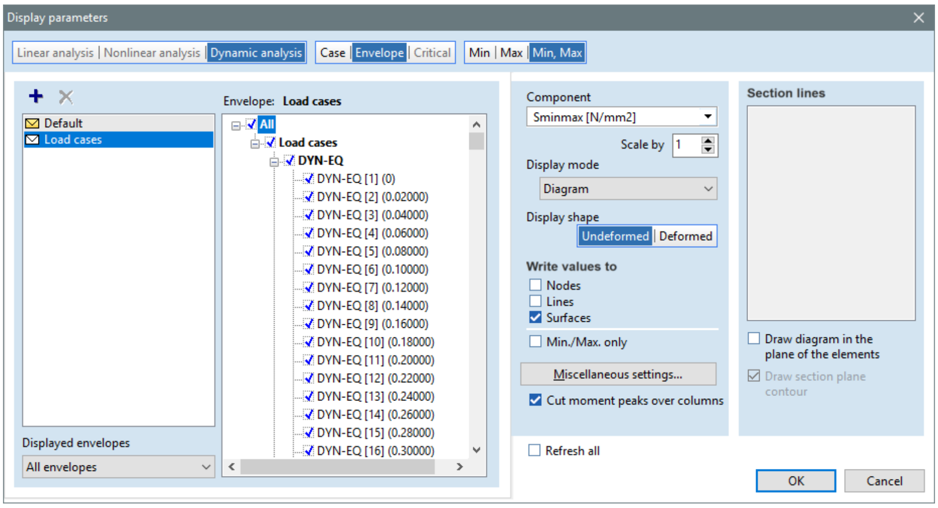
In the figure above, the orange vertical line (it was not created in **AxisVM**) indicates the time when the dynamic nodal load becomes zero and from where the free vibration of the frame starts. The calm down tendency of the waves points to the set damping behaviour of the structure.

Result display parameters

In many cases, it is needed to evaluate the results of various dynamic loads together using envelopes. Present ***Sminmax*** results are an envelope of the results of **DYN-MACHINE** and **DYN-SHOCK** load cases. From the drop-down list of results, select ***Sminmax*** under the ***Beam stresses*** components.
Then, a new custom envelope has to be created. Click on the ***Result display parameters*** icon, and the ***Result display*** window pops up:

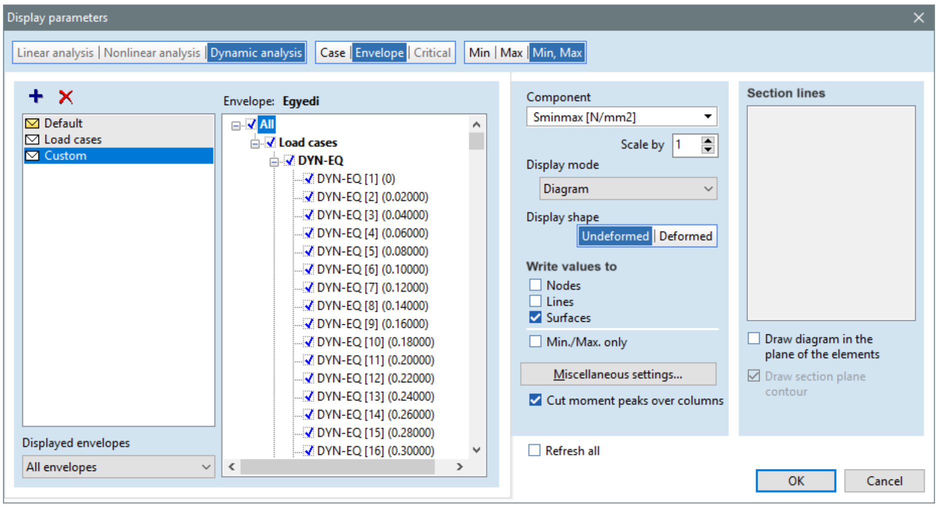


In the top line, click on the ***Envelope*** function.

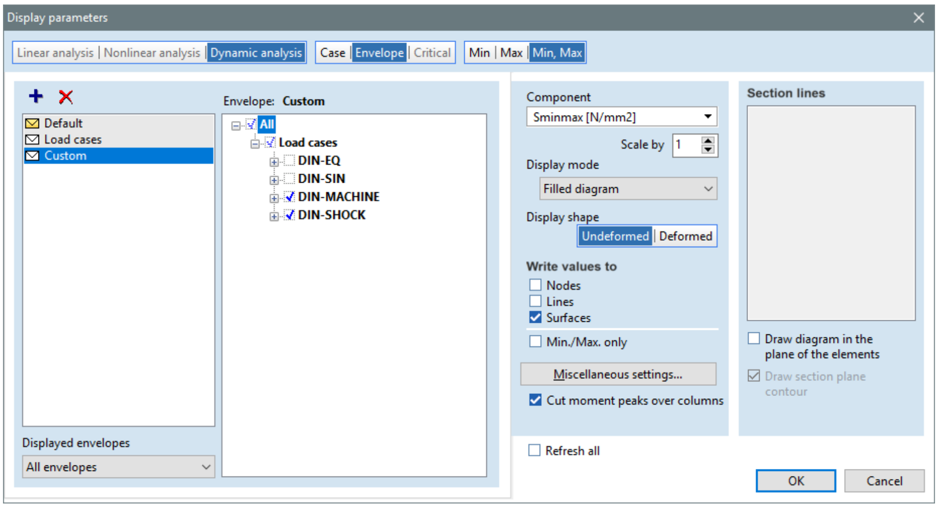


New envelope set

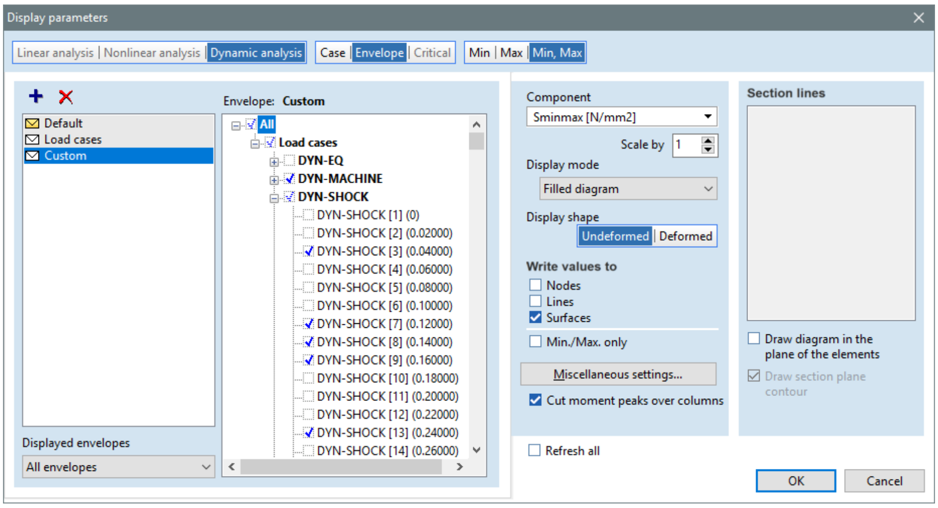
Click on the ***New envelope set*** icon above the left white field. As a result, a new ***envelope set*** will be created, which appears in the list under the icon. Click on its name and rename it as '**Custom**'.



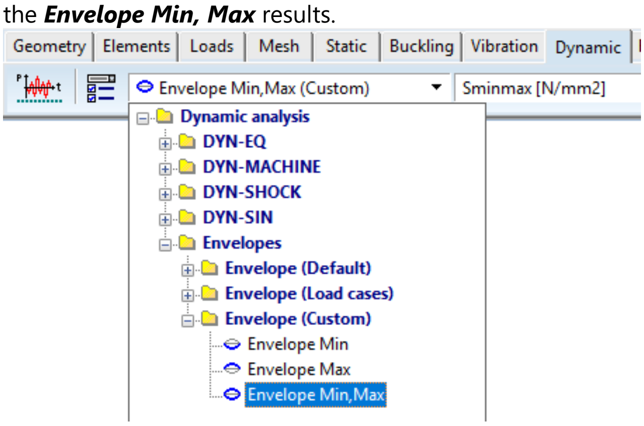
By default, the new envelope will contain all available load groups and cases. Modify the set by the following: in the list named ***Envelope***, only the **DYN-MACHINE** and **DYN-SHOCK** load groups must be kept as checked, the others have to be unchecked. If it is necessary, the load groups can be folded by clicking on the '**–**' sign before the name of the group to hide their time steps.
Finally, change the ***Display mode*** to ***Filled Diagram***.



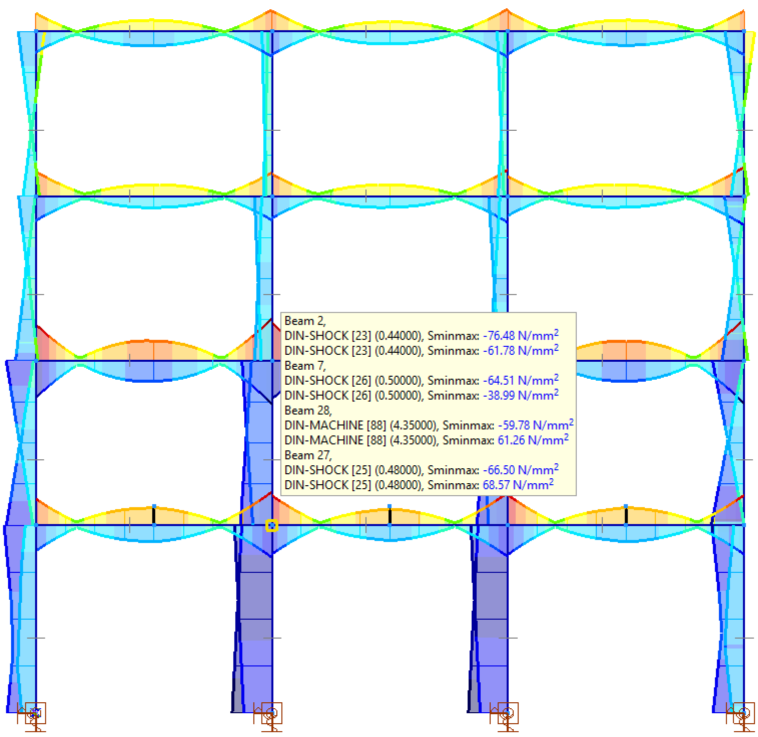
Confirm results by clicking on ***OK***.
Note: each time step in the dynamic load groups can be checked or unchecked individually when an envelope set is created. As an example, see the following figure:



In the main window, find the new envelope – ***Custom*** - in the drop-down list and select the display of the ***Envelope Min, Max*** results.

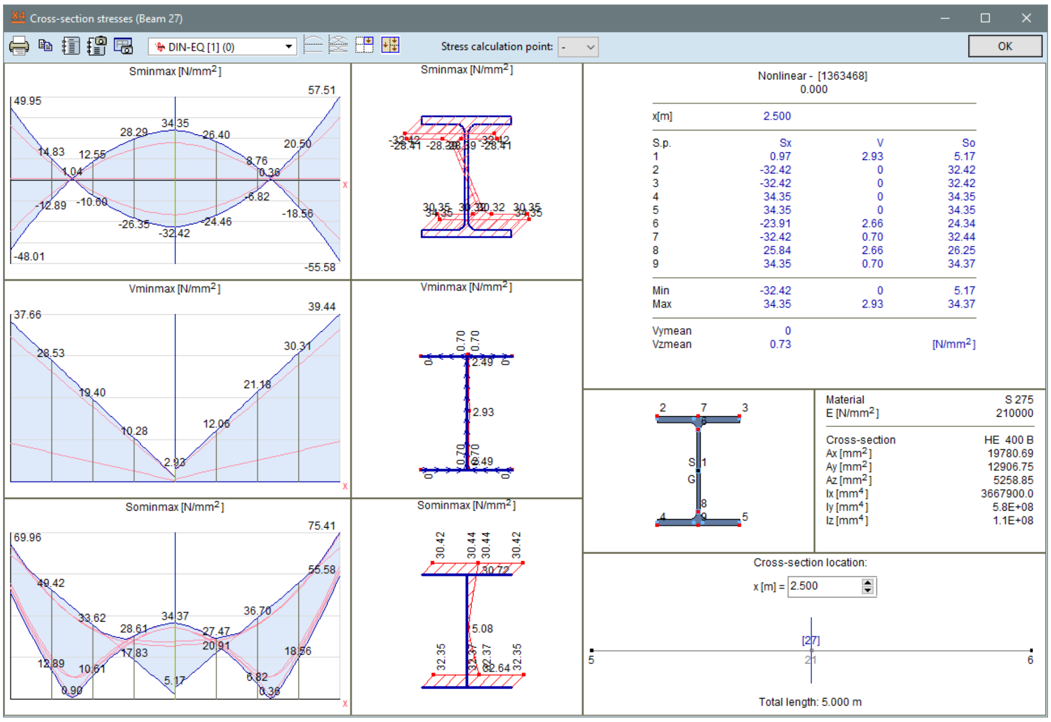


By querying any result component, the envelope result of the set load groups / load cases can be seen in the window.



Cross section stresses

Display the ***Cross-section beam stresses*** (***Sminmax***, ***Vminmax***, ***Sominmax***) in beam **27** by selecting the ***DYN-EQ*** load case. In the main window select the ***DYN-EQ [1] (0)*** load case and click once on the beams, then the following result window will appear:

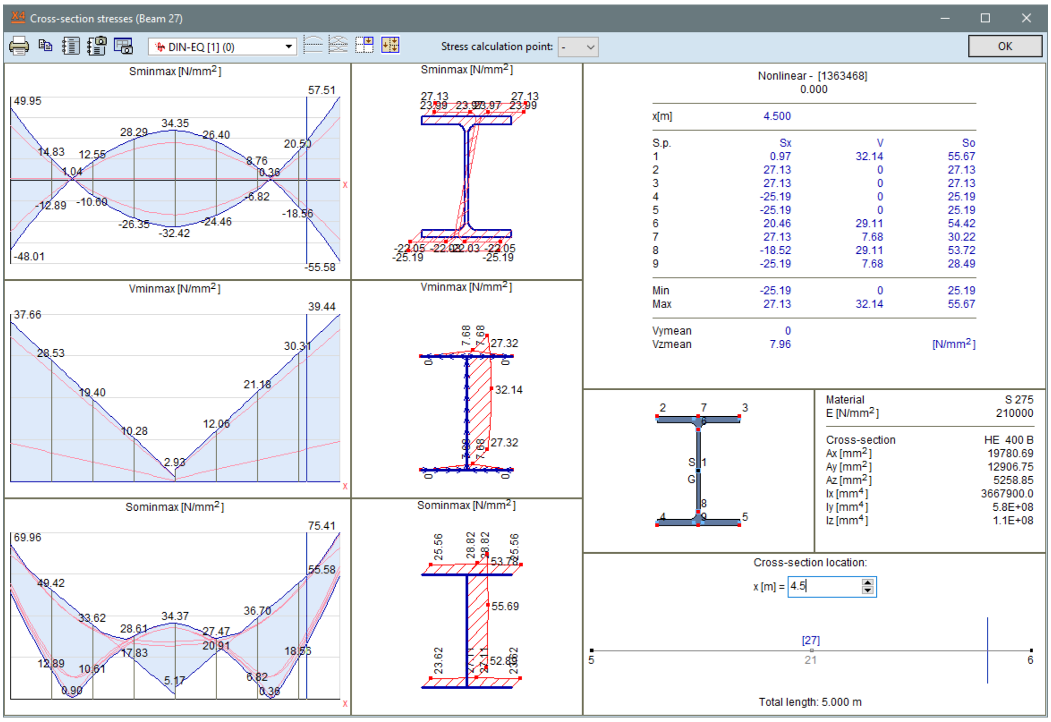


The figures in the left hand shows the stress results along the axis of the beam considering the current time step.

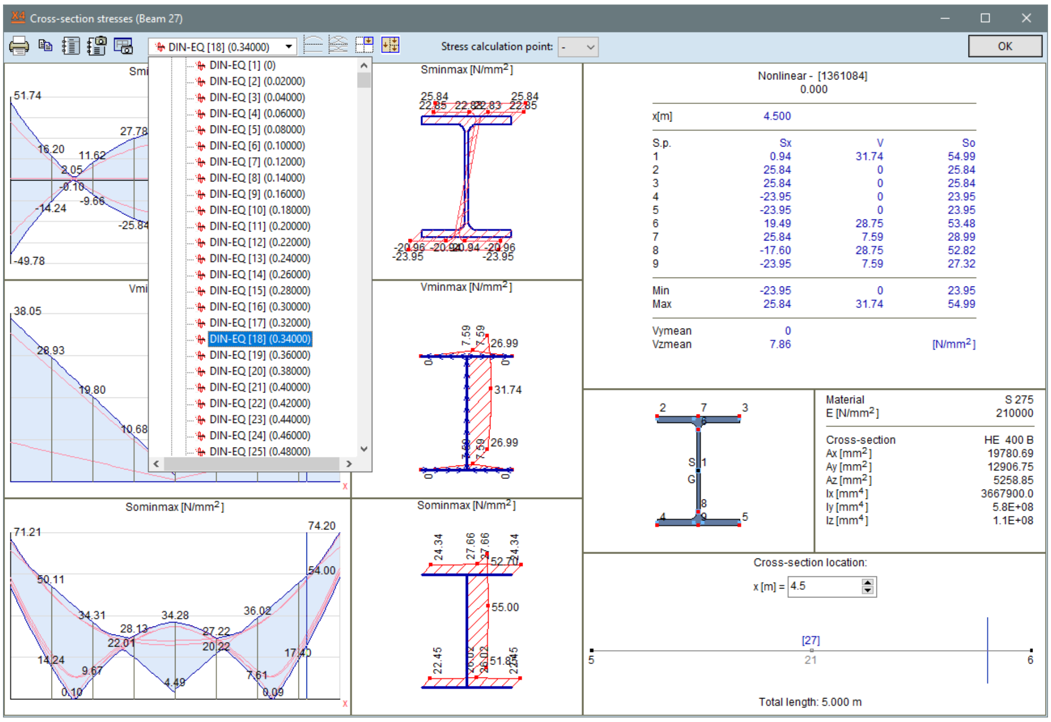
In the middle lane, the different stress diagrams of the selected cross-section and their main values are shown. In the table displayed in the upper right corner, the stresses in the calculated points can be read (these points are marked on the cross section under the table, their position can be edited or modified in the cross-section editor). Presently, the results of the cross section in the half of the beam are illustrated.

The stress results of any cross section along the beam can be queried by dragging the vertical axis of the figures to the desired position. In the lower right corner, a schematic axis can be seen which also shows the actual position (***x[m]=2.500***), the length of the element and the node number of the end-points. The current position can be controlled by overwriting the current value or dragging the vertical axis to the right position. The latter function is more convenient for taking a specific position.

Display the result of the cross-section at position **4.50** m (measured from the startng point – node **5** – of the beam). Overwrite the position value in the lower right window. The following result can be seen in the window after adjustment:



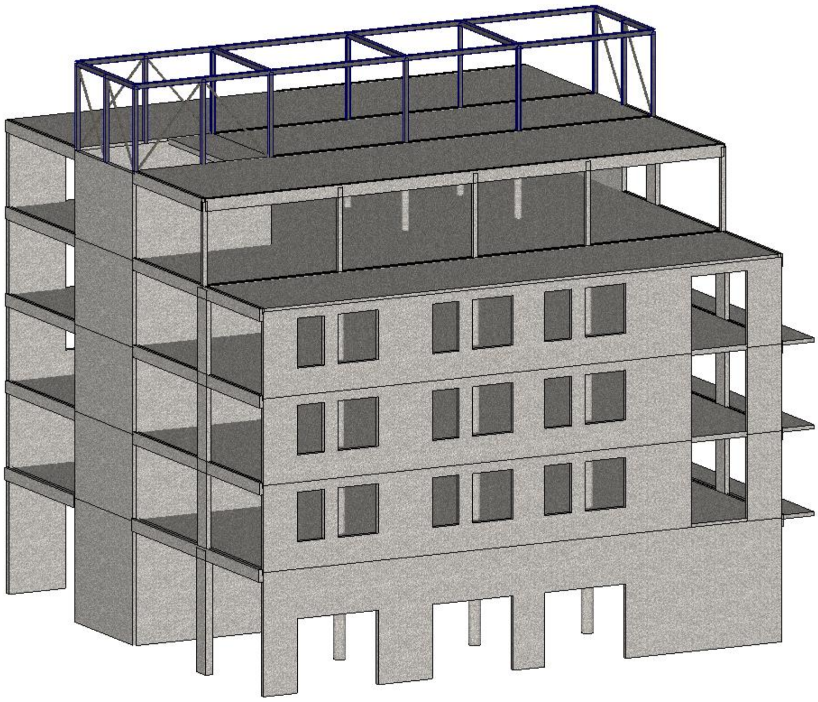
The user can switch between the results of each time step in the drop-down list by clicking on a specific step or by navigating up or down with the arrows keys. In the latter case, the results will automatically be updated in the background:



2. SEISMIC ANALYSIS OF A MULTI STOREY REINFORCED CONCRETE BUILDING USING MODAL RESPONSE SPECTRUM

Objective

Perform the seismic test of the following reinforced concrete (RC) building using modal response spectrum analysis. The basic finite element model file of the building is enclosed to this manual (**MRSA_of_RC_building_0.axs**).



The earthquake test is carried out according to the basic ***EN 1998-1:2008*** independently of any national annex (in real projects the user has the choice to select that code which contains the specific rules for the current country). In our task, the so-called ***M***odal ***R***esponse ***S***pectrum ***A***nalysis (***MRSA***) is used: the earthquake loads are generated according to the standard based on the dominant modal shapes of the structure. The distribution of the seismic load is calculated by the magnitude of the oscillation and the distribution of the masses in each modal shape. After, the seismic results of each modal shape are determined by a linear static analysis. Finally, the (critical) design results (internal forces and displacements) are obtained by summing up the results of the shapes using standard methods (e.g. ***SRSS***, ***CQC***).
Note: this kind of seismic analysis supposes linear behaviour and uses the principle of superposition, therefore nonlinear material or finite elements cannot be considered in ***MRSA***.

Based on the seismic characteristics of the location of the building, a ***reference peak ground acceleration*** **0.10 g** has to be considered applying ***type I response spectrum*** and ***soil class C***. Take the ***behaviour factor*** to **1.50**.

Description of the finite element model and loads

In the task the creation of the basic finite element model will not be described, the model can be easily built up applying the knowledge of our previous examples. Primarily, the main steps of applying ***MRSA*** method will be detailed using the available tools of the program. The starting file (**MRSA_of_RC_building_0.axs**) is enclosed to the example, which contains the followings:

The full geometric and finite element model of the multi storey, reinforced concrete (***RC***) building without the generated seismic loads and results.

The building has a ground floor and 5 storeys, and its main function is office. The characteristic structural dimensions of the building are the following: the overall plan size is **23 x 22 m** and the total height is **22.5 m**. The main structure is built as monolithic reinforced concrete structure (monolithic walls, columns and flat slab), but above the 4th floor there is a steel-structured mechanical unit.
The rigidity of the reinforced concrete structure is ensured by RC walls, while the steel structure on the roof is braced by steel struts.

The thickness of the RC walls is **200 mm**, the flat slabs are **220 mm**. The section of the RC columns varies according to the internal forces. The intermediate columns are made with circle shapes, while the columns behind the facades have a rectangular cross section.
The columns of the steel structure are **HE 160A**, the beams are made of **HE 200A**. The bracing elements are: **N80x80x4,5 mm**.

The structural roof of the installation unit is not modelled in detail, it is assumed that works as a rigid plate in the horizontal plane. According to this assumption, the roof is modelled with a rigid diaphragm. The loads of the roof are placed on ***load panel***s which transfer the loads to the structural elements above it.
Note: the structural role of the top steel roof is subordinate in the main behaviour of the entire building, which makes it possible to make simplifications and to present the using of rigid diaphragm element in our example. Simplifications can lead to a more perspicuous model, but in all cases the user should consider all the assumptions.

In the model, the basic design code called ***Eurocode*** is applied (see the ***Design codes…*** window in ***AxisVM***). The RC structures is made of concrete with a strength of **C30/37**, the material quality of the steel profiles is **S235** uniformly.

Setting the nodal degrees of freedom, every node is considered as free.

The supports on the ground floor are fixed, their stiffnesses are the following: ***Rx=Ry=Rz=1E+6 kN/m***, and ***Rxx=Ryy =7.5E+6 kNm/rad*** (using global coordinate system). The stiffnesses of the walls are defined relative to the edge using the next values: ***Rx=Ry=Rz=5E+5 kN/m/m*** and ***Rxx= 5E+5 kNm/rad/m***.
Note: the result of the earthquake design is significantly affected by the stiffness of the supports, but this effect is not examined in our example. The stiffness of the supports can be estimated based on the soil physics characteristics assuming a dynamic effect (the stiffness loads may be different in case of static loads).

For the analysis, the main load cases have already been created and been separated into load groups. The given loads are the so-called characteristic values. These values are not specifically described here, please have look at the model.
In earthquake analysis, it is not necessary to adapt partial loading. Usually, the total load has to be considered using the specified seismic combination factors (probably the total load will be the critical, hence this provides the greatest mass). The model does not contain meteorological loads (wind and snow) because they are not simultaneous ($\boldsymbol{\psi_2=0}$) with the seismic load.

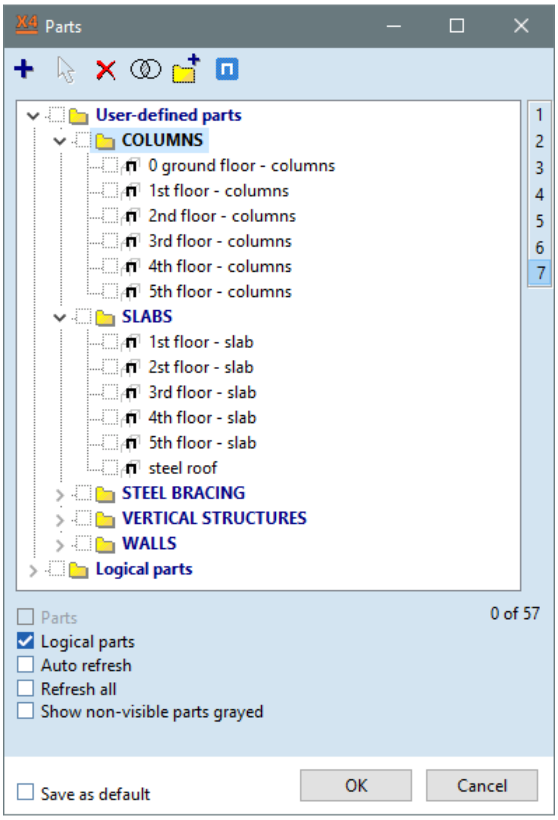
For the vibration analysis a ***custom load combination*** is still needed which It is not included in the starting model. This combination will be defined later.

The seismic loads and cases will be automatically generated based on the set basic earthquake parameters and the main modal shapes considered.

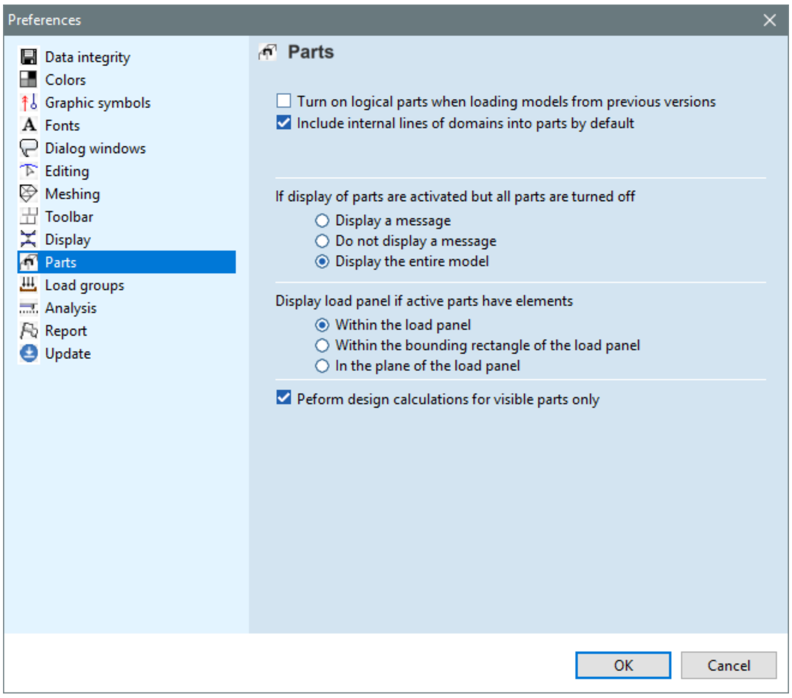
The critical minimum and maximum results of the total analysis will be determined automatically considering the set load groups and combination factors, therefore no need to define any more custom load combinations for this purpose.

Parts in the model

For the better visibility and manageability, parts have been defined considering structural types (walls, columns, slabs related to different storeys) in the model, which have been sorted into various directories as seen below:



Tricks *If parts are used in displaying and the function **Perform design calculations for visible parts only** is checked in the menu of **Settings/Preferences/Parts**, then the derivative results will only be calculated (e.g. calculation of necessary reinforcement in RC elements) for the visible elements, not for the entire model. This option can save time in processing the results of large models, where the complete calculation of the derivative results may require a substantial amount of time.*

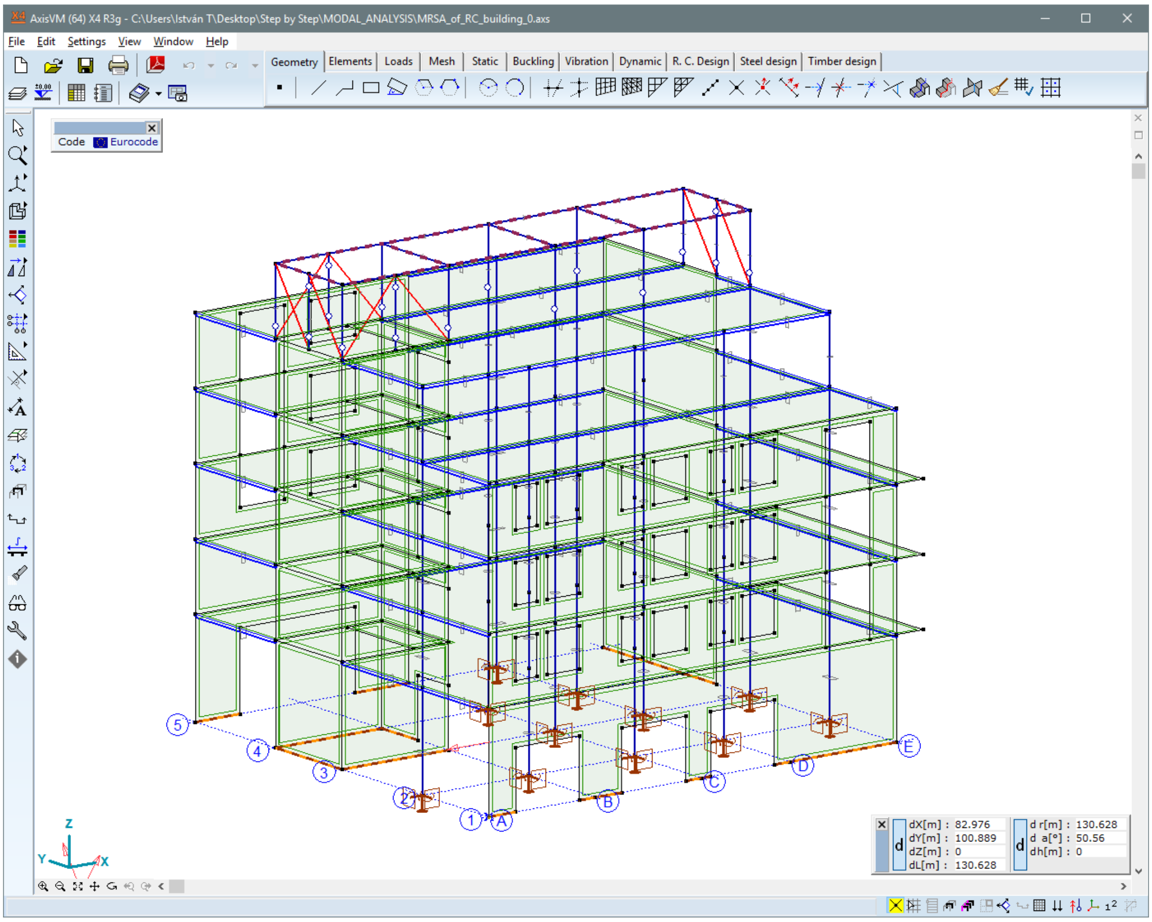


Start

Start ***AxisVMX4*** by double-clicking the ***AxisVMX4*** icon in its installation folder found in the ***Start*** – ***Programs*** menu.

Open

Click on the ***Open*** icon to load the starting file saved on your computer. In the window that appears, choose the directory containing the file and select it (**MRSA_of_RC_building_0.axs**). Click on the ***Open*** icon to load the model. After, the model of the building appears in the main window:



First steps

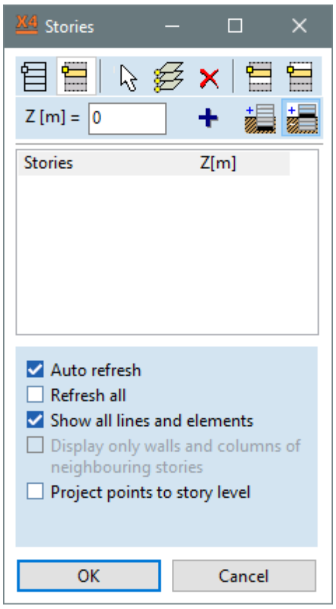
Save the model under a different name (if necessary, we can return to the original model later). In the ***File menu***, look for the ***Save as...*** function and click on it, then rename the model: **MRSA_of_RC_building_0.axs**.

Before starting the seismic analysis, check the model, its geometry, the created finite elements and the settings listed above.

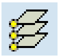
Stories

In the first steps let us define stories (main levels of the building), which can have several roles during the analysis. Stories can be displayed separately, they behave like the ***Parts*** and give help in constructing the model and ensures better visibility.
Stories have also a role in defining the torsional effect of the seismic action because the eccentric forces can be assigned to these levels. The latter function will be presented later.
For more benefits and opportunities of this feature please see the ***User's Manual***.

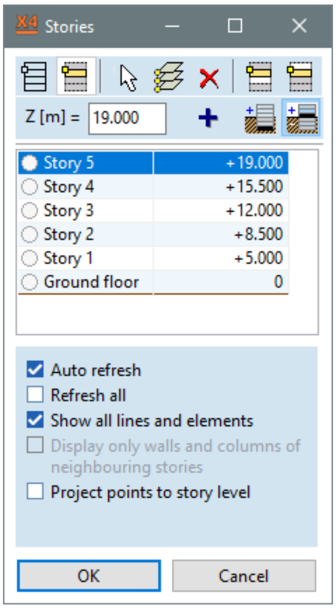
Clicking on the ***Stories*** icon on the left-hand vertical toolbar, the following window pops up:



Find

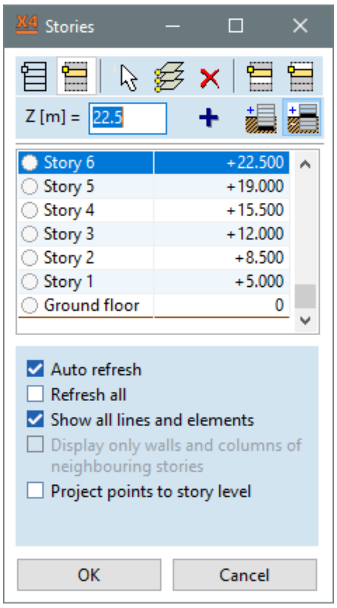


Stories can be specified individually, entering the height value of each storey, but auto search function can also be used. Click on the ***Find*** icon, then the function will search for the ***horizontal slabs*** with different levels. The next will be the result:

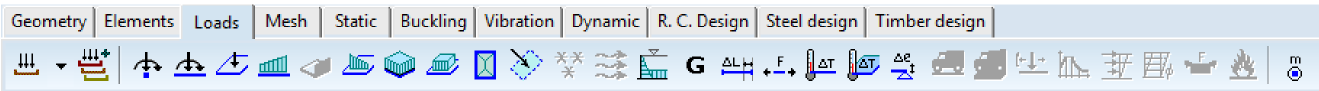


Add

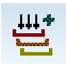
The level of the steel roof (**+22.50 m**) has not been recognized by the program, because the roof was modelled by diaphragm, not with slab. Add this level to the list as well. Enter the height data (**22.50 m**) in the ***Z [m]*** = input field, and then click on the ***Add*** icon:



The generation of stories has been completed with this final step, click on ***OK*** to close the form.



Load combinations



In ***MRSA*** the seismic load is defined based on the mode shapes. Therefore, a vibration analysis has to be done using a load combination, which follows the seismic combination rules specified in the standard (the masses for the vibration analysis are obtained by converting the loads of that combination). In the software, this combination should be defined as '***custom***'. Complete the followings to create this combination.

New row

Change tab to the ***Loads*** and click on the ***Load combinations*** icon. In the appearing window click on the ***New row*** icon, then the software creates a new row in the database. Enter the following name into the first cell: **SEISMIC**. Set the type of the combination to „***- (user-defined combination)***", finally specify the combination factors to the load cases according to general formula:

$$\sum_{j\geq 1} G_{k,j} \text{ "+" } \sum_{i\geq 1} \varphi \cdot \Psi_{2,i} \cdot Q_{k,i}.$$

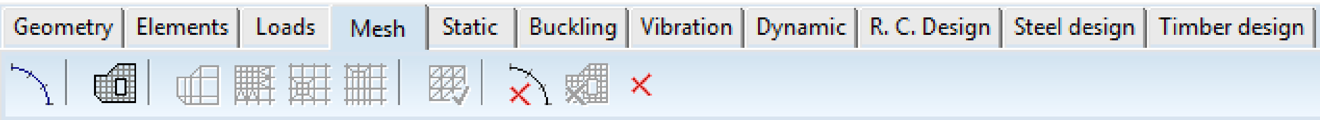
Permanent loads take part in the combination with their characteristic value. Variable loads must be considered by their quasi-permanent value. The factor of φ is usually **1.00** (for the benefit of security).
Note: this combination is only valid for the vibration analysis which precedes the generation of seismic loads. Therefore, it contains only permanent and variable actions, the accidental seismic load is not included in it.

The following table summarizes the combination factors to be entered:

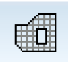
Name of the load case	combination factor
self-weight	1.00
finishes	1.00
susp-ceiling	1.00
facade	1.00
staircase-dead-load	1.00
1st-floor-office	0.30
2nd-floor-office	0.30
3rd-floor-office	0.30
4th-floor-office	0.30
1st-floor-balcony	0.30
2nd-floor-balcony	0.30

3rd-floor-balcony	0.30
4th-floor-terrace	0.30
5th-floor-flat-roof	0.30
1st-floor-div-wall	1.00
2nd-floor-div-wall	1.00
3rd-floor- div-wall	1.00
4th-floor- div-wall	1.00
5th-floor-installation	1.00
6th-floor-steel-roof	0.00
staircase-var-load	0.30

Entering the data above, confirm changing and close the window with clicking on the ***OK*** button.



Domain meshing

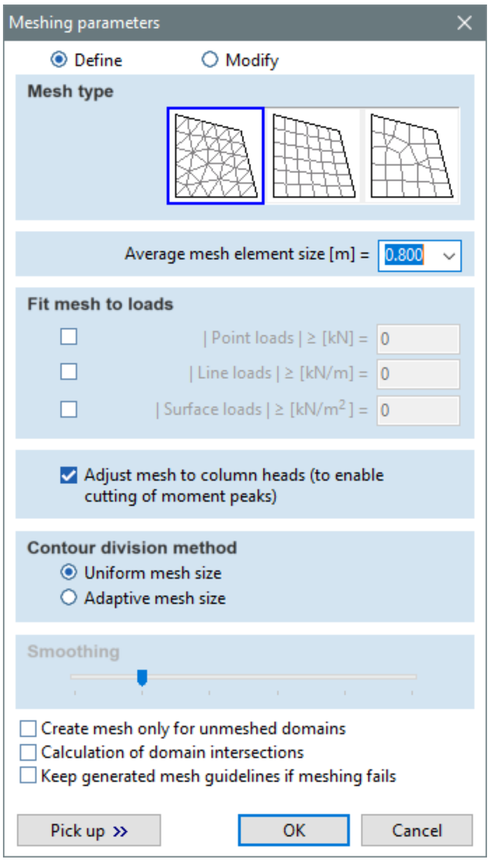


Before the dynamic (and nonlinear) analysis, domains and line elements must be meshed. In our task, it is sufficient to divide only the domains, not necessary to mesh the columns, because the significant part of the total mass is concentrated on the stories and the bending mode shapes of the columns are not dominant in the behaviour of the entire structure.
Note: if a line element is not meshed, then the self-weight or any loads on the element (which will be converted to mass) are distributed to the end nodes in the vibration analysis.

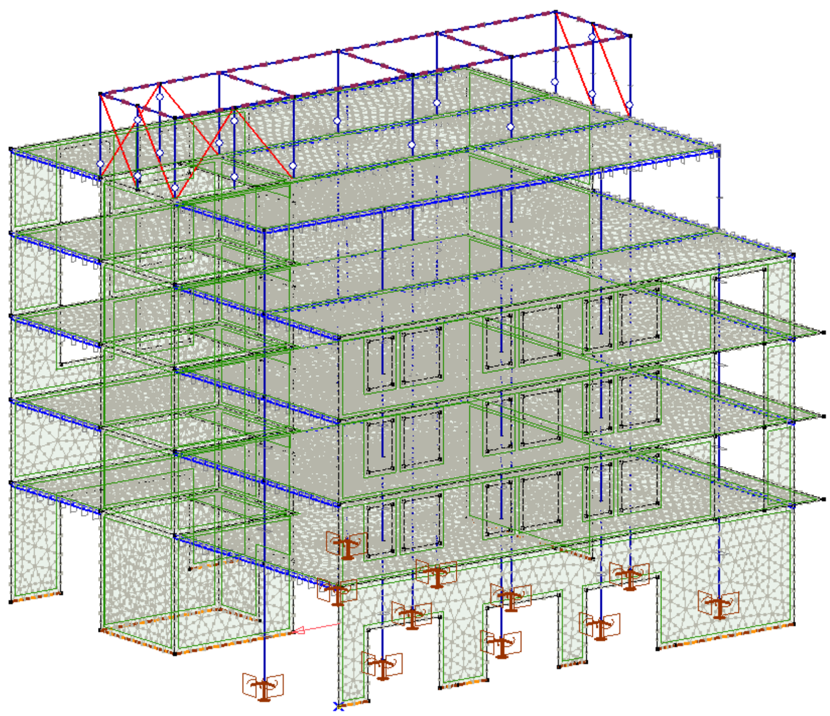
Depending on the size and type of the building, various size and density of mesh can be applied.
In earthquake analysis, it may be appropriate to provide the vertical stiffening (wall) structures with a dense mesh, while less dense mesh can be used in the slabs.
The dynamic behaviour of the structure is primarily determined by the stiffness and behaviour of bracing wall systems not by that of the slabs. In many cases, the slab can be regarded as rigid diaphragms if its stiffness is great enough (see the relevant parts of the standard) and the vertical effect of the earthquake is negligible compared to the static loads.
Optimization of the applied mesh size can reduce the number of finite elements, which can save time in computing and evaluating the results.

In our model, we only ***use a single mesh*** because applying various mesh size does not result significant savings due to the small size of the building.

Change tab to the ***Mesh*** and click on the ***Domain meshing*** icon and select the entire model with ***** (***All***) icon.



In the form, select the T***riangle mesh***, use ***Average mesh element size*** - **0,8 m**, check the ***Adjust mesh to column heads (to enable cutting of moment peaks)*** and apply ***Uniform mesh size***. Start the meshing procedure by clicking on the ***OK*** button.



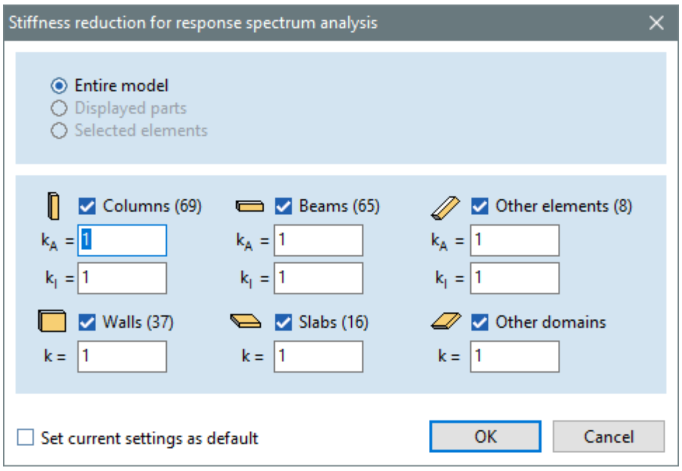
Mesh display on/off

When the meshing procedure is finished, check the regularity of the mesh while rotating the model. Among the ***Speed buttons*** the display of mesh can be turned on or off by clicking on the ***Mesh display on/off*** icon.

Stiffness reduction…

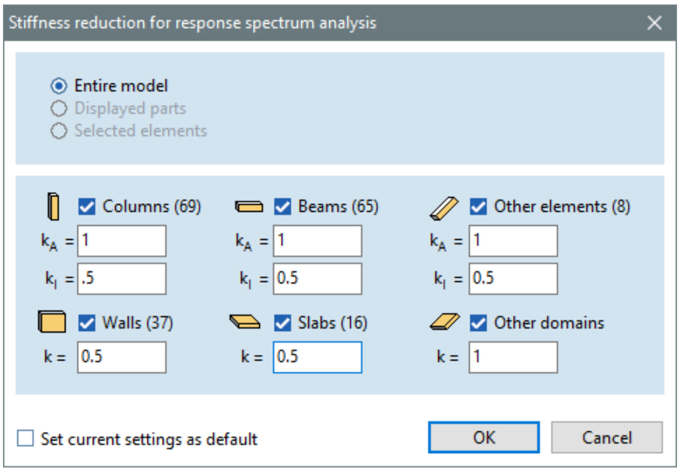
In case of RC structures, the effect of cracking can be considered by using the ***Stiffness reduction***... option (***EN 1998-1:2008*** recommends 50% stiffness reduction to RC elements). This reduction is considered in the vibration and static analysis as well.

The ***Stiffness reduction for response spectrum analysis*** form can be found in the ***Settings*** menu or it can be set among the ***Vibration*** analysis parameters. Adjust the reduction parameters now, before changing tab to the ***Vibration***. Click on the ***Settings*** menu and in the list find and activate ***Stiffness reduction...*** function. As a result, the following window pops up:

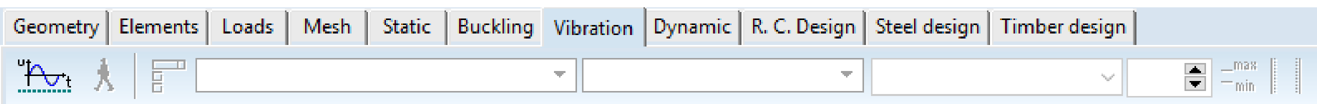


In the form, the basic stiffness reduction parameters can be set for the ***Displayed parts***, the ***Entire model*** or the ***Selected elements***. If the entire building is displayed in the main window and there are no selected items, then the ***Displayed parts*** and ***Selected elements*** checkboxes are inactive, as shown in the figure above.

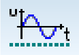
In case of ***Columns***, ***Beams*** and ***Other elements*** (beam), two options are offered: the compressive and tensile stiffness (***A***) and the inertia (***I***) can also be reduced independently from each other.
For ***Walls***, ***Slabs*** and ***Other domains***, only one parameter can be specified, which reduce also the bending, shear and membrane stiffness of the finite elements.
The numbers in the parentheses next to the finite element type show how many elements are in the entire model/in the displayed parts/or among the highlighted elements.
Consider the following settings shown in the figure: reduce the stiffness to ***50 percent*** (***0.5***) for each element, but in case of ***Columns***, ***Beams*** and ***Other elements*** only the bending stiffness ***(k_I)*** should be lowered.



Finishing the data input, confirm the changing and close the window with clicking on the ***OK*** button.



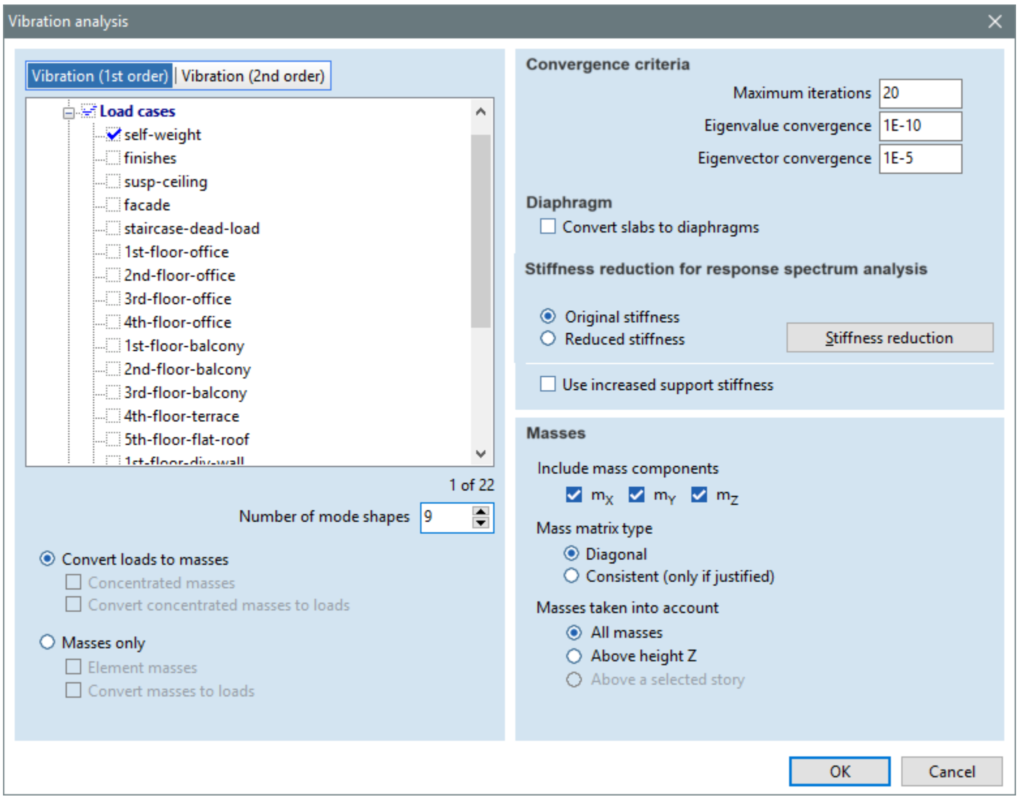
Vibration



Seismic load generation requires information on undamped free vibration frequencies and corresponding mode shapes. Thus, the first step in creating seismic loads is the calculation of a sufficiently large number of mode shapes and corresponding frequencies of vibration.

Change tab to the ***Vibration*** and complete an analysis according to the following.

Click on the ***Vibration analysis*** icon.



In the window that appears, set the required parameters. Let us go through the options:

At the top left of the window, the type of the analysis can be selected. By default, ***Vibration (1st order)*** is selected, which is appropriate for our investigation.
Note: In 2nd order vibration analysis the solution includes the effect of axial forces of truss/beam elements on the system stiffness. Tension axial forces have a stiffening effect, while the compression axial forces have a softening effect. These effects influence the free vibrations of the structure.

In the list under the type of the analysis, the load case or load combination can be selected as the basis of the vibration test. The program automatically selects the first ***load case*** (**self-weight**). Uncheck this load case.

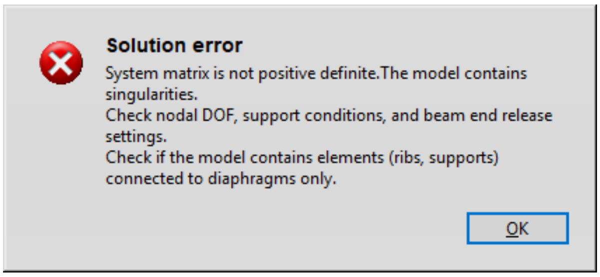
After find and check our custom ***load combination*** called **SEISMIC** at the end of the list, under ***Load combinations***.
Note: at once, user can select more load case/load combinations. In this case, the program performs as many independent calculations as the number of cases have been marked.

In vibration analysis, if masses are not specificially defined, masses can be determined based on the selected load combinations or load cases by converting them to masses. In our case the program automatically offers the ***Convert loads to masses*** option, since it does not contain masses.

Set the ***Number of mode shapes*** to **20**. This number depends heavily on the complexity of the model, the size of the building and the number of stories. (In our task, **20** mode shapes will be sufficient to reach at least **90%** of the total mass of the structure in the directions considered.)

Set the number of ***Maximum iterations*** to **30**. If the model does not converge properly, we may need to change it. The ***Eigenvalue*** and ***Eigenvector convergence*** should be left as default.

Diaphragm: when running a vibration analysis with checking the option of ***Convert slabs to diaphragms***, all slabs (horizontal plates) will be temporarily replaced by diaphragms (always check whether the conditions described in the standard are satisfied).
The running time may be reduced if the model contains only columns and slabs. If structural walls are included, the number of equations will be reduced but the bandwidth will be increased. The resultant running time may be greater than without diaphragms.
If there are ribs or supports in the model that are connected only to slab to be converted to diaphragm, this function cannot be used. If this problem occurs, the program gives an error message at the beginning of the calculation:



Note: the slabs/base plates which are supported (e.g. with surface support) will not be converted to diaphragm even if this function is selected.

Stiffness reduction for response spectrum analysis: here, the stiffness reduction of ***RC*** elements can be specified Previously, the relevant parameters have been set, therefore this setting does not have to be done again, but mark the checkbox to apply ***Reduced stiffness***.

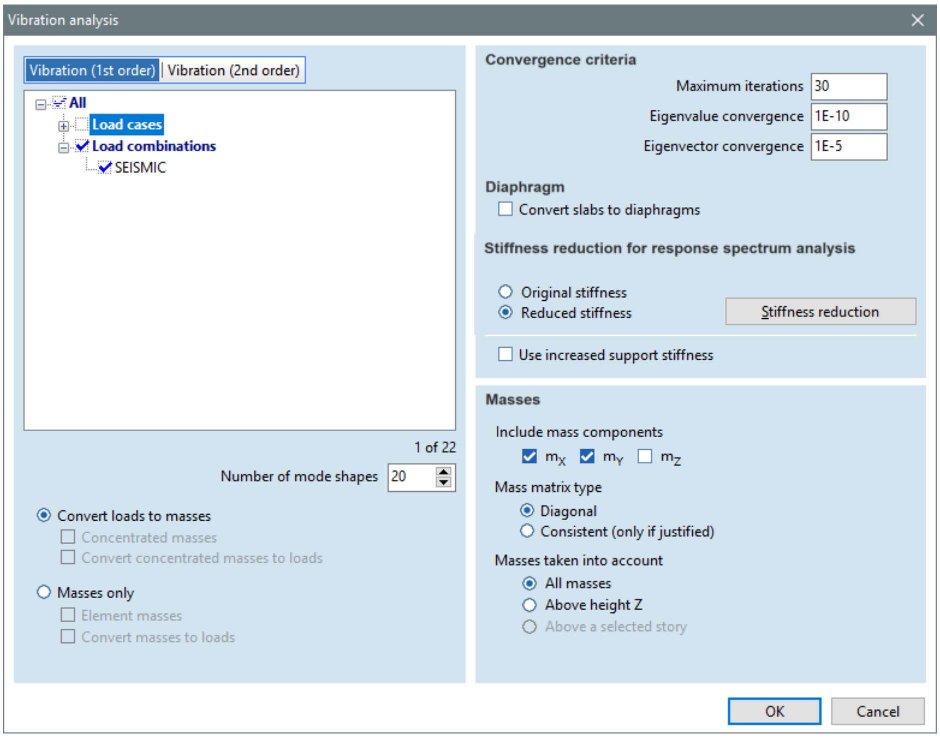
Use increased support stiffness:
Supports act in a different way during vibrations. Using increased support stiffness (**10^{10} kN/m** for nodal supports, **10^{7} kN/m/m** for line supports, **10^{4} kN/m^{2}** for surface supports) may help to get more realistic vibration results. ***Do not check this option now***, the stiffness parameters of the supports have been declared for seismic an dynamic actions.

Masses: in the calculation, only the ***x*** and ***y mass components*** should be considered (the vertical effect of the earthquake can be negligible depending on the country’s regulations).

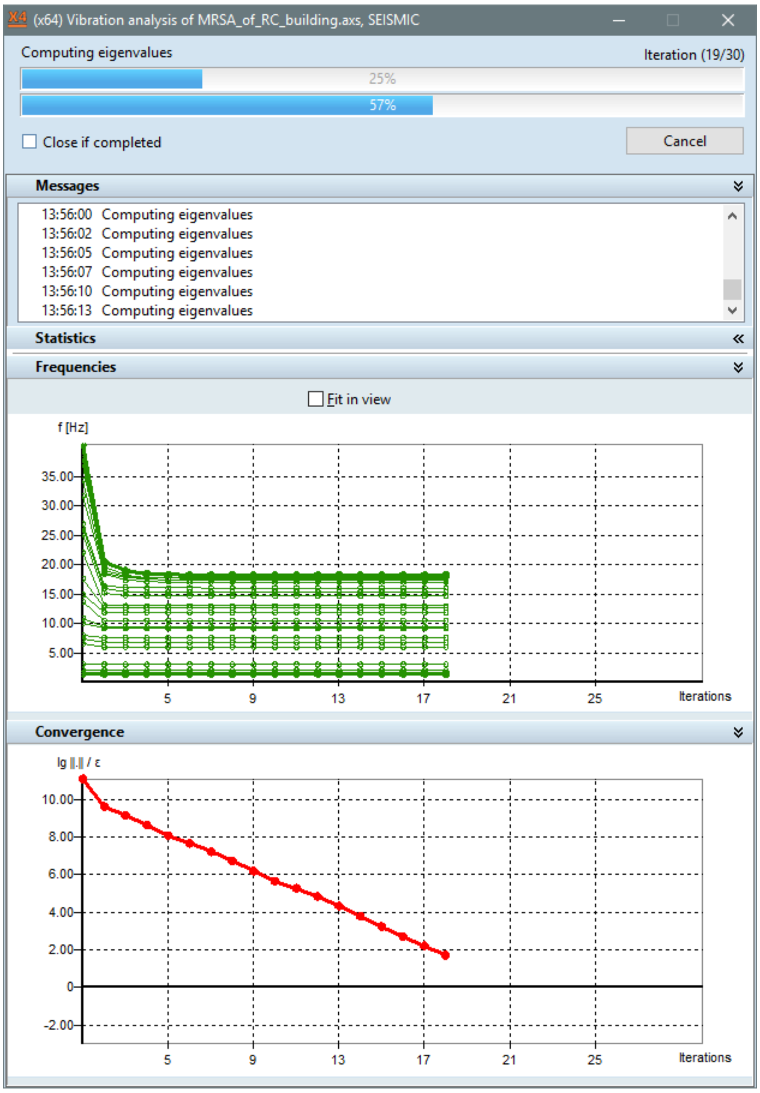
Configuration of ***Masses considered***: beside ***All masses*** it is possible to reduce the masses to those above a given ***Z*** height or a given level (if the model contains levels). An example could be the exclusion of the basement from the vibration analysis, the self-weight and other types of loads are not considered below the ground level, including the base plate (it the model contains base plate).
At this option the default setting - ***All masses*** - has to be selected.

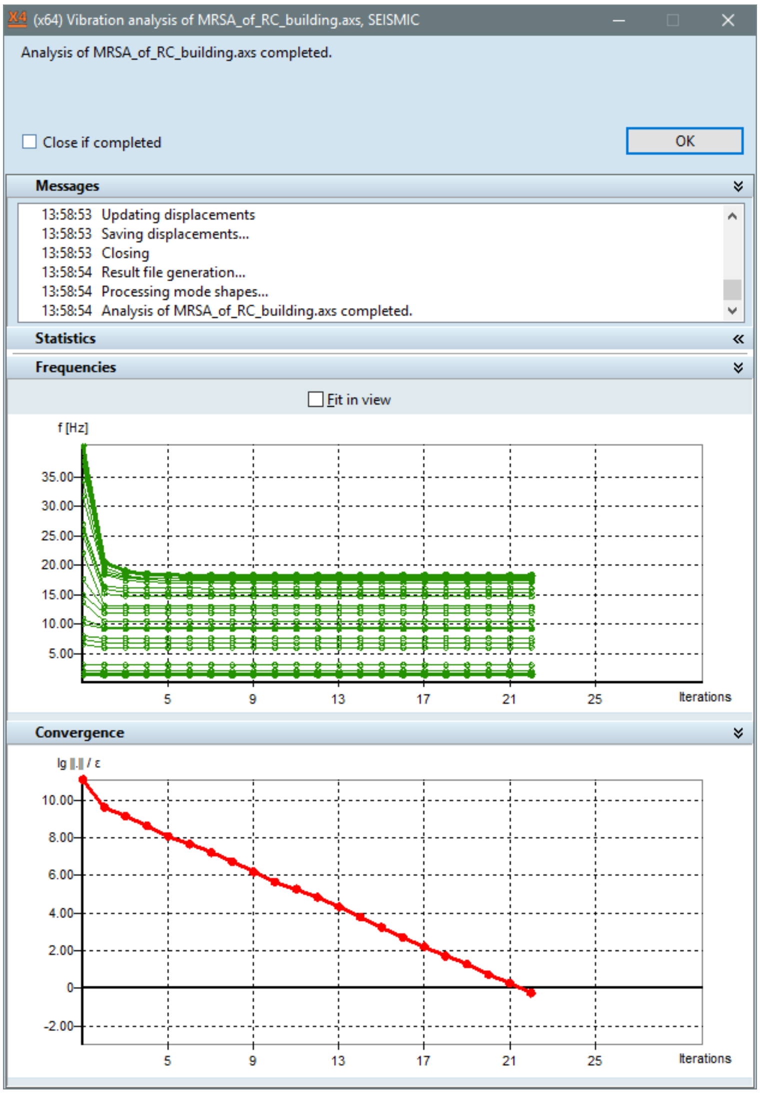
For more details about the parameters and the settings above, see the relevant sections in the ***User’s Manual***.

Set the parameters shown below as a summary, then click ***OK*** to start the analysis.



During the calculation, the ***frequencies of mode shapes*** and ***the slowest convergent shapes*** can be traced ***as a function of the iterations***.





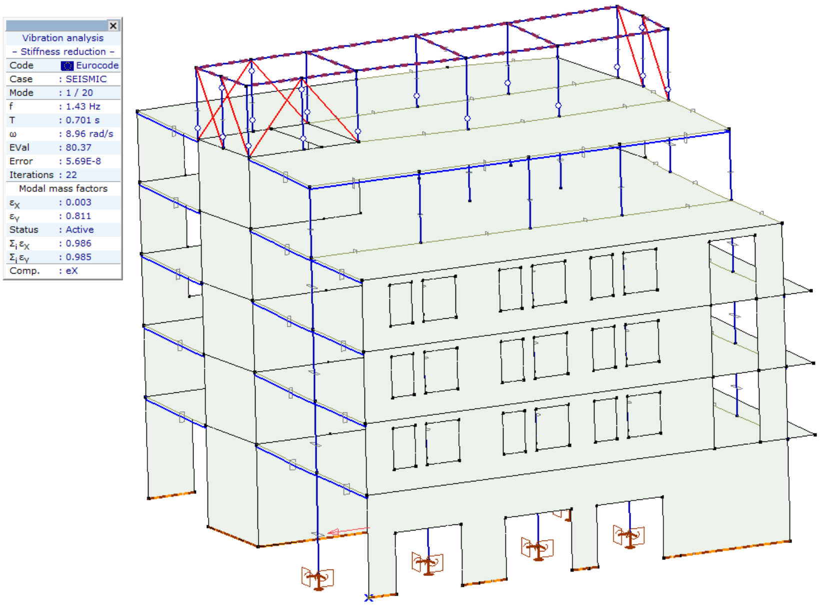
The analysis stopped at **23th** iteration step because the set convergence criteria have been met, no need to continue until the final **30th** step.

Click on ***OK*** to close the window.

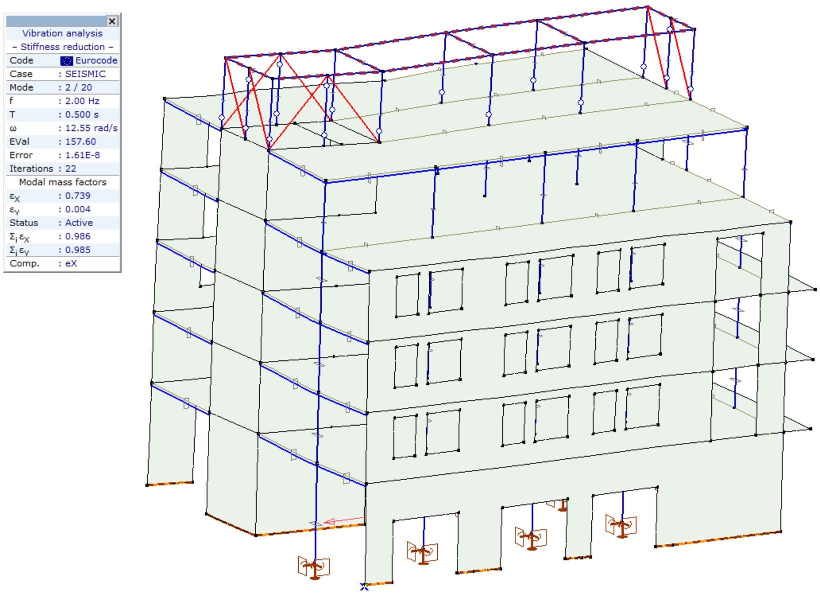
Always check the resulting mode shapes. If there is an error in the model, analyzing the mode shapes may help to filter out the main problems (instability, any problem in stiffness). On the other hand, we get an overview about the dynamic behaviour of the building (e.g. which are the main shapes, our structure is sensitive to the torsional effects or not, etc...).

The first three shapes are shown below displaying the displaced shape applying a zoom scale of **10**:

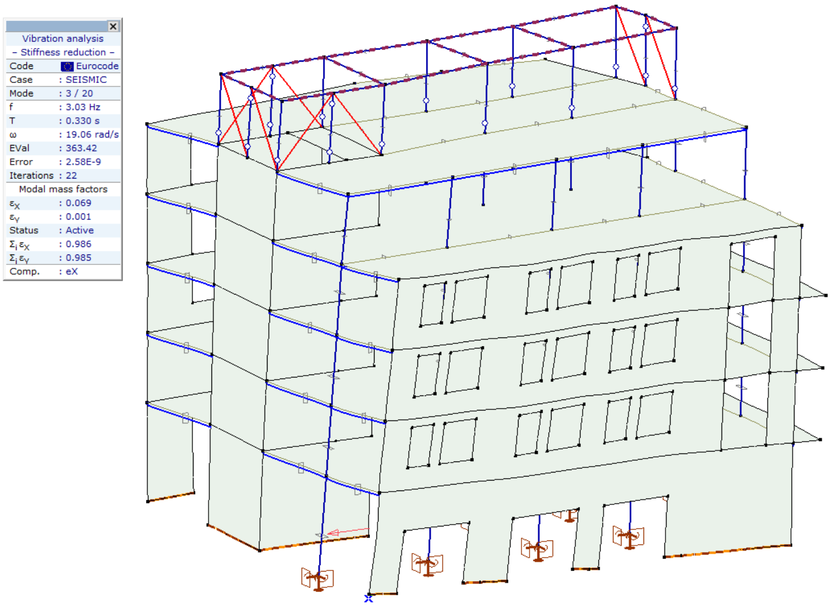
1st mode shape:



2[nd] mode shape:

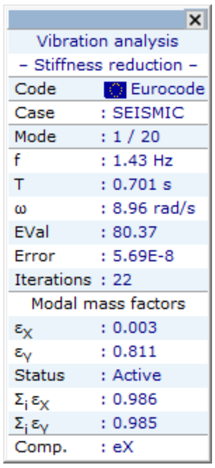


3rd mode shape:



The ***Status*** palette on the left display information about the mode shapes that can be helpful in the evaluating of results.

Let us go through these, select the ***1st mode shape***:



Under the title of ***Vibration analysis***, it is indicated that the calculation was actually performed with using ***Stiffness reduction*** (see the explanation for this function earlier).

Below the applied ***design Code*** is shown.

The ***Case*** shows the ***load case/ load combination*** applied in the vibration analysis (now it is **SEISMIC**).

Mode shows the ordinal number of the mode shape (the **first** from the calculated **20** shapes).

In the next unit of the palette, the main results of the actual mode shape are presented (frequency, period, circular frequency, eigenvalue, error). The list ends with the maximum number of iterations (**22**): the calculation has been finished at the iteration step of **22th**.

The unit of ***Modal mass factors*** provides information about the followings:

- the modal mass factor in each direction (**e_X** és **e_Y**) which belongs to the displayed mode shape (if the global ***Z*** component of the mass is taken into account in the analysis, it also appears in the list).

- ***Status*** informs about the active or inactive state of the displayed mode shape – *see the explanation later*.
- under the ***Status***, the total modal mass factors achieved by all the shapes set as ***active***. Now, these pass the limit of **90%** in both calculated directions ***x*** and ***y***, so the determined number of shapes (***20***) are sufficient. No need to calculate more shapes.
- ***Comp.*** shows the actual result component that is being displayed (***eX***).

Table browser

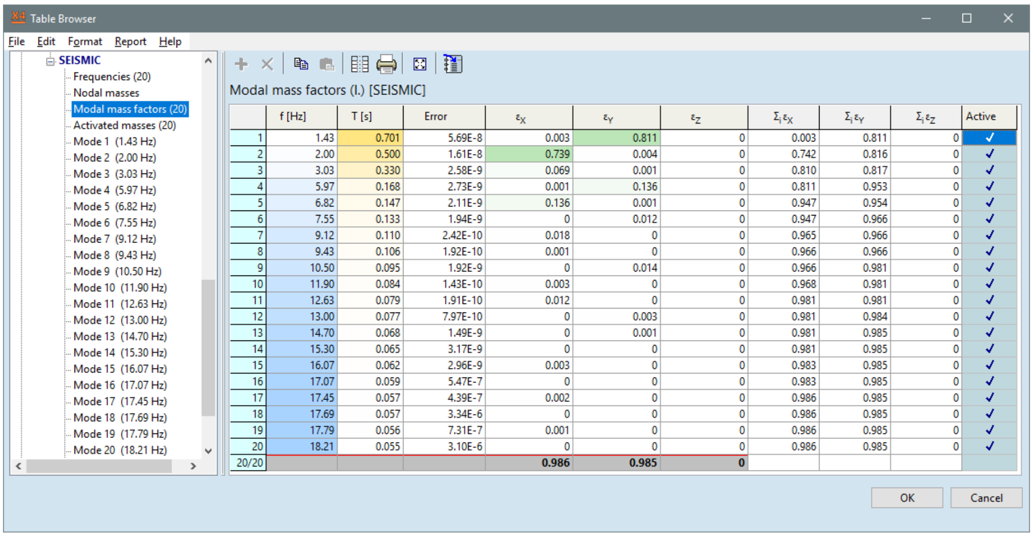
The results of the vibration analysis are presented in the table of ***Modal mass factors*** which is available in the ***Table browser***. Click on the ***Table browser*** icon.

It is extremely important to meet the standard requirements which refer to the total mass factor. The general ***EN 1998-1:2008*** has two requirements:

(1) the total considered modal mass in each direction shall exceed **90%** of the total mass;

(2) all mode shapes corresponding to a modal mass larger than **5%** of the total mass shall be taken into consideration.

The sums of modal mass factors can be checked at the bottom of the table.



Modal mass factors (I.) [SEISMIC]

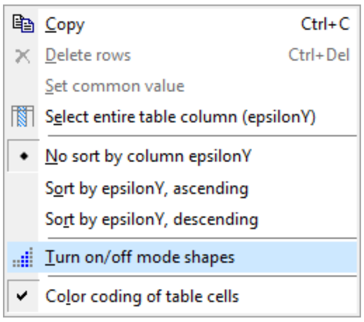
	f [Hz]	T [s]	Error	ε_X	ε_Y	ε_Z	$\Sigma_i \varepsilon_X$	$\Sigma_i \varepsilon_Y$	$\Sigma_i \varepsilon_Z$	Active
1	1.43	0.701	5.69E-8	0.003	0.811	0	0.003	0.811	0	✓
2	2.00	0.500	1.61E-8	0.739	0.004	0	0.742	0.816	0	✓
3	3.03	0.330	2.58E-9	0.069	0.001	0	0.810	0.817	0	✓
4	5.97	0.168	2.73E-9	0.001	0.136	0	0.811	0.953	0	✓
5	6.82	0.147	2.11E-9	0.136	0.001	0	0.947	0.954	0	✓
6	7.55	0.133	1.94E-9	0	0.012	0	0.947	0.966	0	✓
7	9.12	0.110	2.42E-10	0.018	0	0	0.965	0.966	0	✓
8	9.43	0.106	1.92E-10	0.001	0	0	0.966	0.966	0	✓
9	10.50	0.095	1.92E-9	0	0.014	0	0.966	0.981	0	✓
10	11.90	0.084	1.43E-10	0.003	0	0	0.968	0.981	0	✓
11	12.63	0.079	1.91E-10	0.012	0	0	0.981	0.981	0	✓
12	13.00	0.077	7.97E-10	0	0.003	0	0.981	0.984	0	✓
13	14.70	0.068	1.49E-9	0	0.001	0	0.981	0.985	0	✓
14	15.30	0.065	3.17E-9	0	0	0	0.981	0.985	0	✓
15	16.07	0.062	2.96E-9	0.003	0	0	0.983	0.985	0	✓
16	17.07	0.059	5.47E-7	0	0	0	0.983	0.985	0	✓
17	17.45	0.057	4.39E-7	0.002	0	0	0.986	0.985	0	✓
18	17.69	0.057	3.34E-6	0	0	0	0.986	0.985	0	✓
19	17.79	0.056	7.31E-7	0.001	0	0	0.986	0.985	0	✓
20	18.21	0.055	3.10E-6	0	0	0	0.986	0.985	0	✓
20/20				**0.986**	**0.985**	**0**				

The rows of the table show the main results of the determined mode shapes and the corresponding mass factors. The columns named $S_i e_X$; $S_i e_Y$ and $S_i e_Z$ show the summed mass factor in the main directions, but only the results of shapes set as ***active*** are summed up.

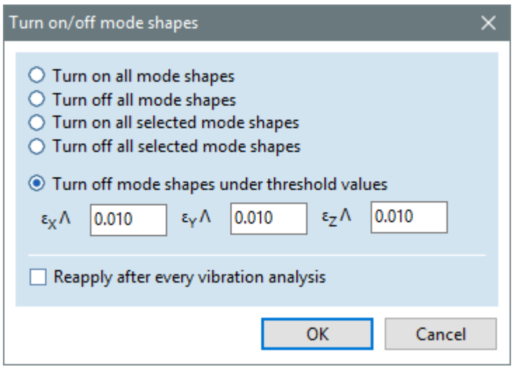
Among the shapes, there are many shapes which mass factor is negligible (e.g. less than 0.01). The negligible shapes can be switched to inactive, but make sure that the total mass factor in each direction does not go under the limit of 0.90 (90%). The inactive shapes will be ignored when generating the seismic load. (Keep in mind that neglecting some of the shapes also proportionally reduces the total seismic load.)

Every mode shape that has been indicated as active will result one or two (even if the torsional effect is considered) seismic load case per direction. Thus, the filtering of the modes shapes is recommended if there are plenty of shapes, as it can significantly reduce the number of load cases and the required calculation time as well.

The active status of the mode shapes can be switched in two ways. We can go through on each row and at the end of the rows the status of the current shape can be modified by clicking on its field. Or standing on the table with the cursor, a right-click opens a menu, in which the ***Turn on/off mode shapes*** dialog can be activated.

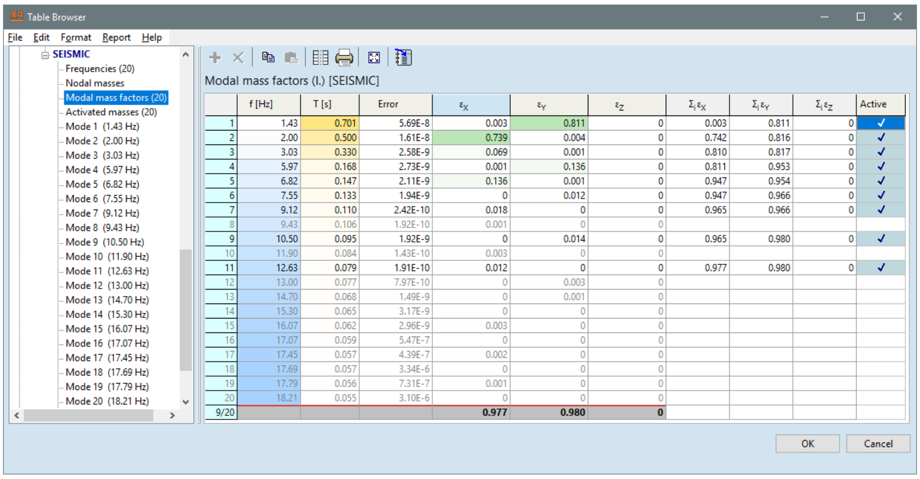


In the window that appears, several filtering condition can be set as shown in the next figure. For example, the mode shapes that has a modal mass factor less than a certain limit can be turn off (different limit can be set for each direction). Let us use this function, adjust the criteria of **1%** for both directions as shown below (now it is enough to enter only the limit of ***x*** and ***y***).



Mark also the function ***Reapply after every vibration analysis***. If a new calculation is made for any reason, the set filtering procedure will be performed automatically, but do not forget to check the achieved total mass factor in the main directions.

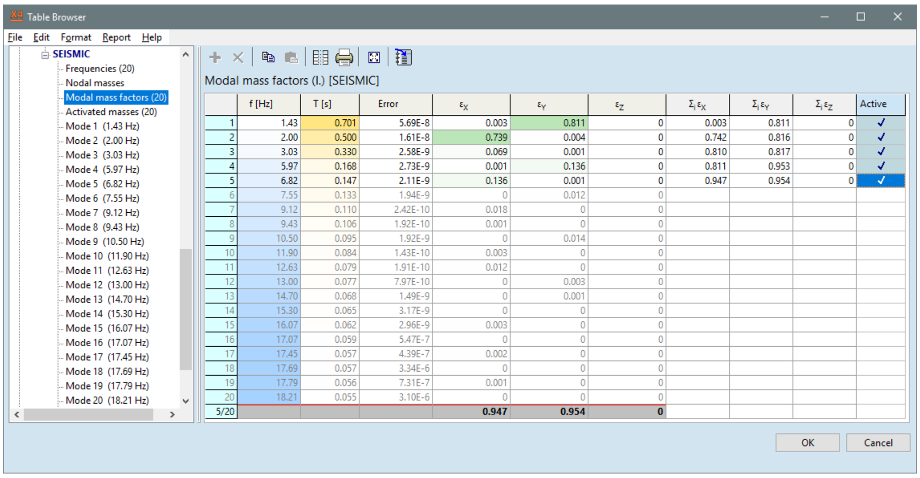
Click on the ***OK*** button to close the dialog. Now, examine the results and the total modal mass factor in each direction:



	f [Hz]	T [s]	Error	ε_X	ε_Y	ε_Z	$\Sigma_i \varepsilon_X$	$\Sigma_i \varepsilon_Y$	$\Sigma_i \varepsilon_Z$	Active
1	1.43	0.701	5.69E-8	0.003	0.811	0	0.003	0.811	0	✓
2	2.00	0.500	1.61E-8	0.739	0.004	0	0.742	0.816	0	✓
3	3.03	0.330	2.58E-9	0.069	0.001	0	0.810	0.817	0	✓
4	5.97	0.168	2.73E-9	0.001	0.136	0	0.811	0.953	0	✓
5	6.82	0.147	2.11E-9	0.136	0.001	0	0.947	0.954	0	✓
6	7.55	0.133	1.94E-9	0	0.012	0	0.947	0.966	0	✓
7	9.12	0.110	2.42E-10	0.018	0	0	0.965	0.966	0	✓
8	9.43	0.106	1.92E-10	0.001	0	0				
9	10.50	0.095	1.92E-9	0	0.014	0	0.965	0.980	0	✓
10	11.90	0.084	1.43E-10	0.003	0	0				
11	12.63	0.079	1.91E-10	0.012	0	0	0.977	0.980	0	✓
12	13.00	0.077	7.97E-10	0	0.003	0				
13	14.70	0.068	1.49E-9	0	0.001	0				
14	15.30	0.065	3.17E-9	0	0	0				
15	16.07	0.062	2.96E-9	0.003	0	0				
16	17.07	0.059	5.47E-7	0	0	0				
17	17.45	0.057	4.39E-7	0.002	0	0				
18	17.69	0.057	3.34E-6	0	0	0				
19	17.79	0.056	7.31E-7	0.001	0	0				
20	18.21	0.055	3.10E-6	0	0	0				
9/20				**0.977**	**0.980**	**0**				

The requirements are satisfied based on the **9** active mode shapes.

Let us look more closely, the number of active shapes could still be reduced. Based on the results, the first five modes would be sufficient to meet the requirements, the remaining four may be negligible:

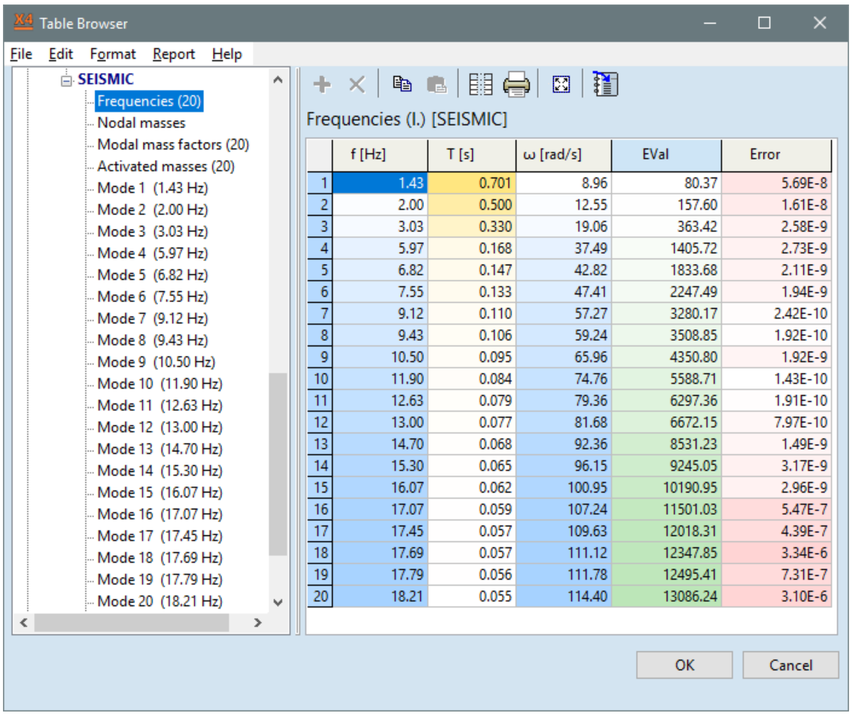


Modal mass factors (I.) [SEISMIC]

	f [Hz]	T [s]	Error	ε_X	ε_Y	ε_Z	$\Sigma_i \varepsilon_X$	$\Sigma_i \varepsilon_Y$	$\Sigma_i \varepsilon_Z$	Active
1	1.43	0.701	5.69E-8	0.003	0.811	0	0.003	0.811	0	✓
2	2.00	0.500	1.61E-8	0.739	0.004	0	0.742	0.816	0	✓
3	3.03	0.330	2.58E-9	0.069	0.001	0	0.810	0.817	0	✓
4	5.97	0.168	2.73E-9	0.001	0.136	0	0.811	0.953	0	✓
5	6.82	0.147	2.11E-9	0.136	0.001	0	0.947	0.954	0	✓
6	7.55	0.133	1.94E-9	0	0.012	0				
7	9.12	0.110	2.42E-10	0.018	0	0				
8	9.43	0.106	1.92E-10	0.001	0	0				
9	10.50	0.095	1.92E-9	0	0.014	0				
10	11.90	0.084	1.43E-10	0.003	0	0				
11	12.63	0.079	1.91E-10	0.012	0	0				
12	13.00	0.077	7.97E-10	0	0.003	0				
13	14.70	0.068	1.49E-9	0	0.001	0				
14	15.30	0.065	3.17E-9	0	0	0				
15	16.07	0.062	2.96E-9	0.003	0	0				
16	17.07	0.059	5.47E-7	0	0	0				
17	17.45	0.057	4.39E-7	0.002	0	0				
18	17.69	0.057	3.34E-6	0	0	0				
19	17.79	0.056	7.31E-7	0.001	0	0				
20	18.21	0.055	3.10E-6	0	0	0				
5/20				0.947	0.954	0				

Let us go back to the previous setting, do not apply this simplification now.

In the tree list on the left, the table called ***Frequencies*** is also accessible. This table gives overall information about the mode frequencies, period, angular frequency, eigenvalue and error for each mode shapes.

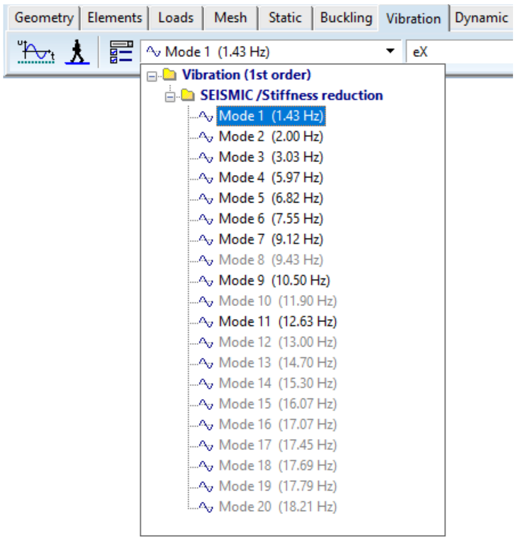


Frequencies (I.) [SEISMIC]

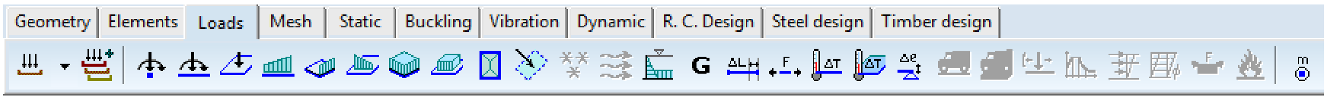
	f [Hz]	T [s]	ω [rad/s]	EVal	Error
1	1.43	0.701	8.96	80.37	5.69E-8
2	2.00	0.500	12.55	157.60	1.61E-8
3	3.03	0.330	19.06	363.42	2.58E-9
4	5.97	0.168	37.49	1405.72	2.73E-9
5	6.82	0.147	42.82	1833.68	2.11E-9
6	7.55	0.133	47.41	2247.49	1.94E-9
7	9.12	0.110	57.27	3280.17	2.42E-10
8	9.43	0.106	59.24	3508.85	1.92E-10
9	10.50	0.095	65.96	4350.80	1.92E-9
10	11.90	0.084	74.76	5588.71	1.43E-10
11	12.63	0.079	79.36	6297.36	1.91E-10
12	13.00	0.077	81.68	6672.15	7.97E-10
13	14.70	0.068	92.36	8531.23	1.49E-9
14	15.30	0.065	96.15	9245.05	3.17E-9
15	16.07	0.062	100.95	10190.95	2.96E-9
16	17.07	0.059	107.24	11501.03	5.47E-7
17	17.45	0.057	109.63	12018.31	4.39E-7
18	17.69	0.057	111.12	12347.85	3.34E-6
19	17.79	0.056	111.78	12495.41	7.31E-7
20	18.21	0.055	114.40	13086.24	3.10E-6

Close the ***Table browser***. Click on ***OK*** to confirm the set filtering criteria.

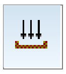
In the main window, open the list containing the result of mode shapes. Due to the filtering criteria applies, some of the shapes are shown by pale grey color which also indicates their ***inactive*** status.
The ***active*** or ***inactive*** status of the shapes is also shown on the ***Status*** palette.



Setting the necessary filtering criteria, return to the ***Loads*** tab to generate the seismic load based on the results of vibration analysis.

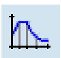


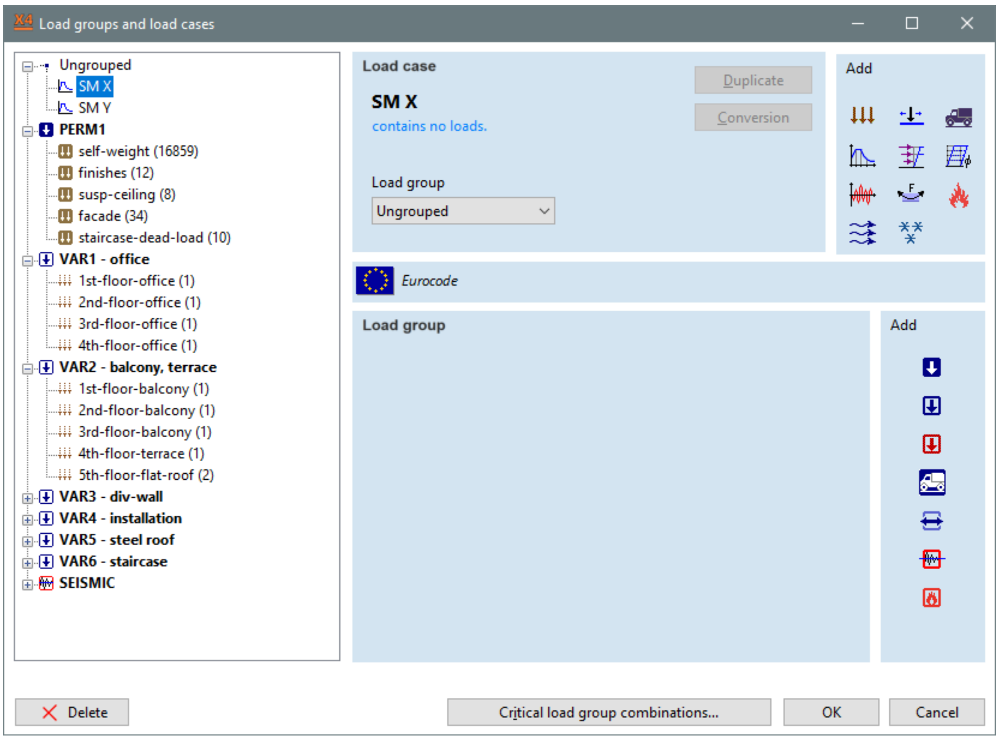
Load cases and load groups



Click on the ***Load cases and load groups*** icon and create a new ***Seismic*** load case.
In the ***Add*** panel, click on the icon that symbolizes a response spectrum curve. As a result, the program creates two ungrouped cases with names **SM X**, **SM Y** and a new ***load group*** (**SEISMIC**) is also added to the list containing **SM +** and **SM –** load cases:

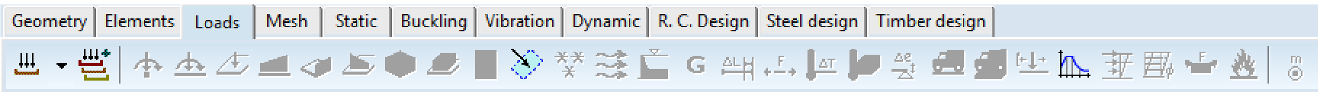
Seismic



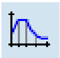


The program will automatically generate other load cases necessary for the analysis after setting the seismic parameters. Confirm changes with clicking on the ***OK*** button.

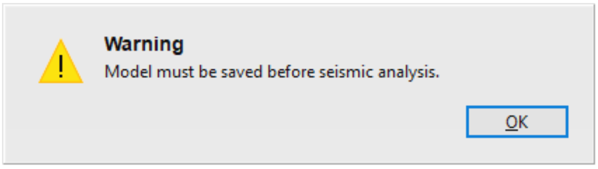
The program automatically set the case **SM X** as current load case and the ***Seismic load*** icon becomes available on the ***Loads*** panel (this function is only active if a seismic type load case is selected).



Seismic load

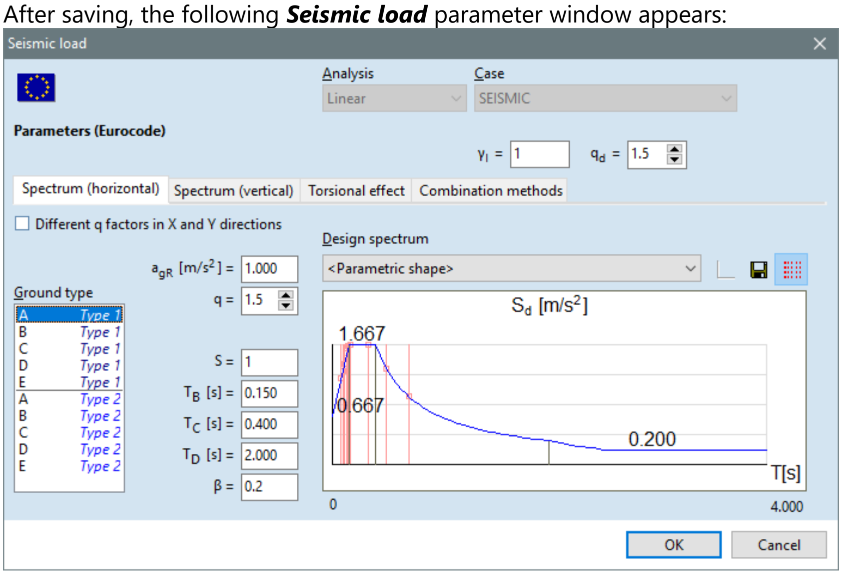


Click on the ***Seismic load*** icon to set earthquake parameters, then the program displays a warning message:



Save the model, click on the ***OK*** button.

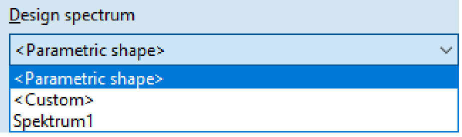
After saving, the following ***Seismic load*** parameter window appears:



Adjust the required parameters for our example, let us go through the options:

- at ***Analysis*** the type of the vibration analysis can be selected, but only those that have been already performed: this can be ***Linear*** / ***Nonlinear*** (see ***User's Manual*** for details of a nonlinear vibration analysis). This function is now inactive because only one, a linear analysis has been performed before, so there is no choice.
- next to the ***Analysis,*** the ***Case*** can be selected (the load case or load combination which was considered in the vibration analysis). Only one analysis has been made (considering ***SEISMIC*** load combination), therefore it is selected automatically and the combo box is in grey, inactive state, which means there is no choice.
- under the ***Case***, the ***Importance factor*** can be specified (g_I). The default value is **1** which equal to the parameter to be applied, so does not have to change this.
- next to the ***Importance factor***, the ***displacement behaviour factor*** has to be set (***EN 1998-1:2008-1 4.3.4 (1)***). Adjust this to **1.50**, this will be the same as the behaviour factor, see later.

- the necessary response spectrum has to be specified on the ***Spectrum (horizontal)*** tab. Under the ***Design spectrum*** several choices are offered:
  a) the response spectrum curve can be given by ***parametric shape*** based on the regulation of the standard,
  b) ***Custom*** spectrum can also be specified,
  c) or ***existing curves*** can be loaded as well:



Spectral function editor…

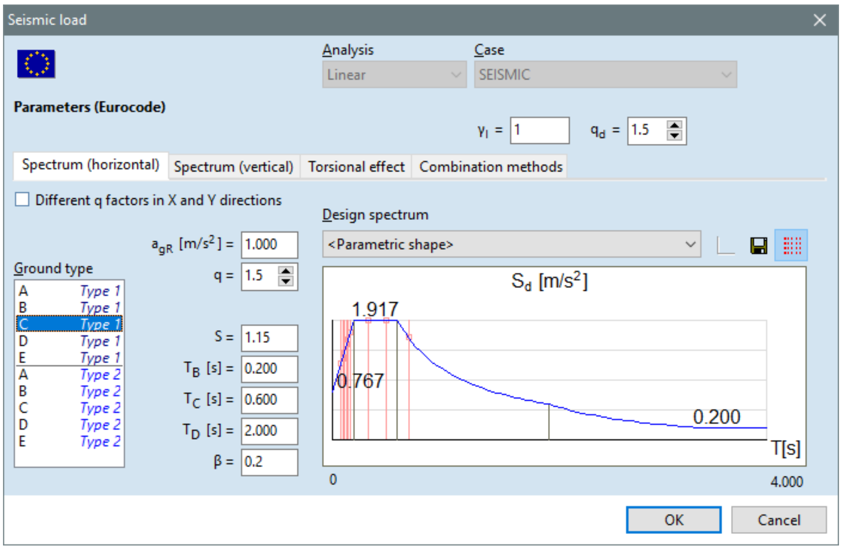
Applying ***Custom*** spectrum, the ***Spectral function editor*** has to be used. The spectrum can be specified by using a formula or complete series of data can be pasted to the cells (e.g. copying ***Excel*** data considering the table format of ***AxisVM***).
The using of the editor is not presented now, for more information please see the relevant chapter of the ***User's Manual***.

In the following, let us define a ***Parametric shape***. The parameters that should be specified are on left side (defining a custom spectrum or loading an existing one, these input fields will be enabled).

- the reference value of peak ground acceleration is **1,0 m/s²**, and the behaviour factor is **1.50** based on the basic data described in the introduction.
  If it is justified by the structural concept, different ***q behaviour factor*** can be applied in ***X*** and ***Y*** directions. In this case, the checkbox in the top left corner must be marked to be enabled the editing of different parameters. Do not mark this option, because the same factor has to be used for both direction in our example.

- on the left side the ***type of the soil*** (***A…E***) according to the classification of the standard and the type of the ***earthquake response spectrum*** (***Type 1*** or ***Type 2***) can be specified. According to the basic data, spectrum ***Type 1*** and ***C soil class*** has to be used. Select the appropriate row in the list: ***C Type 1***.

Changing the parameters, the program displays the response spectrum curve on the right side indicating the main characteristic acceleration values (***Sd***).



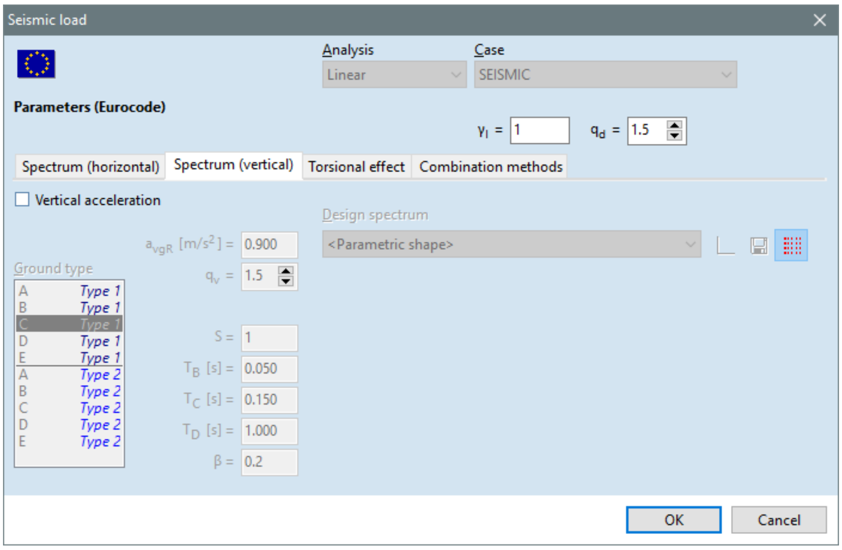
Save spectrum as…

The main parameters of the standard curve (***S, T_B, T_C, T_D***, β) are shown under the ***behaviour factor***. If necessary, these values can also be modified and the result curve can be saved with clicking on the ***Save spectrum as…*** function.

Auxiliary lines

On the response spectrum curve, the ***red auxiliary lines*** show the period time of the active mode shapes. They give information about the range of acceleration values which will be used to generate seismic loads. These ***auxiliary lines*** can be turned off clicking on its icon next to the ***Save spectrum as…*** one.

Setting the parameters above, click on the ***Spectrum (vertical)*** tab. If it is necessary and it is justified based on the type of the expected earthquake, the vertical component of the earthquake can also be considered with setting the relevant parameters on this tab. The data input is similar to the previous one. In our example the vertical component is neglected, let us skip this tab.



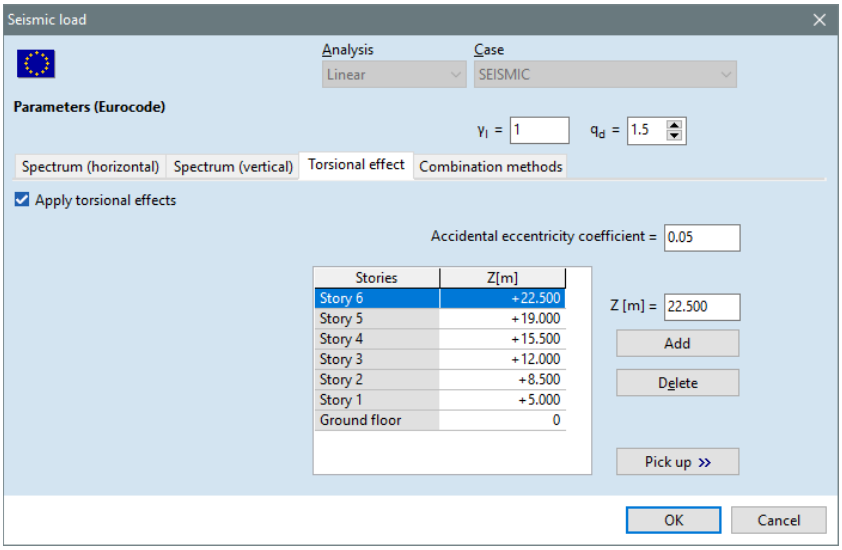
Step to the ***Torsional effect*** tab and define the necessary data:

Firstly, mark the checkbox (***Apply torsional effects***) in top left corner.

According to the standard the recommended ***Accidental eccentricity coefficient*** is **0.05**. It is the default value, no need to overwrite it.

In the next step, the stories (to which the eccentric loads will be assigned) have to be specified:

If there are existing stories in the model (see earlier), the list is automatically uploaded as shown in the figure below. If necessary, the generated list can be modified (independently of the ***Stories*** that has a role in displaying parts of the model only): items can be removed (***Delete***) or new ones can be inserted.
New stories can be added to the list in different ways:
Set the specific height in the field of ***Z[m]*** and click on the ***Add*** icon to insert stories to the list one by one. Using the ***Pick-up*** function, the height points of the stories has to be marked by the cursor in the main window. More than one can also be specified at once with this function.



The uploaded stories are correct, do not modify the set.

beszúrt

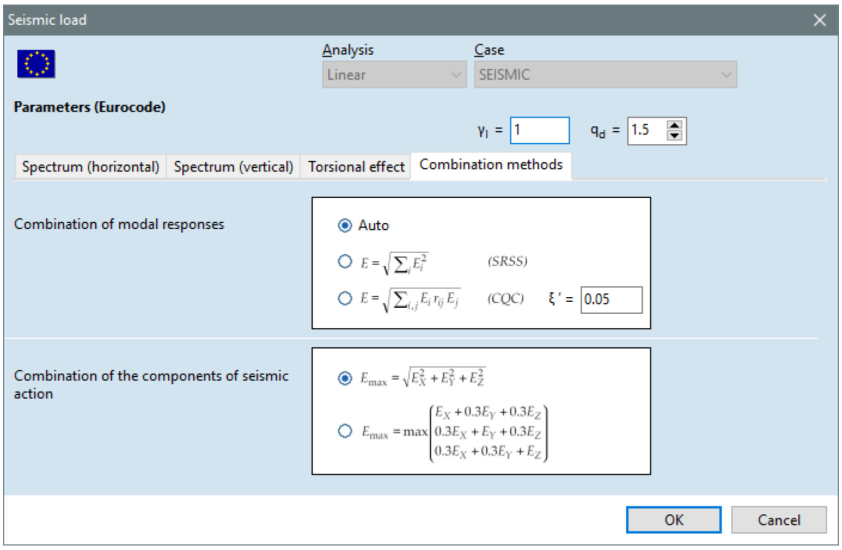
Step to the last tab called ***Combination methods***.
The last tab of the ***Seismic Load*** dialog window provides control over the combination of results for individual modes and individual directions.
Results from individual modes can be combined using either:

the ***Square Root of Sum of Squares*** (***SRSS***),
the ***Complete Quadratic Combination*** (***CQC***) method.

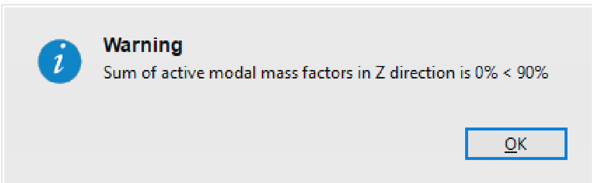
The latter is considered more appropriate if the vibration modes of the structure are not well separated (i.e. vibration frequencies are close to each other).
Selecting the automatic option lets the program automatically decide if application of the CQC method is warranted by the vibration results. The program considers modes ***i*** and ***j*** well separated if the following condition holds: $\boldsymbol{T_j / T_i < 0.9}$.
The last option is the default, ***do not change this in our example***.

The value of ***viscous damping*** **x’** can also be set. The default value is **0.05**, no need to change this in our example.

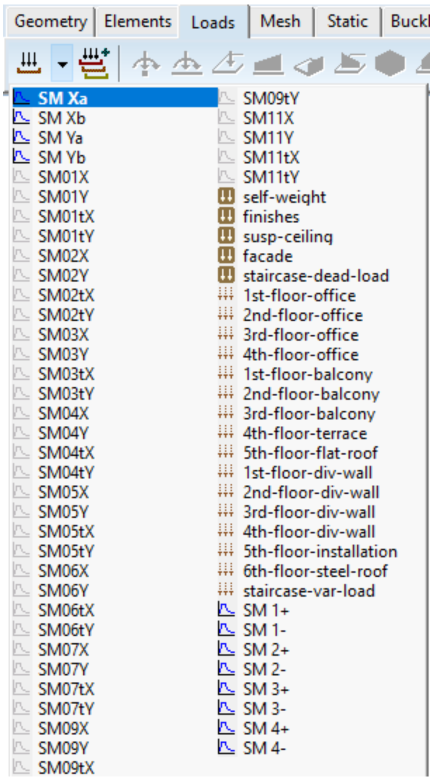
Results in the ***two horizontal and the vertical direction can be combined*** using either of the two commonly used combination methods displayed in the dialog window. Use also the default setting as shown below:



Finish the data input and close the window with clicking on the ***OK*** button. After the program shows a warning message about the lack of the required modal mass factor in ***Z*** direction (report shows **0%** because the vertical component of the masses was not considered in our vibration analysis).



Since the vertical (global ***Z***) seismic component is not considered, this message can be ignored. Go on with clicking on the ***OK*** button, then the software generates the load cases for each mode and direction according to the settings defined by the user.

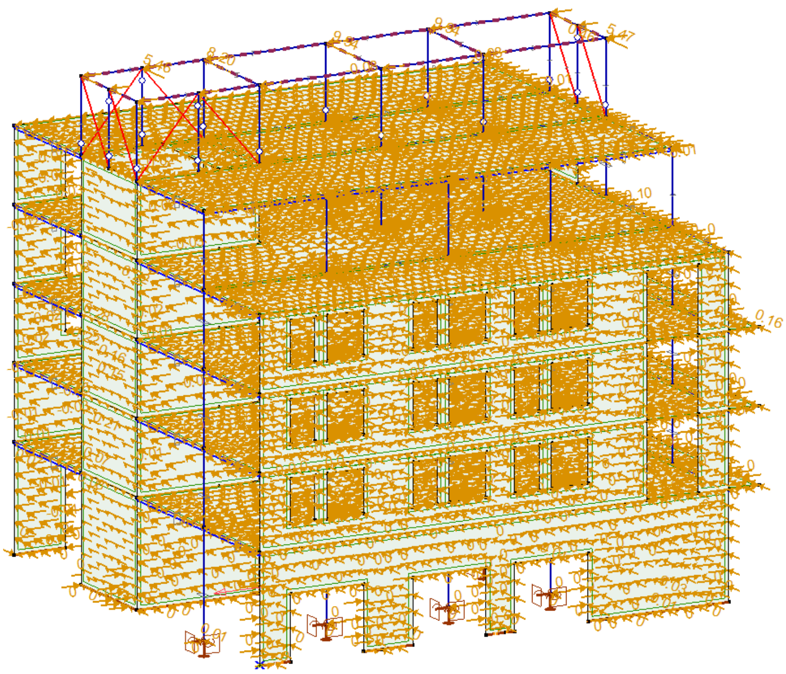


The following naming convention is used for these load cases: Load cases ending in a ***01X, 02X, ….nX, 01Y, 02Y, ….nY, 01Z, 02Z, ….nZ*** are the equivalent forces in ***X***, ***Y*** or ***Z*** direction corresponding to individual mode shapes. Load cases ending in ***01tX, 02tX, ….ntX, 01tY, 02tY, ….ntY*** are the torsional moments due to accidental eccentricity in ***X*** or ***Y*** direction.

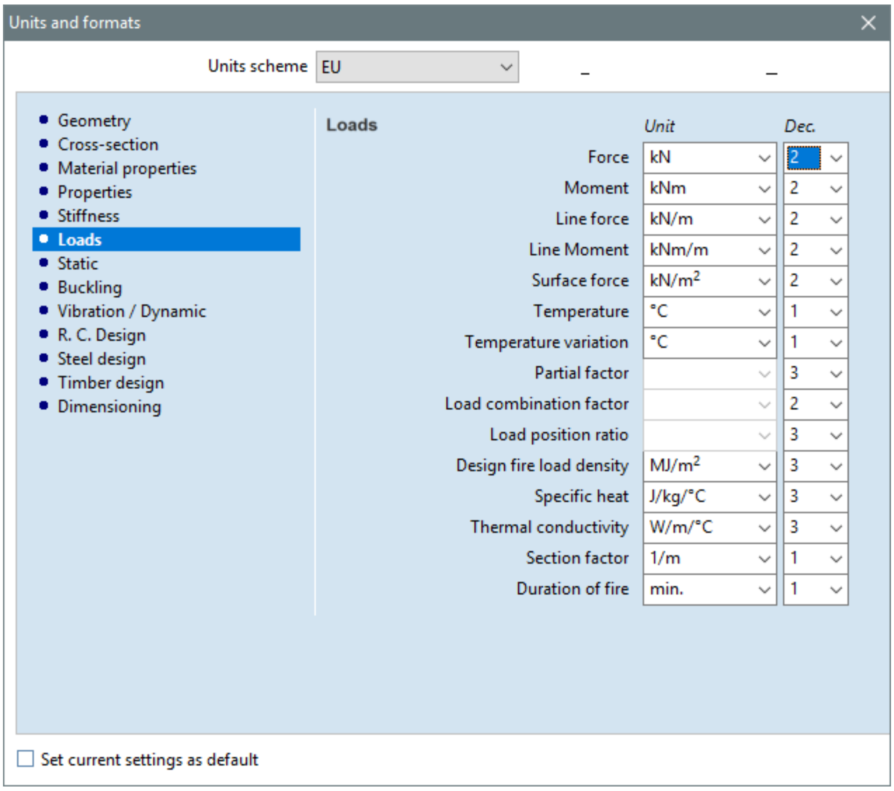
Note: in our example, **9** mode shapes were marked as active. Considering ***X*** and ***Y*** directional seismic loads and their torsional effect, the program generates **2x**(**1+1**)**=4** cases for each active mode shape. A total of **9x4=36** load cases have been generated.

(It was previously mentioned that the first **5** modes shapes would have been sufficient according to the standard. On this way, a total of **20** cases would have only been created.)

In the drop-down list of the load cases, choose ***SM01Y*** load case and switch on the displaying of loads. The program displays the concentrated seismic loads on the nodes (inner mesh nodes as well) that have been assigned to the current load case:



If the nodal load is relatively 'small', the value of the concentrated load may seem to be 'zero' depending on the displayed decimals. If necessary, modify the decimals assigned to the unit (***Loads***/***Force***) to see more decimals. The function is available here: ***Settings***/***Units and formats***/***Loads***/***Force***.



Note: if the mesh or a part of it is deleted because of any modification, then the seismic load will also be cleared.

The other generated load cases are reserved for the results, the following serve for their explanation:

Modal response spectrum analysis uses a combination of linear static analysis results to get the design seismic effects for the structure under consideration. After the load cases for each vibration mode in each direction are created, the next step in the ***MRSA*** procedure is to run the linear static analyses. The program will automatically calculate the effect of each mode in each direction and combine these effects according to the settings in the ***Seismic Load*** dialog window.

After running linear static analyses, there are several seismic load cases available in the list of results under the ***Static*** tab. We use the following naming convention for seismic load cases:

Besides load cases corresponding to individual modes, there are two types of additional results. Load case names ending in ***X***, ***Y***, or ***Z*** contain the combined response from modal results in the ***X***, ***Y*** or ***Z*** direction. If accidental eccentricity is considered, there is an additional ***a*** or ***b*** in the name of load cases corresponding to horizontal directions. The letters ***a*** or ***b*** correspond to torsional effects with positive or negative eccentricity, respectively. ***Ya*** for instance is a combination of modal results in the ***Y*** direction considering the effect of positive eccentricity of seismic masses in the ***X*** direction.

Combination of load cases in ***X***, ***Y***, ***Z*** directions is performed by the program automatically. Such combination yields a single, unanimous result if there is no accidental eccentricity. If accidental eccentricity is assumed, the combination of effects from several directions is more ambiguous. There are four basic cases considered in the program depending on the direction of eccentricity in the ***X*** and ***Y*** load cases. Each of the following combinations are performed and stored in a load case with its name ending with the particular number:

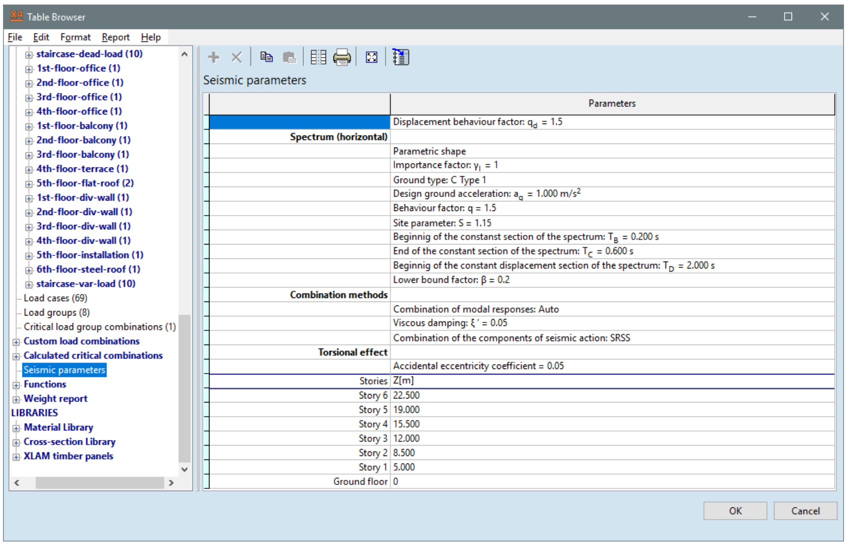
1 = Xa + Ya + Z
2 = Xa + Yb + Z
3 = Xb + Ya + Z
4 = Xb + Yb + Z

MRSA results are absolute values by definition. Therefore, the load cases presented above would contain only positive values. The least favorable loading scenario is a combination of seismic effects and effects from other sources such as gravity loading. Creation of this load combination is facilitated in the program by providing two load cases for each seismic load: seismic effects with only positive and seismic effects with only negative values are identified by a **+** and a **–** sign at the end of the name of their load cases. Note that the absolute values of corresponding responses (internal forces, displacements, etc.) are identical in the **+** and **–** load cases.

Note: displacement results shown in the ***Static*** tab are automatically scaled by the q_d factor specified in the ***Seismic load*** dialog window.

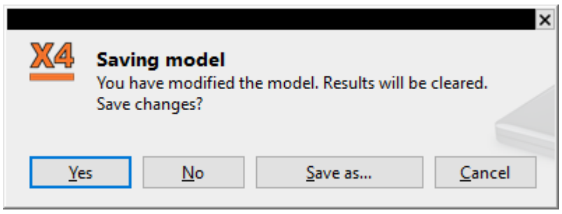
Table browser

Let us check the set earthquake parameters.
Click on the ***Table Browser*** icon, on the left side of the window that pops up find and choose the ***Seismic parameters***:



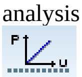
The table on the right shows the seismic parameters set before.

The model is ready for the seismic analysis.
Change tab to ***Static***. A warning dialog window pops up to save the model before the analysis start. (Certainly, if there were previous results of static calculations, they will be deleted.)

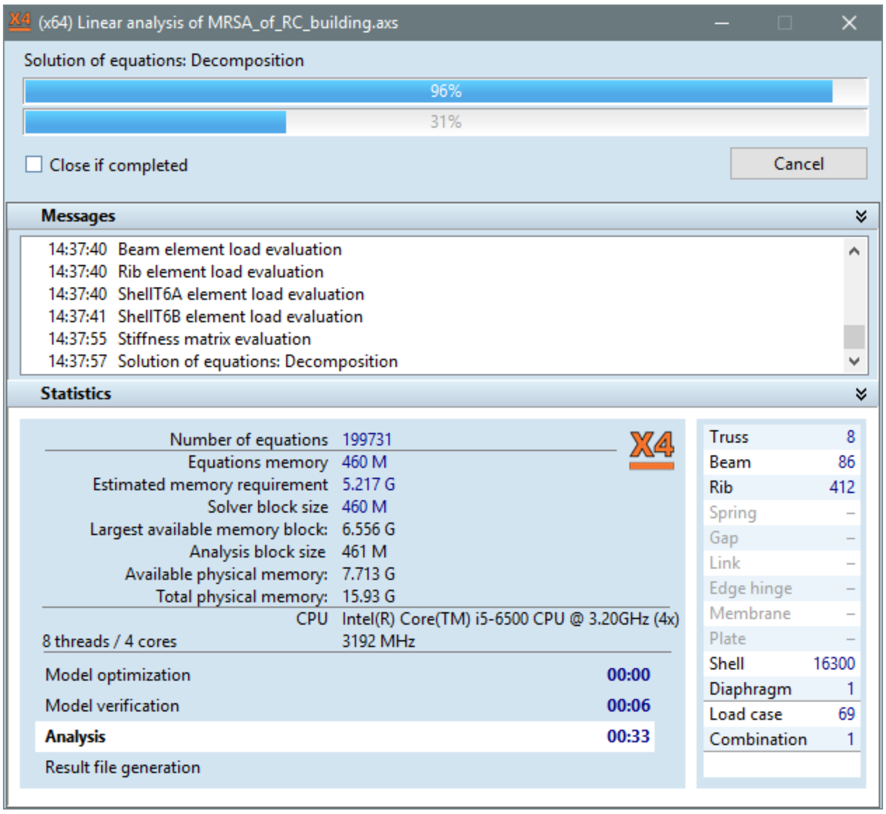


Click on the ***Yes*** button to save the model.

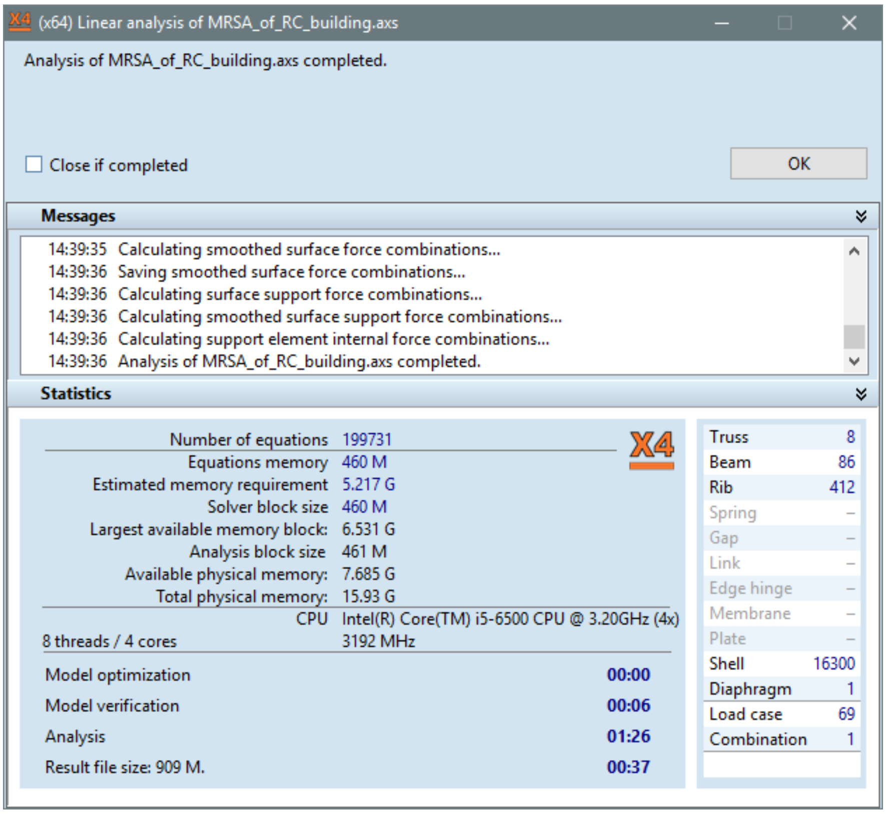
Linear static analysis



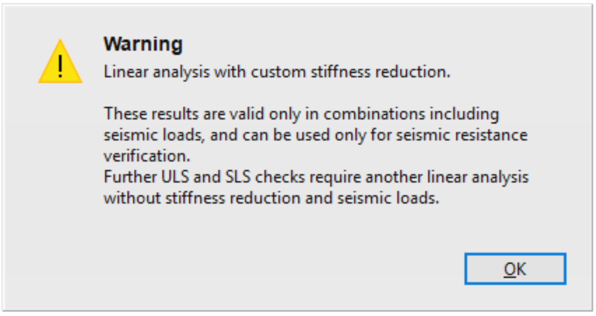
After click on the ***Linear static analysis*** icon, then the calculation starts.



During the analysis, the program gives information about the actual steps of the calculation, the number of the equations, the available memory, etc... in the usual way. When the analysis is completed, click on the ***OK*** button to close the window and to return to the main window.



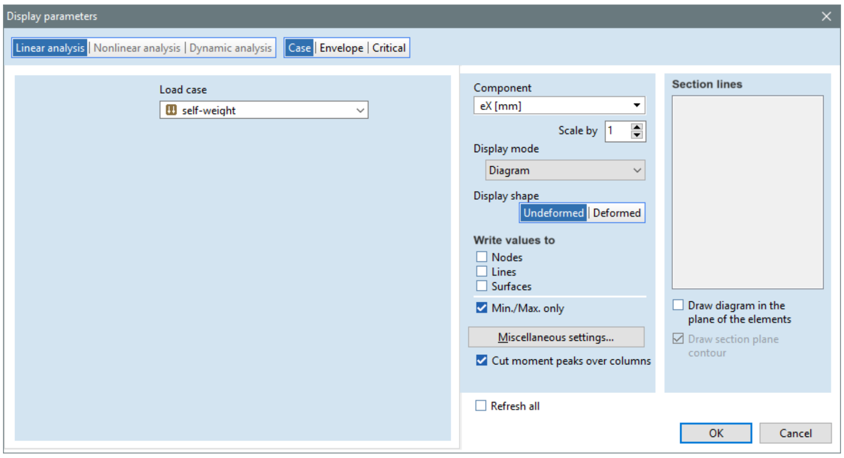
The program shows a warning message about the validity of calculation results because ***stiffness reduction*** has been applied previously for concrete elements.



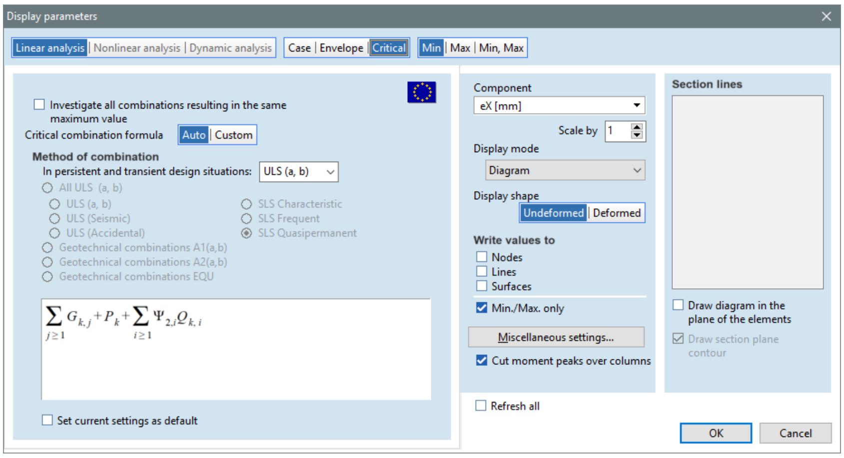
Keep in mind and go on with clicking on the ***OK*** button.

Result display parameters

Considering the above, let us create a seismic combination to inquiring critical values of results. Click on the ***Result display parameters*** icon, then the following window pops up:

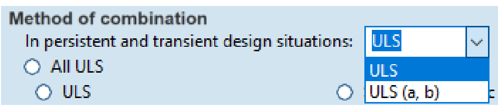


Here, click on the ***Critical*** button below the header, then the combination method of results appears in the left side of the window. By default, the ***Critical combinations formula*** is set to ***Auto***, which contains all the ***ULS*** combinations. In this set the static and seismic type results are also included. Therefore, we have to separate the seismic results.

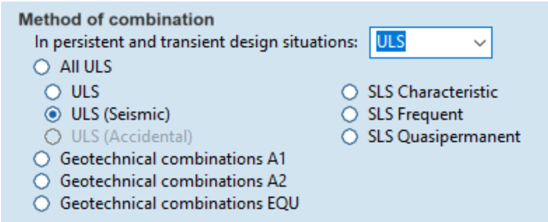


Select ***Custom*** combination, then the program creates different ***ULS*** and ***SLS*** combinations to separate the results.

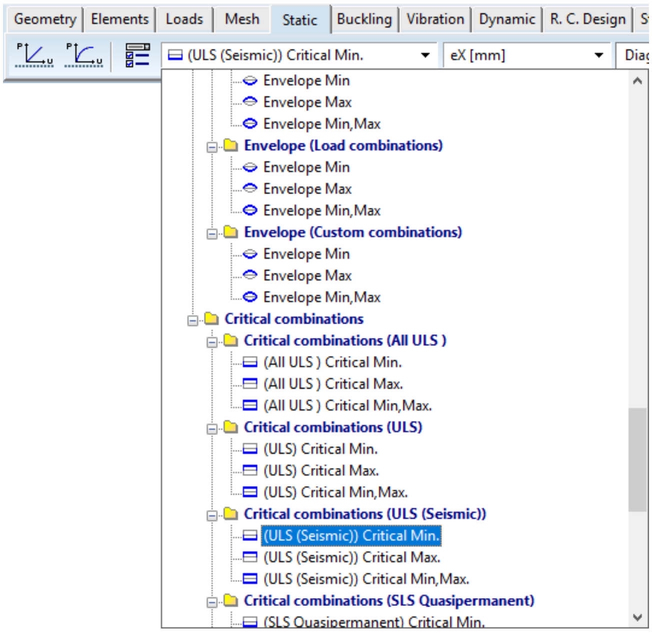
In persistent and transient design situation, there are two options: ***ULS*** and ***ULS (a,b)***. Let us use the default setting - ***ULS***:



In the list of different combinations (being enabled by clicking on the ***Custom*** switch) mark the ***ULS (Seismic)*** combination. The latter will only result that this combination mode will be active when querying results after closing the present window.



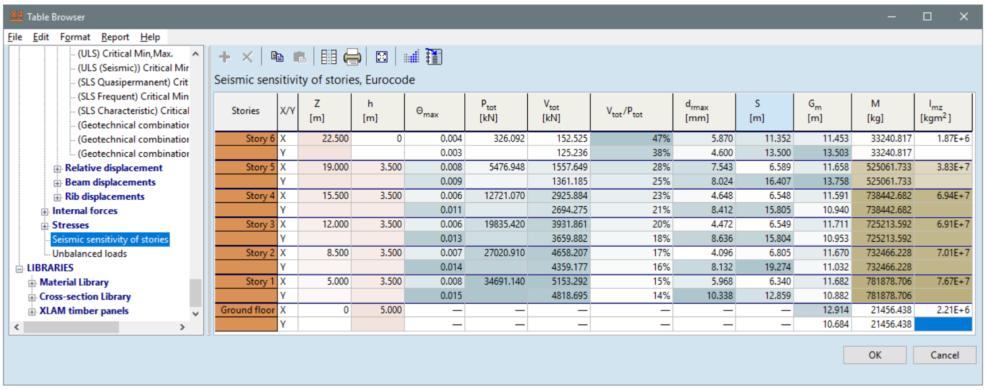
Click on ***OK*** to close the window. As a result of above, the different combination modes appear in the combinations list, including the seismic one. The latter may be used to get the critical results ***(ULS(Seismic)) Critical Min; Max and Min, Max***.



The results may be evaluated in the same way as it was presented before in our previous examples, it is not described in this chapter.

Table browser

Click on the ***Table browser*** icon and have a look at the table of ***Seismic sensitivity of stories*** (this is only available if the torsional effect has been considered in the analysis).



In the table, the stories are summarized showing their height ***Z [m]***, the distance between the stories ***h [m]*** and the following quantities calculated by the program:

θ_{max} – plastic stability index, also known as the interstory drift sensitivity coefficient as per ***EN 1998-1:2008 -1 4.4.2.2 (2)***. Second-order effects (P-Δ effects) need not be taken into account if the **θ ≤ 0.1** condition is fulfilled in all storeys. If **0.1 < θ ≤ 0.2**, the second-order effects may approximately be taken into account by multiplying the relevant seismic action effects by a factor equal to **1/(1 - θ)**. The value of the coefficient **θ** shall not exceed **0.3**. For more details see the relevant chapter in the standard.

P_{tot} – total gravity load above the storey,

V_{tot} – total seismic storey shear,

d_{max} – design interstory displacement calculated as the relative displacement of corresponding storey centroids,

S – location of the shear center. ***This is a result of approximate calculation: AxisVM calculates story shear centers by finding wall sections and using the same method as for thin-walled cross-sections.***

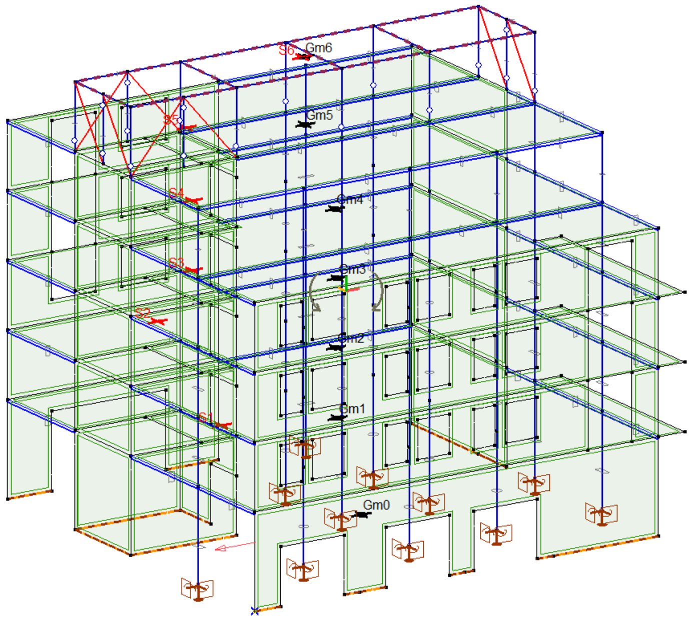
G_m – location of the centroid,

M – storey mass,

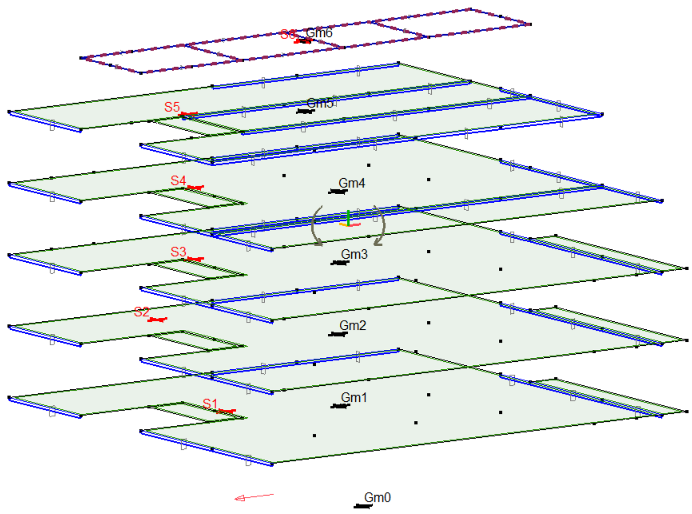
I_{mz} – moment of inertia at the centroid about the ***Z*** axis,

Close the ***Table browser***. The location of the shear center and the centroid is also shown on the finite element model:

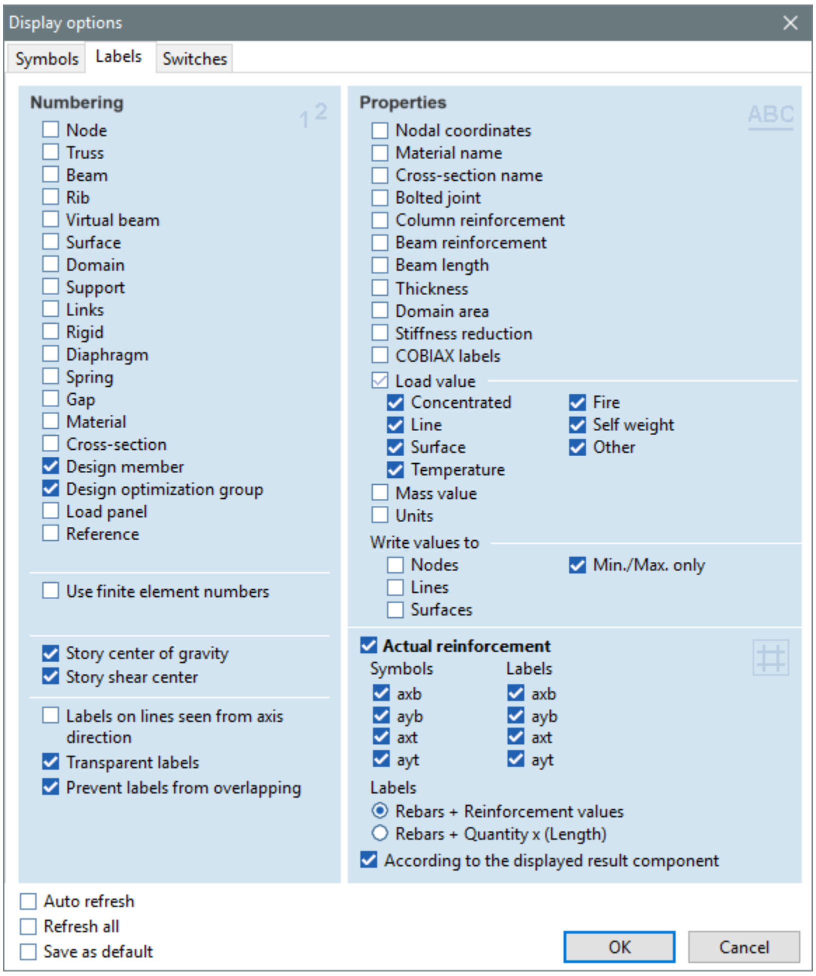
- shear center: ***S1…n*** (displayed as red **+**, with a label ***Si***, where ***i*** is the level number)
- centroid: ***Gm1…n*** (a cross in a black circle with a label ***Gmi***, where ***i*** is the level number).



For better visibility, display only the ***parts of the slabs***:



If the markers of the shear center and the centroid are not visible in the model, it can be switched on in the ***Display options*** (on ***Labels tab*** – ***Story center of gravity***, ***Story shear center***):



Additional steps

The results of the ***MRSA*** can be evaluated in the design modules similar to the simple static results. This procedure is not described in this example.

If it is justified because of the sensitivity of stories, the second order effect can be considered multiplying the relevant seismic action effect with f_{se} factor.

In case of dissipative structures with ***DCM*** or ***DCH*** ductility class, in order to avoid shear failure in RC columns, beams and ribs, the design values of shear forces are determined in accordance with the capacity design rule (if seismic load case is included in the selected load combination). Capacity design is only available if the ***q behaviour factor*** is ***greater*** than ***1.50*** (choosing ***Eurocode – RO*** code, it is available if ***q is greater than or equal to 1.50***).

Notes

Notes