CHAPTER 13. DYNAMICS+ SYSTEM

General

The DYNAMICS+ calculation and graphical system implements the method of direct integration of the equations of motion over time, which makes it possible to perform computer simulation of the structure's response to dynamic loading, both during its action and on its completion. The DYNAMICS+ system is used to solve linear and non-linear problems.

The following types of finite elements are allowed for the calculation of Time-dependent dynamics problems:

- all linear elements;
- unilateral connections (with and without friction);
- soil elements — flat and three-dimensional;
- all physically non-linear elements based on the theory of elastoplasticity;
- all geometrically non-linear elements.

The task of linear dynamic calculation is formulated as:

$$b\left(\frac{\partial^2 u}{\partial t^2}, v\right) + c\left(\frac{\partial u}{\partial t}, v\right) + a(u,v) = (f(t), v), \quad t > 0,$$
$$u(0) = u_0, \quad \partial u/\partial t(0) = u_1$$

This problem is solved by the finite difference method using an unconditionally stable difference scheme of the second order of accuracy:

$$b(\gamma_m u, v) + c(\beta_m u, v) + a(\alpha_m u, v) = (f_m, v);$$
$$t_m = m\theta; \quad u_m = u(t_m); \quad \alpha_m u = \frac{u_{m+1} + u_{m-1}}{2};$$
$$\beta_m u = \frac{u_{m+1} - u_{m-1}}{2\theta}; \quad \gamma_m u = \frac{u_{m+1} - 2u_m + u_{m-1}}{\theta^2}.$$

This is a system of equations for u_{m+1}, the right-hand sides of which depend on u_m, u_{m-1}. The matrix does not depend on m. The values u_0, u_{-1} are determined from the initial conditions.

As a result of the calculation, the displacements, speeds and accelerations of the nodes are being determined, as well as the forces and stresses in the elements calculated at all times t_m.

For a nonlinear problem, a nonlinear term is added to the equation $d(u,v)$, depending on the nature of the nonlinearity, and a term $d(u_m, v)$, which falls on the right side of the difference scheme.

13.1 Initial data for the scheme creating

To create a calculation scheme the checkbox **«DYNAMICS+» system will be used in project** is to be selected in the **Project Parameters** window (Fig. 13.1).

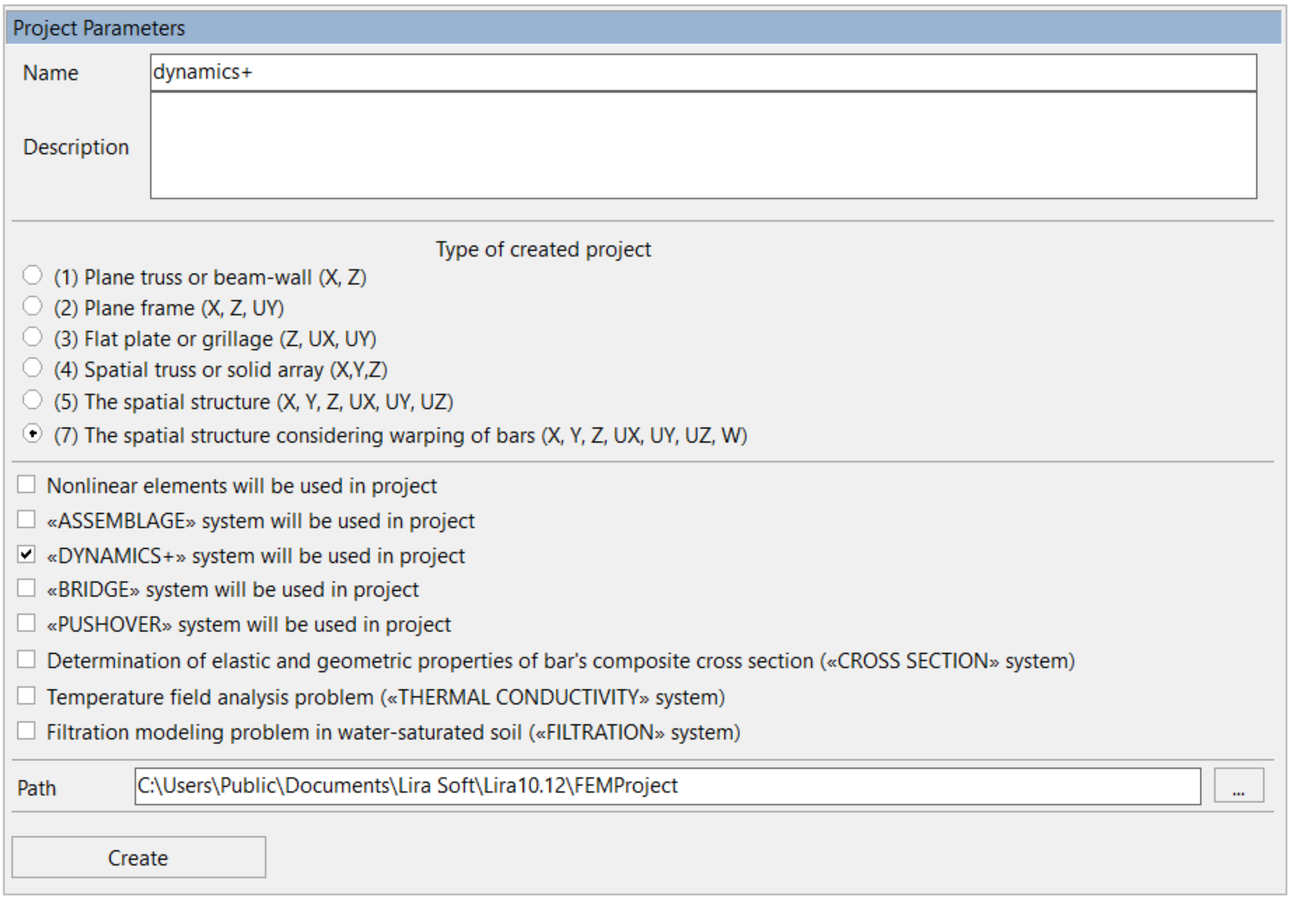


Fig. 13.1. Project Parameters

Calculation considering the Time-dependent dynamics s available both in linear and non-linear formulations, with and without thermal conductivity, as well as with and without installation. Thus, to calculate additional tasks in the **Project Parameters** window (Fig. 13.1), the corresponding checkbox **Nonlinear elements will be used in the project "ASSEMBLAGE" system will be used in project, Temperature field analysis problem ("THERMAL CONDUCTIVITY" system)** must be selected.

13.1.1 Creating of loadings

When starting to create **Loading State** (Fig. 13.2), firstly the **Static loading** must be created, which can later be converted to masses to form **mass matrix for the dynamic loading**. If it`s necessary to create more than one static loading, add **Concurrent static loading** to previously created one.

Note that **Static loading** and all concurrent to it will be combined into one loading during the calculation. Thus, in the calculation results there will be no possibility to display the results for each static load case separately.

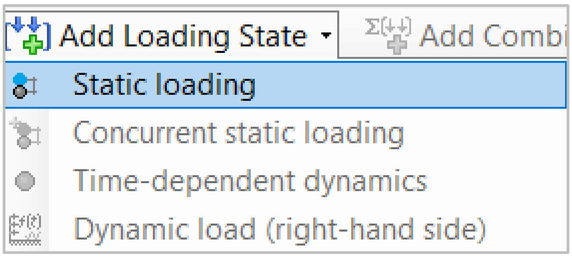


Рис. 13.2. Add Loading State

After adding **Static loading**, the **Time-dependent dynamics** addition can be used. To use it, the appropriate item in the **Add Loading State** drop-down list is selected (Fig. 13.2). In the window that appears (Fig. 13.3), the **Integration step** and **Integration time** are set, on the basis of which the minimum number of time points will be obtained, for each of which the results will be generated.

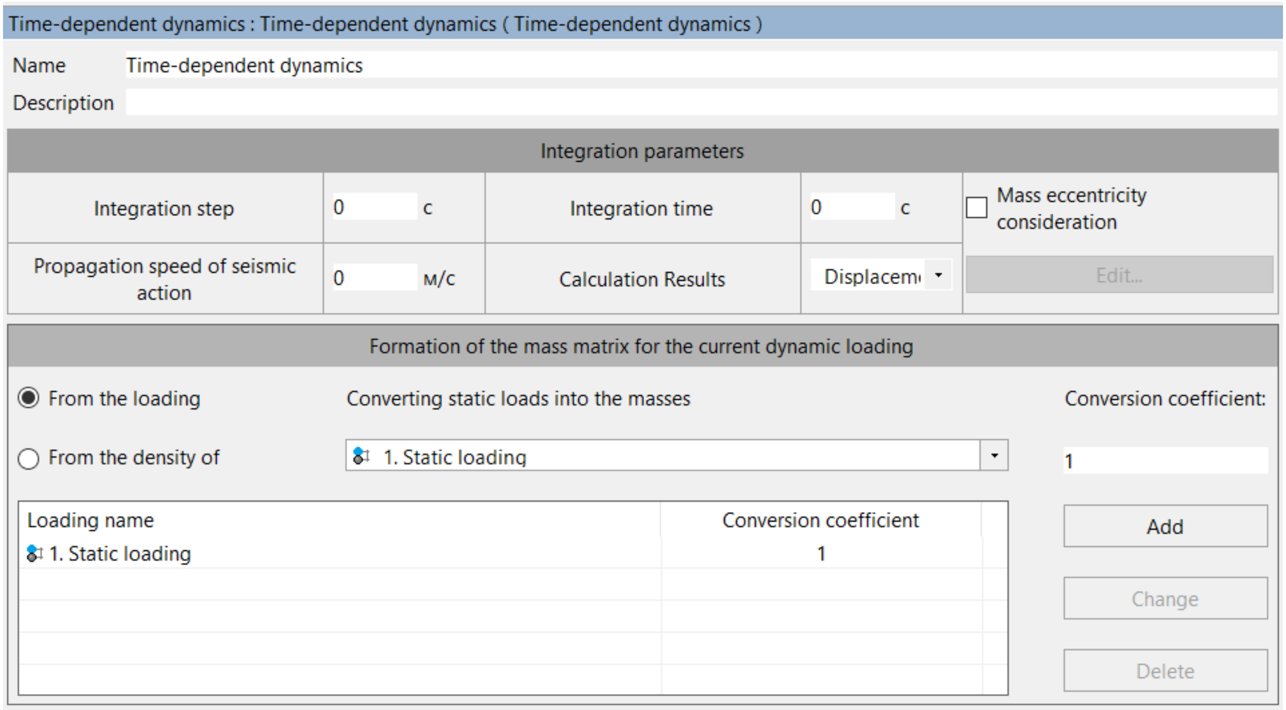


Fig. 13.3. Specifying data for Time-dependent dynamics

Setting of the **Propagation speed of seismic action** is required in case of setting a load of Seismogram- type. In this case, the Seismogram will be applied to the selected nodes with a delay in time equal to the time it takes for the seismic wave to travel from the extreme horizontal point on the diagram to the point where the seismogram is applied.

Next, you need to select the resulting **Calculation Results** from the drop-down list:

- Displacements only;
- Displacements and internal forces;
- Displacements, internal forces and DCF;
- Displacements, internal forces, DCF and stresses;
- Temperature field only.

When solving the problem of non-stationary heat conduction, the temperature will be calculated in any case.

When solving nonlinear problems, stress result files can take up a lot of hard disk space. This must be taken into account when ordering the results obtained from the drop-down list.

To take into account the effects of random twist, it is necessary to select **Mass eccentricity consideration** and set, by clicking on the **Edit…** button, mass eccentricities in the horizontal and vertical directions, for the entire scheme or for selected floors. If eccentricities are set for specific floor numbers, then these floors must be previously created in **Editors and Structural Design ⇨ Groups of Elements** (for details on creating the floors, see p. 2.11.10).

The formation of the mass matrix can be carried out by **Converting static loads into the masses** or by getting the masses **From the density of**. To do this, select the required criterion, set the **Conversion coefficient** in the corresponding field and click the **Add** button. When converting static loads to masses, you need to select a load case from the drop-down list for which the conversion will be performed. When solving problems with the assembly, masses can only be collected from the last stage of assembling.

The conversion coefficient of the resulting masses can be edited. To do this, select the corresponding line, enter a new coefficient value and click the **Change** button. Deleting masses is carried out by selecting the appropriate entry and pressing the **Delete** button.

The **Time-dependent dynamics** load includes the following load cases:

- **Dynamic load (nodal forces)**;
- **Damping**;
- **Dynamic load (right-hand side)**.

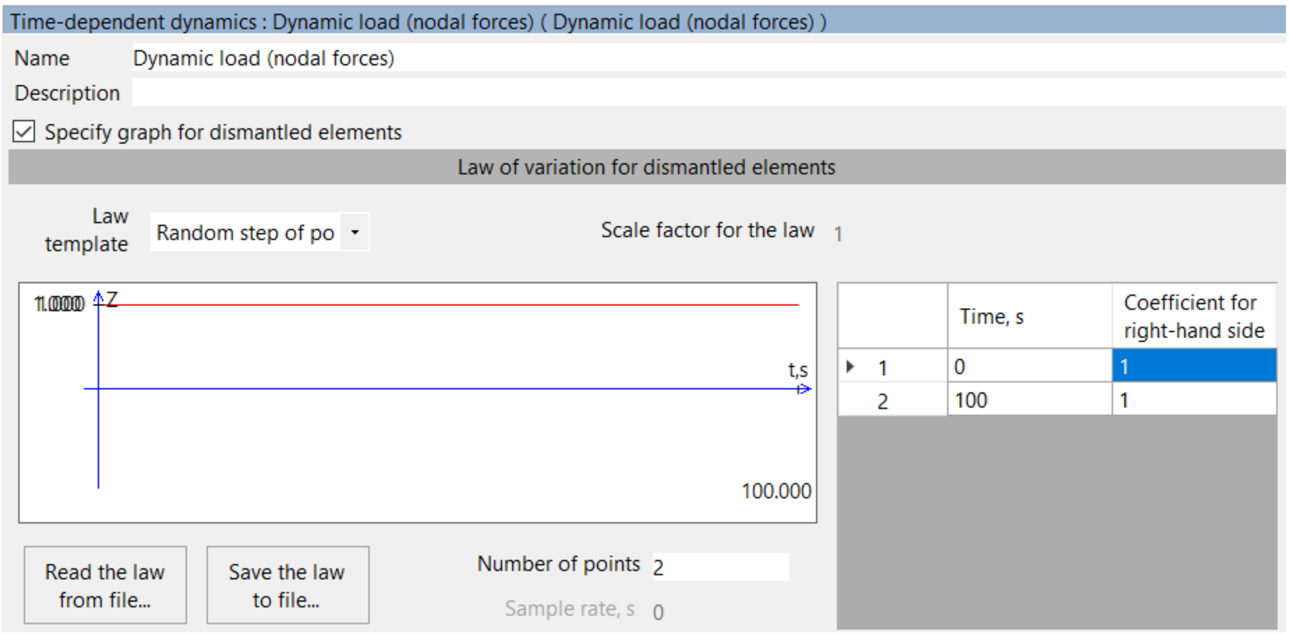


Рис. 13.4. Graph for dismantled elements

When solving problems of dynamic dismantling, in **Dynamic load (right-hand side)** (Fig. 13.4) it is necessary to set the schedule of dismantled elements (coefficient to the right side of the reactions in the nodes of dynamically dismantled elements). It is possible to create a graph as

Random step of polyline line, Even step of polyline line, Polyligonal line as a time-dependent function. To do this, select the appropriate item in the **Low template** drop-down list. Then, to build a graph, specify the **Number of points** and fill in the table that appears on the right. It is also possible to write graphics to a file, and read from a file.

If it is necessary to take into account **Damping** (Fig. 13.5), go to the corresponding item **Loading States Library**, and set the **Consider Rayleigh proportional damping** checkbox. Damping can be set in one of three ways:

1. Direct indication of the coefficients α and β.
2. Calculation of coefficients through two modes of oscillation.
3. Taking into account the damping properties of individual materials. To do this, the damping α and β coefficients must be set in the material editor and the **Consider damping properties of the material** checkbox is to be enabled (Fig. 13.5). In the case of simultaneous consideration of the general damping and material damping, when constructing the damping matrix, the coefficients α and β will be summed up.

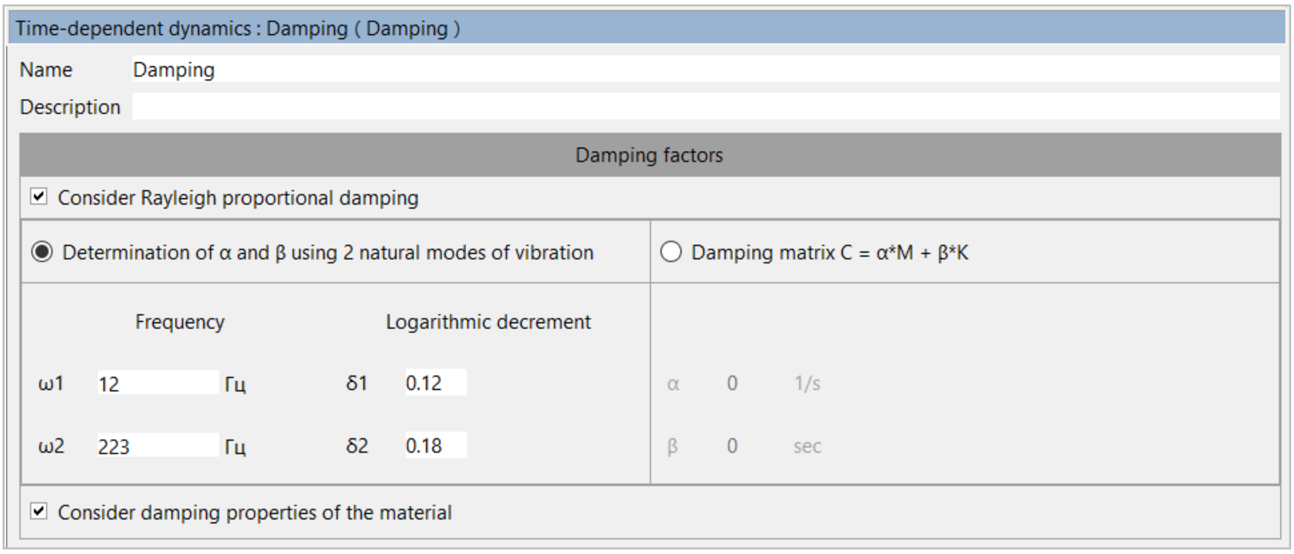


Fig. 13.5. Damping

When calculating α and β through the frequencies of natural oscillations and logarithmic decrements of oscillations, the dependence is used, which is obtained from the system of equations:

$$\begin{cases} \alpha + \beta \cdot \omega_1^2 = 2 \cdot \omega_1 \cdot \sqrt{\dfrac{1}{\dfrac{4 \cdot \pi^2}{\delta_1^2} + 1}}; \\ \alpha + \beta \cdot \omega_2^2 = 2 \cdot \omega_2 \cdot \sqrt{\dfrac{1}{\dfrac{4 \cdot \pi^2}{\delta_2^2} + 1}}. \end{cases}$$

When calculating α and β through the frequencies of natural oscillations and logarithmic decrements of oscillations, the dependence is used, which is obtained from the system of equations:

$$\alpha = \frac{2 \cdot \sqrt{\frac{1}{\frac{4 \cdot \pi^2}{\delta^2} + 1}} \cdot \omega_1 \cdot \omega_2}{\omega_1 + \omega_2};$$

$$\beta = \frac{2 \cdot \sqrt{\frac{1}{\frac{4 \cdot \pi^2}{\delta^2} + 1}}}{\omega_1 + \omega_2},$$

where ω_1, ω_2 — the minimum and maximum value of the angular frequency of oscillations from the range studied;

δ — logarithmic decrement.

δ_1 and δ_2 — logarithmic decrements, which correspond to the frequencies ω_1 and ω_2.

Be careful! When carrying out the calculations of damping, it can also be specified through other quantities that should not be confused with each other:

- *Ψ — absorption coefficient;*
- *γ — inelastic material resistance coefficient;*
- *ξ — damping factor (in the literature it can also be called «damping ratio», «damping coeficient»)*

Along with the Logarithmic decrement δ these four parameters are interconnected by dependence:

$$\gamma = \frac{\psi}{2 \cdot \pi} = \frac{\delta}{\pi} = \frac{2 \cdot \xi}{\sqrt{1 - \xi^2}}.$$

When assigning the frequency range ω1, ω2, as a rule, the range of natural oscillations of the structure is used, in which a sufficient amount of modal mass is accumulated. If to choose the entire range of fluctuations of the structure from 0 to 100% of the modal mass, then this approach will be considered as conservative. With smaller ranges, some frequencies will be counted as "with a margin", while others will be counted as not "with a margin". The choice of one or another range of natural frequencies ω1, ω2 must be substantiated by the engineer independently.

The last component of the load **Time-dependent dynamics** is **Dynamic load (right-hand side)**, which allows the user to specify the general law of force or thermal load changes in time. To set the right side, first select **Time-dependent dynamics**, and then click the **Add Loading State** (Fig. 13.2) and select the appropriate item from the drop-down list.

In the workspace of this mode, it is possible to set the law of forces action manually or read it from a file. To load from a file, click the **Read the law from file…** button and select the appropriate file on the disk in the dialog box that appears. In the same way, it is possible to save the created law using the **Save the law to file...** function.

There are 3 templates available for setting the law manually, the template is selected from the corresponding drop-down list. When using the **Random step of polyline line** template (Fig. 13.6), it is only needed to specify the **Number of points**. In this case, the time intervals between points on the graph can be arbitrary. To complete the formation of the law, it is necessary to fill in the table, indicating **Time** and **Coefficient for the right-hand side** in it.

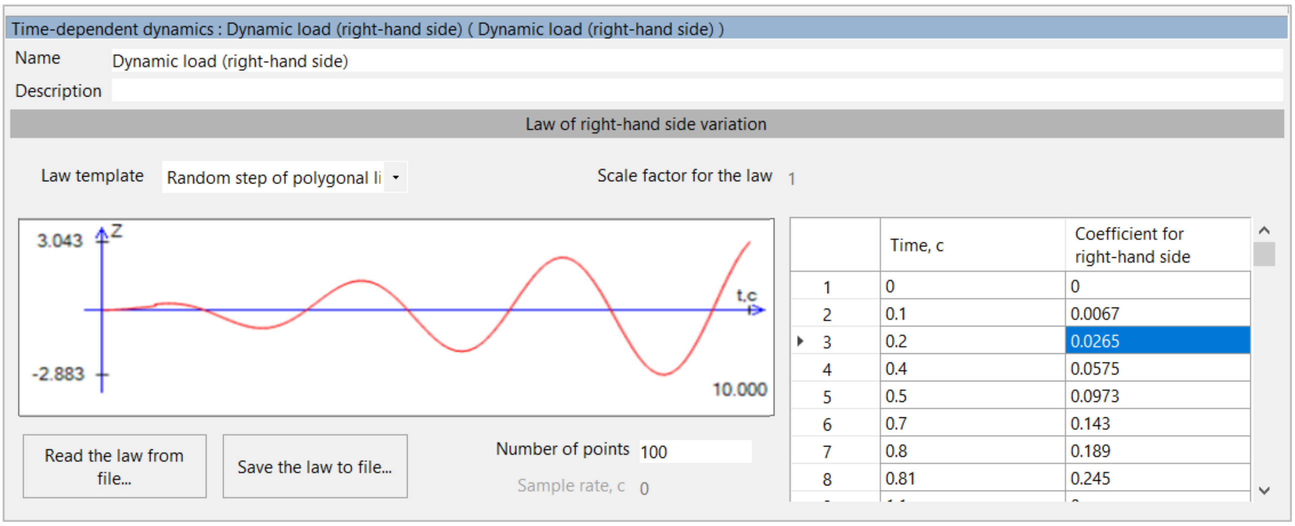


Fig. 13.6. Random step of polyline line

When choosing the **Even step of the polyline line** template (Fig. 13.7), it is crucial to additionally specify the **Sample rate**, so the first column of the table will be filled in automatically, unlike the previous option. When this template is selected, it becomes possible to specify the **Scale factor for the law.**

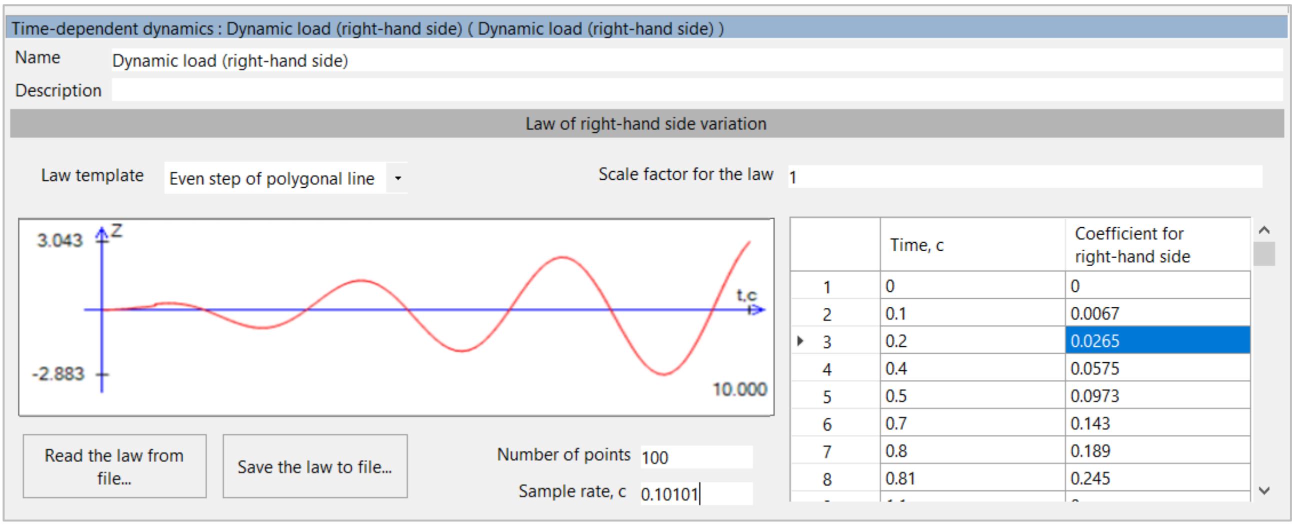


Fig. 13.7. Even step of the polygonal line

The final law template is the **Polyligonal line as a time-dependent function** (Fig. 13.8). To set the law in this way, select the appropriate template from the drop-down list, and set the function Z(x), where *x* is time. It is also necessary to specify the start time of the function and the duration. Attention should be paid to the fact that the function time is always set in seconds, frequencies - in rad/s, angles - in rad.

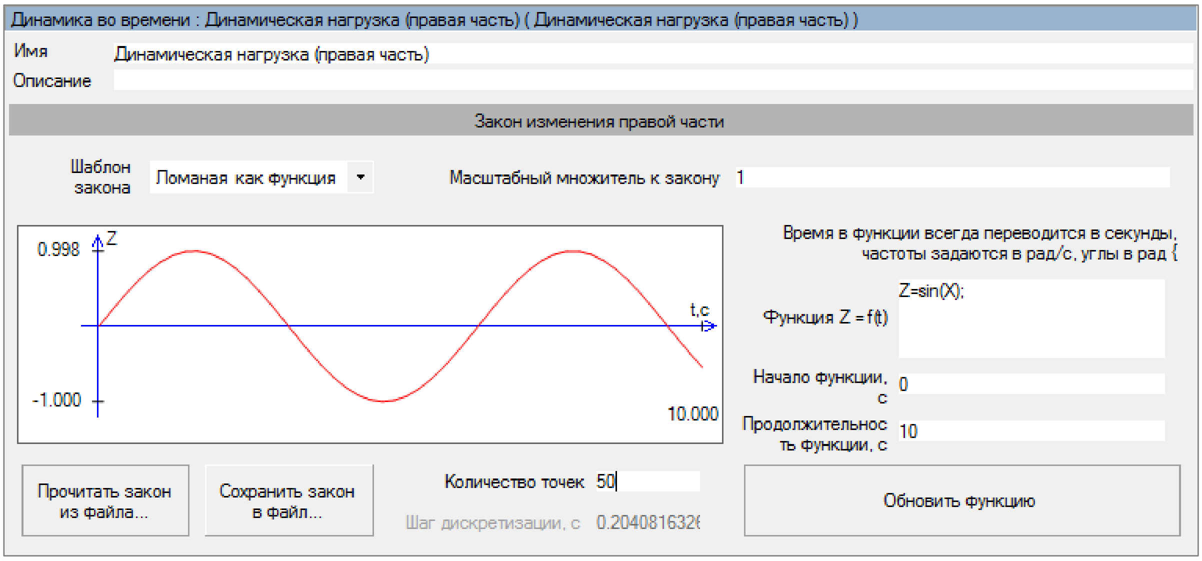


Fig. 13.8. Polyligonal line as a time-dependent function

13.1.2 Loads assignment

Loads for static load cases are similar to loads from static problems, and they are assigned similarly.

For loading **Time-dependent dynamics** (Fig. 13.9), it is possible to assign weights of dynamic and nodal masses to elements and nodes of the scheme, respectively (Fig. 13.10).

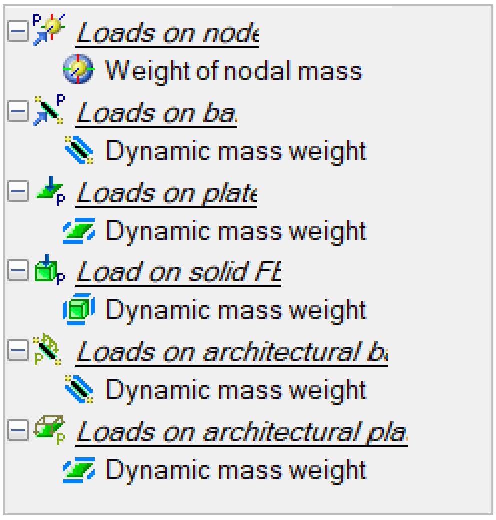


Fig. 13.9. Loads: **Time-dependent dynamics**

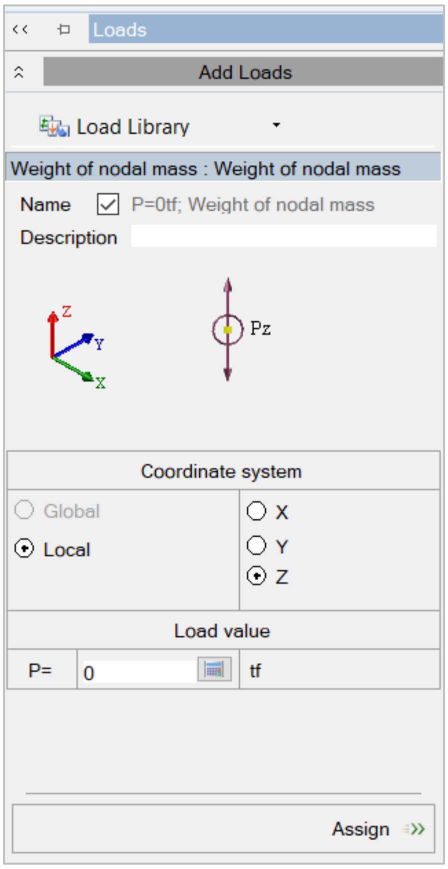


Fig. 13.10. Assignment of nodal mass weight

To assign the weight of dynamic masses on the appearing mode panel, enter the value **P**, select the elements and press the **Assign button**. Also, if it is necessary to take into account the effect of the load on the rigid insert (if its projection lengthens the flexible part), it can be specified in the mode window by clicking the checkbox.

For loading the **Dynamic load** (**nodal forces),** it is possible to assign loads only to nodes and as a function of time (Fig. 13.11). Due to this loading, it is possible to set loads that act according to different laws, in contrast to the right-hand side setting.

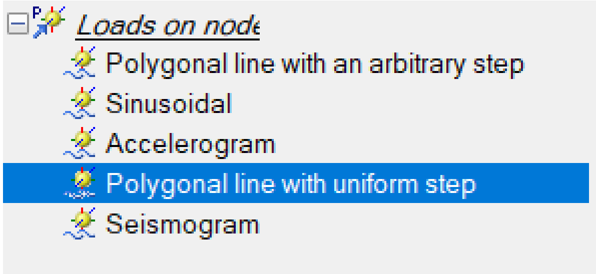


Fig. 13.11. Loads. Nodal forces

The loads available in the **Damping** load case are similar to the loads available in the static load case, except that the specified displacements and rotations cannot be assigned. The loads assigned in this load case will be interpreted as dampers during the calculation and will be included in the damping matrix.

All constant loads are available when assigning the **right-hand side**, except for the specified displacement and rotation. Despite the fact that these loads are constant, during the calculation, they will change according to the law specified in the **Dynamic load (right-hand side) load case**.

13.1.3 Seismic Ground Motion Records Conversion Utility

The utility for converting records of seismic ground motion becomes available when loading **Dynamic load (nodal forces)** and selecting the appropriate load on the node. For the **Accelerogram** load, transformation from the existing seismogram is available, and for the **Seismogram** load, on the contrary, it is available from the existing accelerogram (Fig. 13.12).

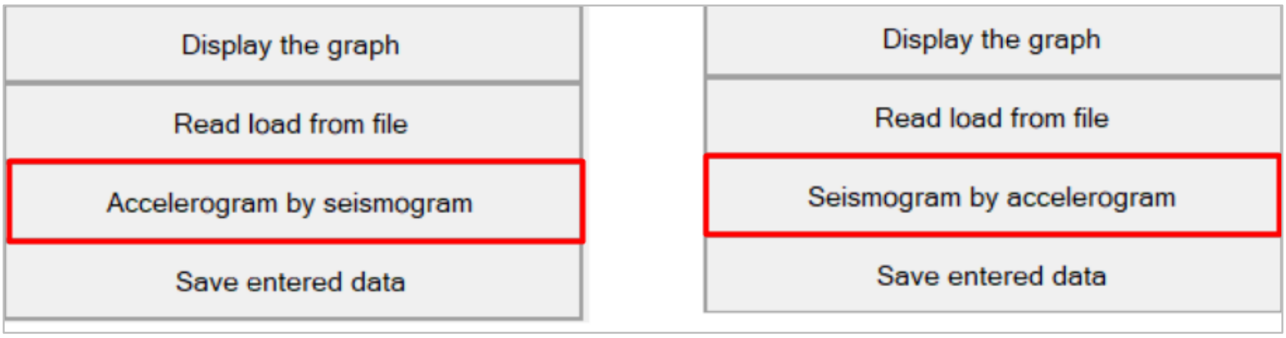


Fig. 13.12. Utility launcher buttons

To calculate a seismogram/accelerogram, select the required file and, if necessary, set additional calculation conditions (**Accelerogram's trend removal, Method of obtaining**

accelerogram, Seismogram's trend removal, Rotation of zero Line). Each of the obtained results can be visualized.

- **Accelerogram's/ Seismogram's trend removal** - in this case, the average values obtained by linear, quadratic or cubic approximation are subtracted (Fig. 13.13).

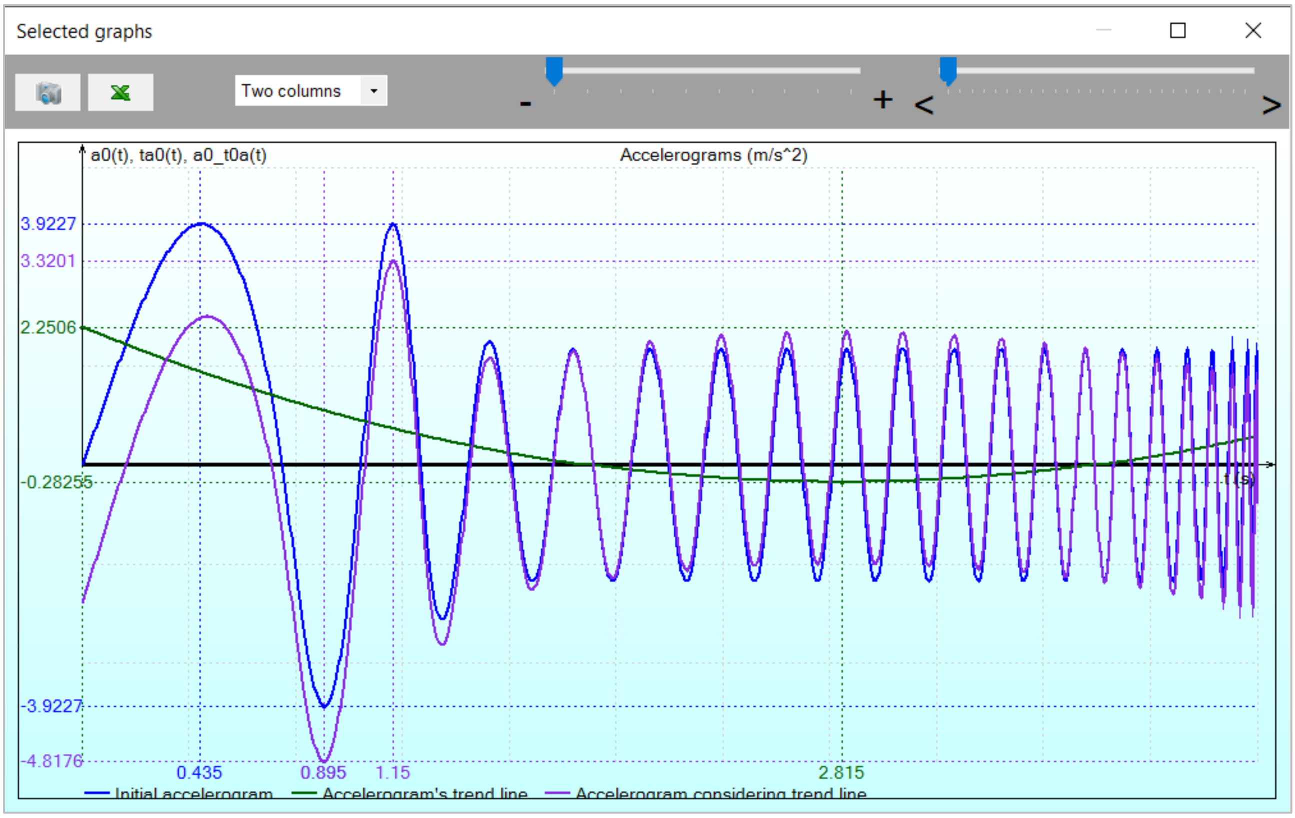


Fig. 13.13. Removing a Trend Using Cubic Approximation

- **Method of obtaining accelerogram/ seismogram** is a set of formulas and methods for data transformation:
  - **Uniform acceleration Method** — kinematic data transformation, in which the acceleration vector remains unchanged in magnitude and direction. Only available for seismogram calculation from accelerogram.
  - **Central difference method** — finite difference approximation, the original differential equation is replaced by a finite difference relative to the grid function.
  - **By Fourier transformation** — transformation of the accelerogram function into frequency components using the DFT algorithm.
- **Rotation of zero Line** — subtraction from the input data of the coordinates of the straight line passing through the first and last points of the original chart (Fig. 13.14).

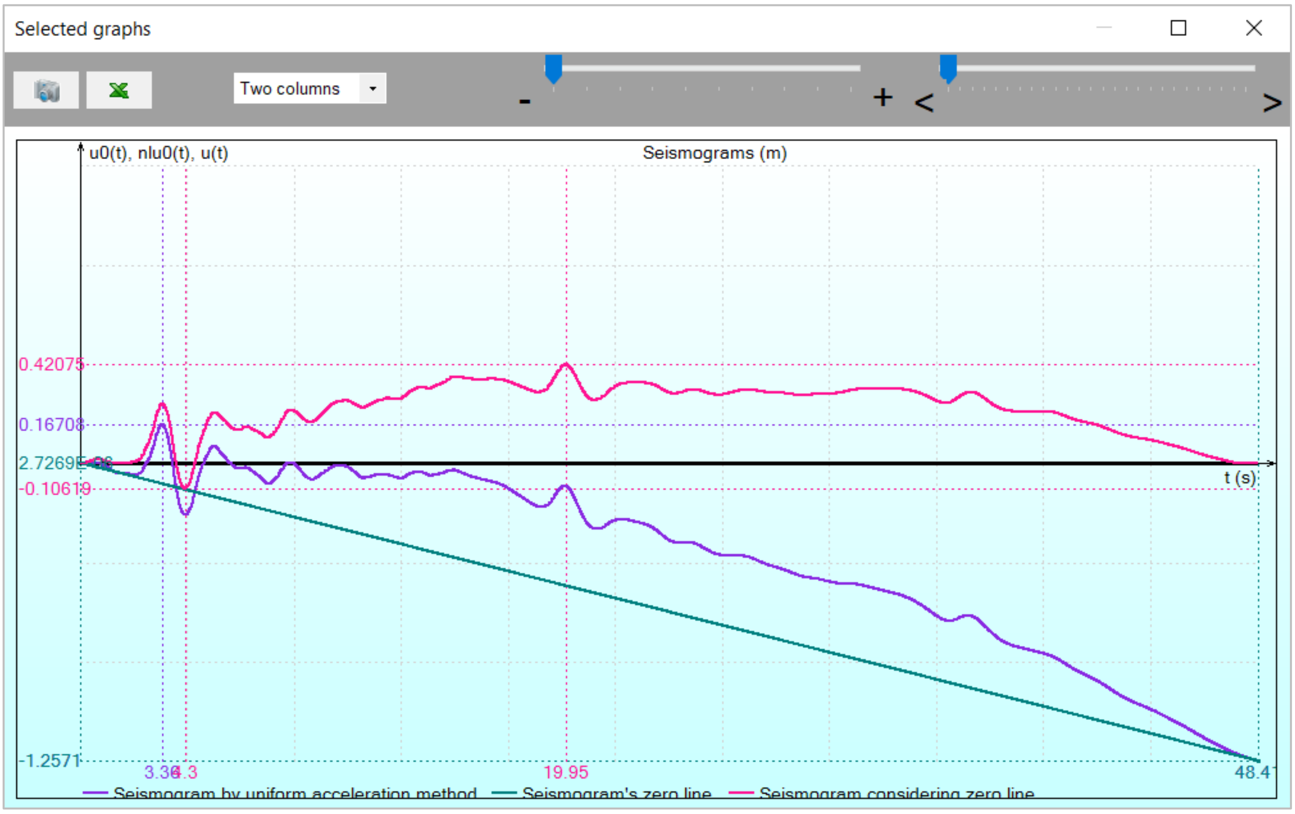


Fig. 13.14. Rotation of zero Line

In addition to the functions, mentioned above, the utility provides for the selected earthquake accelerogram the possibility of analyzing the frequency composition using the Fourier transformation, obtaining response spectra of accelerations, velocities and displacements at specified relative damping (Fig. 13.15).

Please note that the visualization of the initial data and the obtained results is possible only for graphs with the similar units of measurement.

The **Seismogram** obtained as a result of transformations can be used in the calculation for seismic actions in dynamics in time (**DYNAMICS+**), and the calculated reaction spectra of accelerations, velocities and displacements can also be used in the calculation for seismic action according to the response spectrum of a single-mass oscillator for the method of expansion in terms of natural vibrations.

Saving the results with the specified accuracy is carried out in files:

- **.ar1** (for *accelerogram* under seismic load **Seismogram**);
- **.sr1** (for *seismogram* under seismic load **Accelerogram**);
- **.vr1** (for *velocigram*, irrespective of seismic load representation);
- **.rsa** (for *acceleration response spectrum*);
- **.rsv** (for *speed response spectrum*);
- **.rsd** (for *the displacement response spectrum*).

When graphs are displayed, it is possible to create a copy of the screen image (by clicking on  button) and export data to an .xlsx file (by clicking on  button), see Fig. 13.16.

✍ *The Fourier transformation and the calculation of response spectra are available when choosing the original or received accelerogram, regardless of the choice of seismic load representation.*

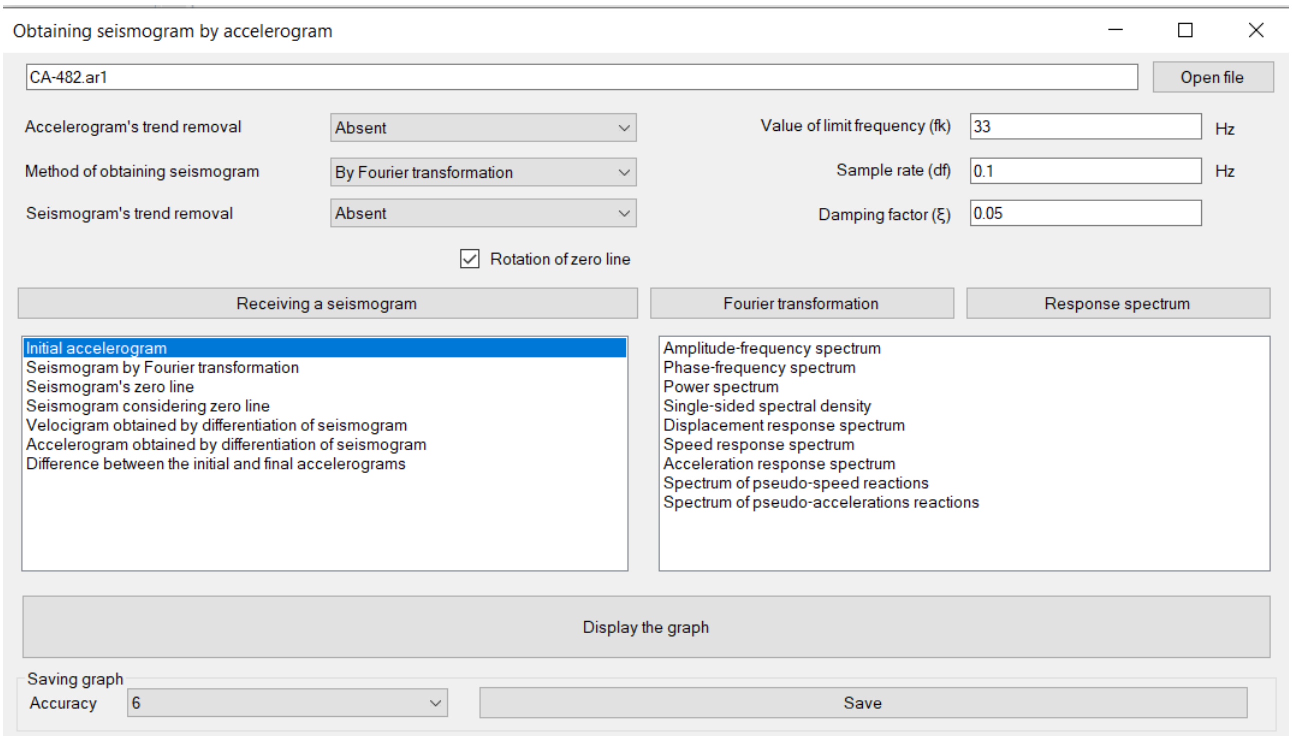


Fig. 13.15. Fourier transformation and getting response spectra of the specified accelerogram

	A	B	C
1	Frecuency, (Rad/s)	Initial accelerogram, m/s^2	Difference between the initial and final accelerograms, m/s^2
2	0	-0.013999776	0.039999366
3	0.125663703	-0.107998282	-0.028749542
4	0.251327407	-0.100998394	-0.001499976
5	0.37699111	-0.087998599	0.00499992
6	0.502654813	-0.094998486	0.004499929
7	0.62831854	-0.11999809	-0.000749991
8	0.75398222	-0.14199774	-0.008999852
9	0.879645947	-0.127997965	-0.000999987
10	1.005309627	-0.109998249	-0.001749973
11	1.1309734	-0.084998645	0.006249901
12	1.25663708	-0.084998645	0.011499817
13	1.38230076	-0.130997911	-0.000249997
14	1.50796444	-0.175997198	-0.006749895
15	1.63362812	-0.193996906	-0.012499796
16	1.759291894	-0.161997423	0.003499939
17	1.884955667	-0.143997699	-0.004499923
18	2.010619253	-0.107998282	0.002499957
19	2.136283027	-0.081998691	-0.003499943
20	2.2619468	-0.041999329	0.015999746
21	2.387610387	-0.065998949	0.010249837
22	2.51327416	-0.130997911	-0.001499979
23	2.638937747	-0.189996973	-0.013249785
24	2.76460152	-0.195996881	-0.033999462
25	2.890265294	-0.065998949	0.008499866
26	3.01592888	0.029999522	-0.003749942
27	3.141592654	0.140997753	0.0752488

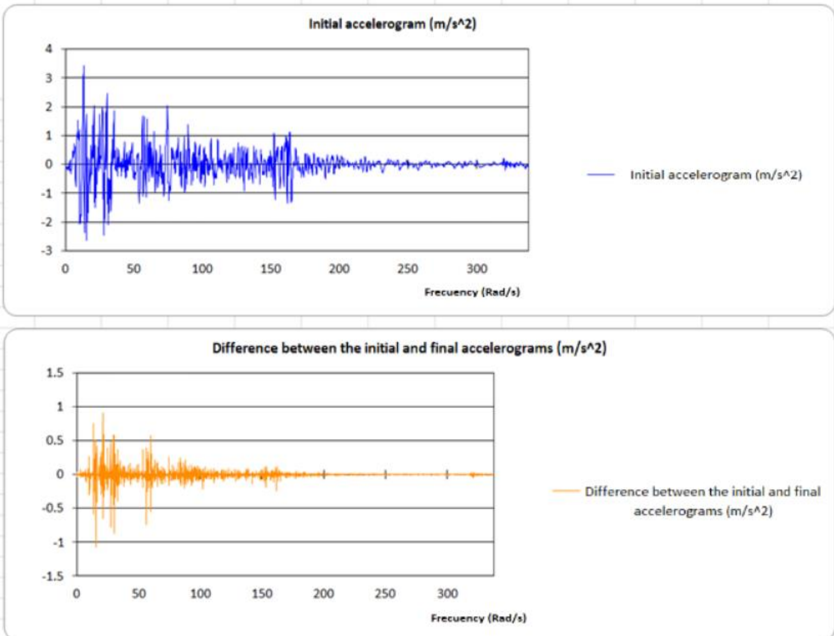
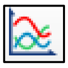


Fig. 13.16. Data export to .xlsx file

13.2 RESULTS ANALYSIS

For static load cases in the **Calculation results** mode, data is displayed for the sum of all static load cases. For the total load case, you can display the displacements of the nodes, the forces in the bars and plates, the forces in the special. elements, stresses in solid elements, DCF results, principal and equivalent stresses in plates and solid elements.

Calculation results of the **Time-dependent dynamics** are displayed for each point in time. In addition to the results described above, it is possible to display **Accelerations, Velocities, Displacements and Constructing Response Spectrum** graphs.

To construct a response spectrum, execute the **Special Result ⇨ Constructing Response Spectrum** command or press the button  In the appearing window (Fig. 13.17), specify the nodes for which the response spectrum will be built by selecting them on the diagram and clicking the **Replenish List of Nodes** or **Replace List of Nodes**, if it is necessary to change an existing one. Next, specify **Direction** (X, Y, Z, UX, UY, UZ) and **Response Spectrum** (**Displacements**, **Velocities**, **Accelerations, Pseudo velocities, Pseudo Accelerations**). It is important to select the damping criterion (**Logarithmic decrement** or **Damping factor**) and its value by clicking the checkbox next to the value, needed. If the numerical value of the damping criterion does not match the proposed list, then it can be set manually: by checking the box on the value **0**, then it becomes possible to enter an arbitrary number (Fig. 3.17). The last parameter is an indication of the value that will be plotted along the abscissa - **Frequency** or **Period.**

If it is necessary to set the **Value of limit frequency** and **Fragmentation of standard steps** it`s worth using the **Parameters of construction** tab.

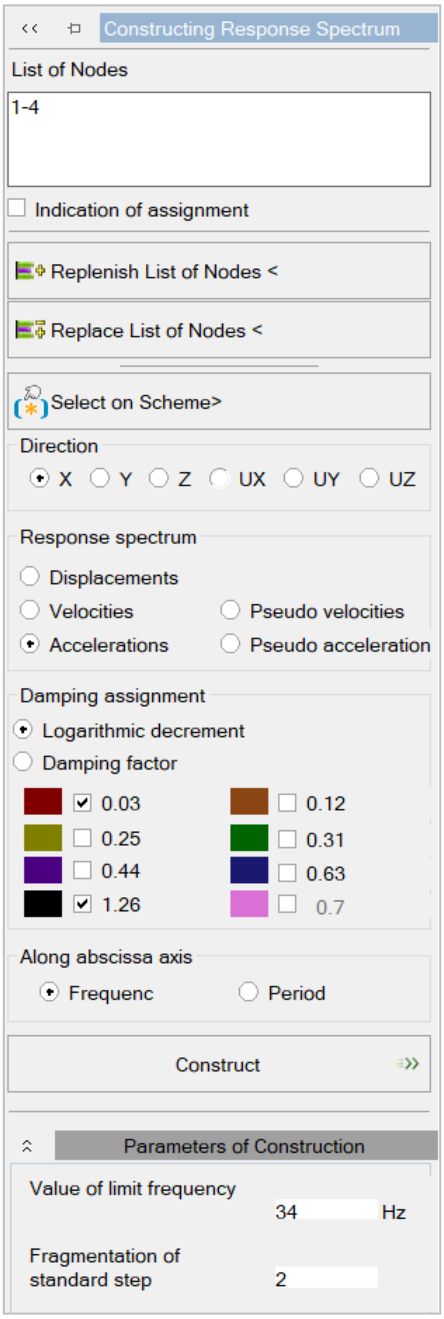


Fig. 13.17. Construction of a response spectrum

When constructing a response spectrum for two or more nodes, the total response of all nodes is calculated as the arithmetic mean of the responses at each node.

The resulting response spectrum (Fig. 13.18) can be imported into MS Excel, saved to a file, or saved as an image. Also, using two scroll bars, the scaling and moving of the spectrum can be done.

To plot accelerations, velocities, and displacements versus time, switch to the **Resuts for Nodes** mode (using the **Results ⇨ Nodes** or by clicking  button on the toolbar) and display the **Plots** tab on the active mode panel (Fig. 13.19). On this tab, you need to specify the direction and output factors (displacements, speeds, accelerations). After that, select the node for which the graphs will be displayed by clicking on it with the left mouse button.

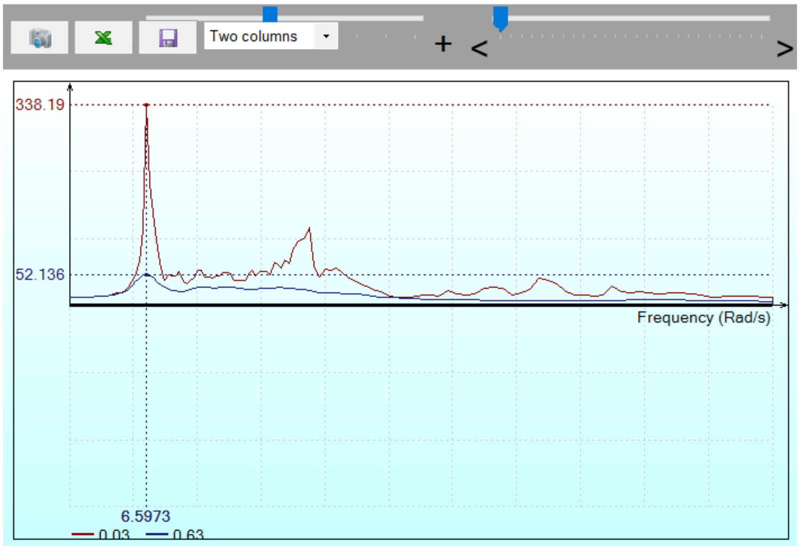


Fig. 13.18. Response spectrum

To plot force-time dependency graphs for bar, plate, or solid elements, switch to **Resuts for Bars, Resuts for Plates, or Resuts for Solid FE**. In the mode window that appears (Fig. 13.20), go to the **Plots** tab and click the checkboxes for the forces that will be displayed. Next, click on the element for which the graphs will be built.

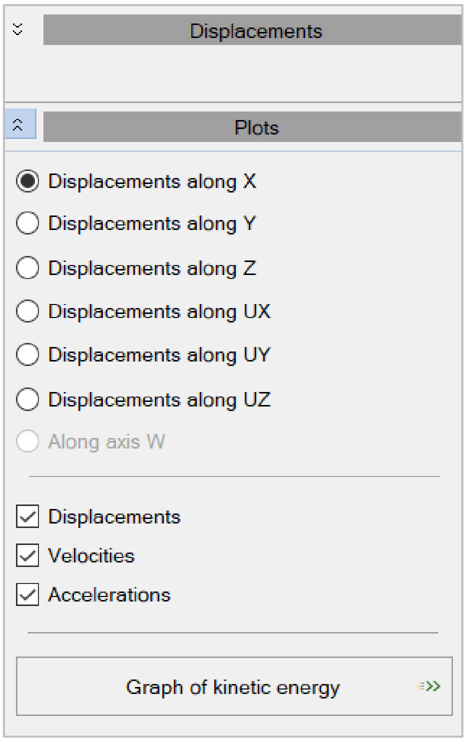


Fig. 13.19. Graph plotting

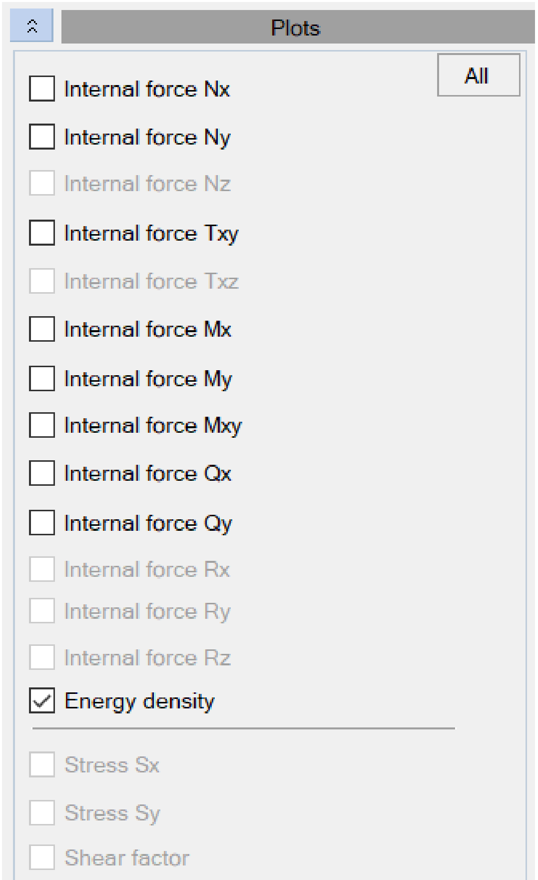


Fig. 13.20. Plate forces