
Version
January 2020

Add-on Module

# RF-DYNAM Pro

Natural Vibration Analysis, Response Spectra, Time History, Equivalent Static Forces

## Program Description



**© Dlubal Software GmbH 2020**
**Am Zellweg 2**
**93464 Tiefenbach**
**Germany**

**Tel.: +49 9673 9203-0**
**Fax: +49 9673 9203-51**
**E-mail: info@dlubal.com**
**Web: www.dlubal.com**

# Contents

## Contents

# 1 Introduction

## 1.1 Add-on Module RF-DYNAM Pro

Whether you are a beginner or already an expert user working with one of the previous RF-DYNAM versions, everybody can quickly learn how to use the new add-on module. All the valuable hints from our customers telling us about their everyday experience helped us to develop and improve this add-on module. There are many new features available, which improves and enhances the possibilities of a dynamic analysis.

The RF-DYNAM Pro module is split into four parts: The add-on module *RF-DYNAM Pro - Natural Vibrations* is the basic module that performs natural vibration analyses for member, surface and solid models. A multi-modal and multi-point response spectra analysis and a linear time history analysis of the given structure can be performed with the module *RF-DYNAM Pro - Forced Vibrations*. A nonlinear time history analysis is available in the add-on module *RF-DYNAM Pro - Nonlinear Time History*. The add-on module *RF-DYNAM Pro - Equivalent Loads* covers the equivalent static force analysis (based on the multi-modal response spectra analysis) in accordance to various building standards.

We hope you enjoy working with RF-DYNAM Pro.

Your DLUBAL team

### Natural Vibrations

*RF-DYNAM Pro - Natural Vibrations* determines the lowest eigenvalues of the model. The number of the eigenvalues can be adjusted. Masses are directly imported from load cases or load combinations (with the option to import the total mass or only the Z-component of loads). Additional masses can be defined manually at nodes, lines, members or surfaces. Furthermore, you can influence the stiffness matrix by importing axial forces or stiffness modifications of a load case or a load combination.

The main features are listed below:

- Automatic consideration of masses from self-weight
- Direct import of masses from load cases or combinations
- Optional definition of additional masses (nodal, line, member, and surface masses as well as mass moments of inertia)
- Combination of masses in different mass cases and mass combinations
- Pre-set combination factors according to EN 1998-1 CEN
- Masses can be neglected on arbitrary parts of the model
- Optional consideration of normal forces to modify the geometric stiffness matrix
- Stiffness modification (for example, you can import deactivated members or stiffnesses from *RF-CONCRETE*)
- Consideration of failed supports or members possible as initial conditions
- Definition of several natural vibration cases possible (for example to analyse different masses or stiffness modifications)
- Output of eigenvalue, angular frequency, natural frequency and period
- Determination of mode shapes and masses in FE mesh points
- Output of modal masses, effective modal masses, and modal mass factors

- Visualization and animation of mode shapes
- Different scaling options for mode shapes
- Documentation of numerical and graphical results in the printout report

Four powerful eigenvalue solvers are available in *RF-DYNAM Pro - Natural Vibrations*:

- Root of the characteristic polynomial
- Lanczos method
- Subspace iteration
- ICG iteration method (Incomplete Conjugate Gradient)

The selection of the eigenvalue solver primarily depends on the size of the model.

After the calculation, the eigenvalues, natural frequencies and periods are listed. These result tables are embedded in the main program RFEM. The mode shapes of the structure are tabulated and can be displayed graphically or as an animation. All result tables and graphics are part of the RFEM printout report. Moreover, exporting the tables to Excel is possible.

## Forced Vibrations

*RF-DYNAM Pro - Forced Vibrations* is an extension of the add-on module *RF-DYNAM Pro - Natural Vibrations*. Mechanical structures that are excited by transient or harmonic force-time or acceleration-time diagrams can be analysed using the modal analysis or the linear implicit Newmark solver. Furthermore, a multi-modal and multi-point response spectra analysis can be performed. The required spectra can be created according to the standards or individually. The add-on module contains an extensive library of accelerograms from earthquake zones. They can be used to generate response spectra.

The features of the time history analysis are listed below:

- Combination of user-defined time diagrams with load cases or load combinations (nodal, member and surface loads as well as free and generated loads can be combined with functions varying over time)
- Combination of several independent excitation functions possible
- Extensive library of earthquake recordings (accelerograms)
- Modal analysis based on the eigenvalue analysis or direct integration with the implicit Newmark solver in the linear time history analysis available
- Modal analysis:
  - Direct import of initial deformations from a load case possible
  - Stiffness modifications from *NVCs* are taken into account
- Implicit Newmark solver:
  - Independent of eigenvalue analysis
  - The member type *Dashpot* provides the possiblity to model local linear viscous damping elements with parallel connected linear spring elements (Kelvin-Voigt-Model).
- Structural damping using the Rayleigh damping coefficients or the Lehr's damping values
- Graphical result display in a time course monitor
- Export of results in user-defined time steps or as an envelope

The features of the response spectra analysis are listed below:

- Response spectra of numerous standards (EN 1998-1 [1], DIN 4149 [2], IBC 2012 [3] etc.)
- Response spectra can be user-defined or generated from accelerograms
- Direction-relative response spectra approach
- Different response spectra can be assigned to different supports (multi-point option)
- Relevant mode shapes for the response spectra can be selected manually or automatically (the 5% rule from EN 1998-1 [1] can be applied)
- Calculation is performed within RF-DYNAM Pro and is therefore linear
- Modifications of the stiffness matrix as defined in the *Natural Vibration Cases* are also used for the calculation of internal forces and deformations
- Combination of the modal responses (*SRSS* rule or *CQC* rule) and combination of the results from different excitation directions (*SRSS* or 100% / 30% rule)

The results from the time history analysis are displayed in a time course monitor. Here, you have the possibility to superimpose different nodes or positions within one member. All results are displayed as a function of time. The numeric values can be exported to MS Excel. In case of a time history analysis, you can export results of a single time step or filter the most unfavourable results of all time steps.

As the response spectra analysis is based on the eigenvalues and mode shapes and the internal forces, deformation and further results are calculated internally within *RF-DYNAM Pro*, the calculation is purely linear, but the stiffness modifications as defined in the *Natural Vibration Cases* are used for the whole analysis. Only result combinations are exported. A combination of the modal responses and a combination of the results due to the components of the earthquake action are done internally.

The input data of RF-DYNAM Pro and the exported load cases and result combinations are part of the printout report.

## Nonlinear Time History

*RF-DYNAM Pro - Nonlinear Time History* is an extension of the *RF-DYNAM Pro - Forced Vibrations* module. Its GUI is fully integrated in the structure of that add-on module. It is extended by two nonlinear time history solvers – the nonlinear implicit Newmark solver and the explicit solver.

This add-on module allows for nonlinear dynamic analyses to external excitations, either time diagrams or accelerograms. Various excitation functions can be defined tabularly as harmonic loads or as a function of time. An extensive library of accelerograms is available. This module extension removes the limitation to linear systems in dynamics.

The main features are listed below:

- Combination of user-defined time diagrams with load cases or load combinations (nodal, member and surface loads as well as free and generated loads can be combined with functions varying over time)
- Definition of user-defined time diagrams in tabular form, as periodic or as a function term
- Combination of several independent excitation functions possible
- Extensive library of earthquake recordings (accelerograms)
- Nonlinear time history analysis either with the nonlinear implicit Newmark solver or an explicit solver

- Supported nonlinearities:
  - Types of members (tension, compression members, cables)
  - Member nonlinearities (failure, tearing or yielding under tension/compression, plastic hinge)
  - Support nonlinearities (failure, friction, diagram, partial activity)
  - Hinge nonlinearities (fixed options, friction, diagram, partial activity)
  - Nonlinear material models
- The member type *Dashpot* provides the possibility to model local linear viscous damping elements with parallel connected spring elements (Kelvin-Voigt-Model).
- Calculation independent of eigenvalue analysis
- Definition of structural damping via the Rayleigh damping coefficients
- Graphical result display in a time course monitor
- Export of results in user-defined time steps or as an envelope

The results from the nonlinear time history analysis are displayed in a time course monitor. All results are displayed as a function of time. Results of single time steps or dynamic envelopes of all time steps are available. All results can be exported to MS Excel.

The input data of RF-DYNAM Pro and the exported load cases and result combinations are part of the printout report.

## Equivalent Loads

*RF-DYNAM Pro - Equivalent Loads* is an extension of the *RF-DYNAM Pro - Natural Vibrations* add-on module. With this module, seismic analyses can be performed with the multi-modal response spectrum analysis. The required spectra can be created according to the standards or user-defined, from which the equivalent static loads are generated.

The main features are listed below:

- Response spectra of numerous standards (EN 1998-1 [1], DIN 4149 [2], IBC 2012 [3] etc.)
- Input of user-defined response spectra
- Direction-relative response spectra approach
- Relevant mode shapes for the response spectra can be selected manually or automatically (the 5% rule from EN 1998-1 [1] can be applied)
- Generated equivalent static loads are exported to load cases, separately for each mode and direction
- The calculation of these load cases is performed in RFEM. Thus, a non-linear calculation can be performed.
- Stiffness modifications that are applied in the *Natural Vibration Cases* are not automatically applied in the *Load Cases*
- Combination of the modal responses (*SRSS* or *CQC* rule) and combination of the results from different excitation directions (*SRSS* or 100% / 30% rule)
- Results envelopes with maximum and minimum results or signed result in accordance to a dominant mode shape

Equivalent static loads are generated separately for each relevant eigenvalue and excitation direction. They are exported to static load cases, and a static analysis is performed in RFEM. Those load cases are quadratically combined in result combinations, followed by the combination of different excitation directions.

The input data of RF-DYNAM Pro and the exported load cases and result combinations are part of the printout report.

## 1.2 Using the Manual

Topics like installation, graphical user interface, evaluation of results, and printout are described in detail in the manual of the main program RFEM. This manual, however, focuses on typical features of the RF-DYNAM Pro add-on module.

The sequence and structure of the manual follows the input and results windows of the module. In the text, the **buttons** are given in square brackets, for example [Edit]. At the same time, they are pictured on the left. The **expressions** that appear in dialogue boxes, windows, and menus are set in *italics*.

At the end of the manual, you find the index. If you do not find what you are looking for, you can use the search option for our Knowledge Base to find a solution among the articles on dynamics. The FAQ pages are also highly recommended. Furthermore, you can watch the recorded Webinars and study the Verification Examples.

## 1.3 Opening the RF-DYNAM Pro Module

In RFEM, you have the following possibilities to start the add-on module RF-DYNAM Pro.

### Menu

To open the add-on module, select

**Add-on Modules** → **Dynamic** → **RF-DYNAM Pro**

on the RFEM menu.

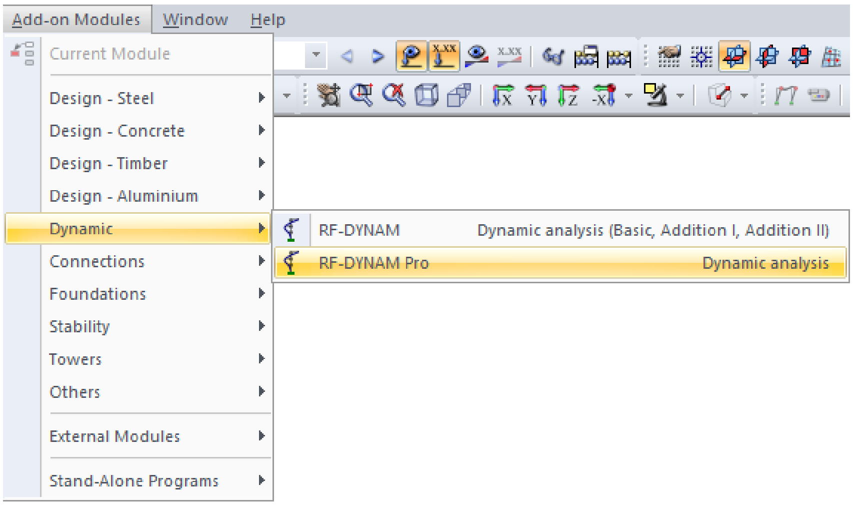



Figure 1.1: Menu *Add-on Modules* → *Dynamic* → *RF-DYNAM Pro*

If RF-DYNAM Pro was opened before and is your current module, you can also select

**Add-on Modules** → **Current Module**.

## Navigator

Alternatively, you can open the add-on module in the *Data* navigator via

**Add-on Modules** → **RF-DYNAM Pro**.

When you right-click on the add-on module to open its shortcut menu, you can add RF-DYNAM Pro to your *Favourites* (see Figure 1.2).

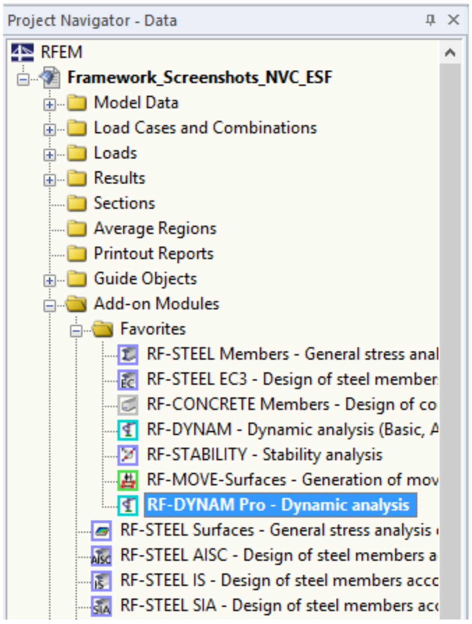



Figure 1.2: *Data* navigator *Add-on Modules* → *RF-DYNAM Pro*

## Panel

If RF-DYNAM Pro results are already available, you can also open the add-on module via the panel.


To display the panel, select the RF-DYNAM Pro case in the drop-down menu of the main program RFEM and make the results visible with the [Show Results] button. If the panel is not shown, you can activate it by clicking the [Panel] button.

RF-DYNAM Pro

Use the [RF-DYNAM Pro] button in the panel to re-open the add-on module.

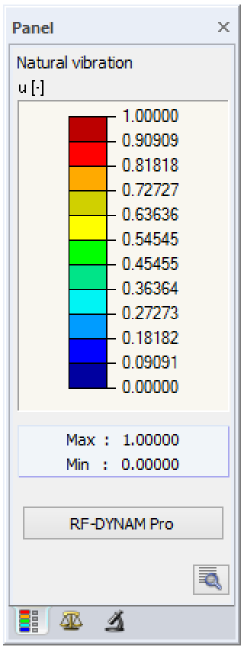



Figure 1.3: Panel with [RF-DYNAM Pro] button

# 2 Input Data

When you start the add-on module, a new window opens. The window is organized in several tabs and sub-tabs which you should go through from left to right when you enter data for the first time. Not all tabs are shown right from the beginning, some tabs belong to special settings that appear as soon as you have selected the corresponding check boxes.

The first window that appears when you open RF-DYNAM Pro is shown in Figure 2.1.

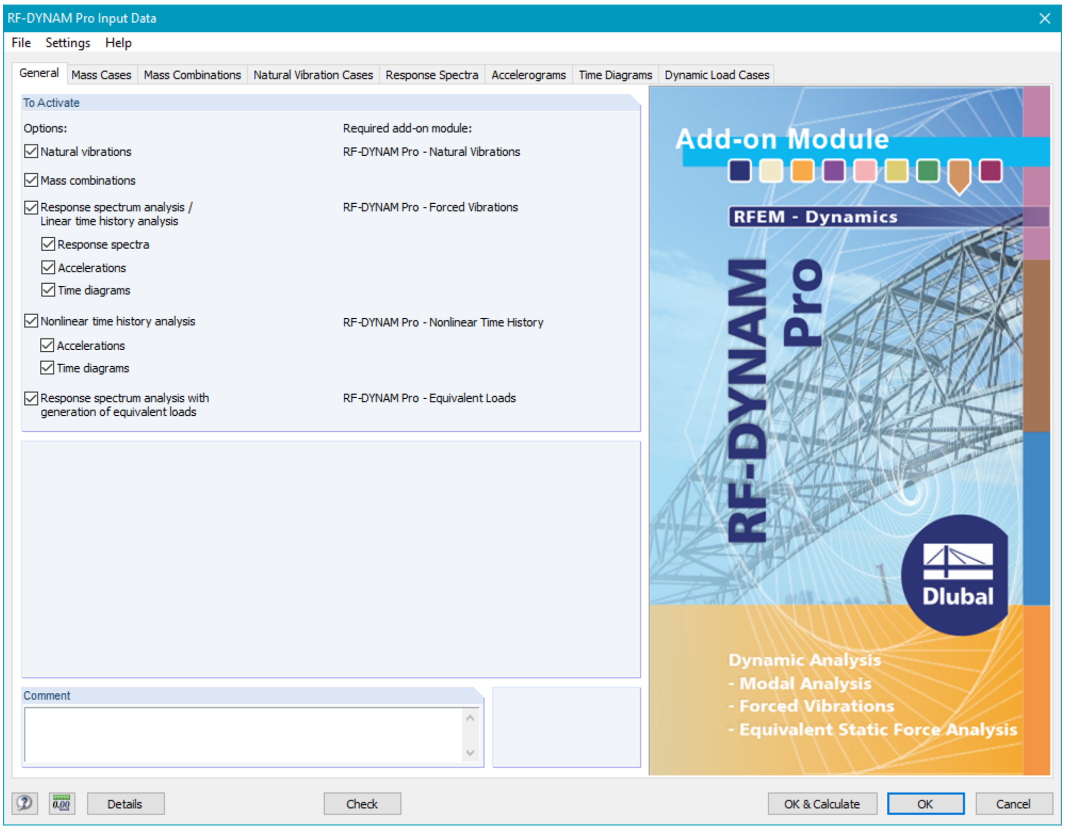



Figure 2.1: Module Window *General Data*

In the *General* tab, you decide which of the add-on modules of RF-DYNAM Pro you want to activate. You can analyse natural frequencies and mode shapes of the structural system with *RF-DYNAM Pro - Natural Vibrations*. With *RF-DYNAM Pro - Forced Vibrations*, you can perform a multi-modal response spectrum analysis or a linear time history analysis. A time history analysis under consideration of the defined nonlinearities in your model is possible with *RF-DYNAM Pro - Nonlinear Time History*. The module *RF-DYNAM Pro - Equivalent Loads* offers you a multi-modal response spectrum analysis with the export of equivalent static forces.

Add-on modules that were not purchased can be opened, but they run only as a demo version. You can also activate a 90-day trial version of those add-on modules.

## Natural Vibrations

This option is selected by default because a natural vibration analysis of the structure is required in most cases. The tabs *Mass Cases* and *Natural Vibration Cases* belong to this option by default. When you select the *Mass combination* check box, the corresponding tab appears (see Figure 2.1).

### Forced Vibrations

With this module, you can either perform a multi-modal response spectrum analysis or a linear time history analysis. When you select both (*Response spectra* and *Accelerations*), you can generate response spectra from an acceleration-time diagram. The *Response Spectra* tab appears when *Response spectra* option is selected, the *Accelerograms* tab appears when *Accelerations* option is selected. The *Time Diagrams* tab appears when the *Time diagrams* option is selected. The *Dynamic Load Cases* tab is available for all three options of the *RF-DYNAM Pro - Forced Vibrations* module.

### Nonlinear Time History

With this add-on module, a nonlinear linear time history analysis can be performed. The user interfaces of the linear and the nonlinear time history analyses are identical. The *Accelerograms* tab appears when *Accelerations* option is selected, the *Time Diagrams* tab appears when *Time diagrams* option is selected. The *Dynamic Load Cases* tab is available in both cases.

### Equivalent Loads

This option allows the generation of equivalent static forces in accordance with various design standards (**EN 1998-1** [1], **IBC2012** [3] and many others). The *Response Spectra* and the *Dynamic Load Cases* tabs belong to the module *RF-DYNAM Pro - Equivalent Loads*.

## 2.1 Structure of Main Tab Window

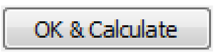
The module window of *RF-DYNAM Pro* always contains the following buttons:

The [Help] button gives direct access to the manual and further on-line help. The help system can also be reached by **Help** → **...** or by pressing the function key [F1].

To change the units or number of decimal places of any input data or results, you have direct access to the *Unit and Decimal Places* dialogue box as known from the main program RFEM. It can also be reached by the **Settings** → **Units and Decimal Places** menu. This is further discussed in Section 2.13 and Section 4.7.


In the *Details* dialogue box you define global settings that apply to whole dynamic calculation performed in RF-DYNAM Pro. This dialogue box is also accessible by **Settings** → **Details**. The settings are explained in Section 2.12.


To check the input data, click [Check]. This does not perform the calculation and the module window stays open.




To perform the calculation and exit the *RF-DYNAM Pro* module, press the [OK & Calculate] button.


To save the input data and exit the *RF-DYNAM Pro* module, press [OK]. The calculation is not performed.


To exit the module without saving the new data, click [Cancel].

As shown in Figure 2.1, the main tabs that are available are *General*, *Mass Cases*, *Mass Combinations*, *Natural Vibration Cases*, *Response Spectra*, *Accelerograms*, *Time Diagrams*, and *Dynamic Load Cases*. Beside the *General* tab, all main windows are structured in a similar manner. This is explained using the *Mass Case* Window shown Figure 2.2.

On the left hand side of each main window, you have a list of existing cases together with their descriptions. It is marked by an orange box in Figure 2.2. Those can be *Mass Cases*, *Mass Combinations* or *Natural Vibration Cases*, for example.

At the bottom of this list, marked with a red box in Figure 2.2, you find several buttons to create new cases, copy existing cases or delete existing cases.

There are also buttons to select all cases, deselect all cases, and to invert the selection of cases.

In the right part of the main window, you find the number of the selected case together with the case description at the top (marked by a blue box in Figure 2.2). In the description box, you can enter a case description manually or choose one from the drop-down list. Below you find the main entering area for your input data which firstly opens in the *General* sub-tab. Some more sub-tabs may appear depending on the selected check boxes.

In the *General* sub-tab, you can enter a comment. This space is marked by a green box in Figure 2.2.

## 2.2 Mass Cases

In RF-DYNAM Pro you have the possibility to define several *Mass Cases*. The *Mass Case* window is shown in Figure 2.2.

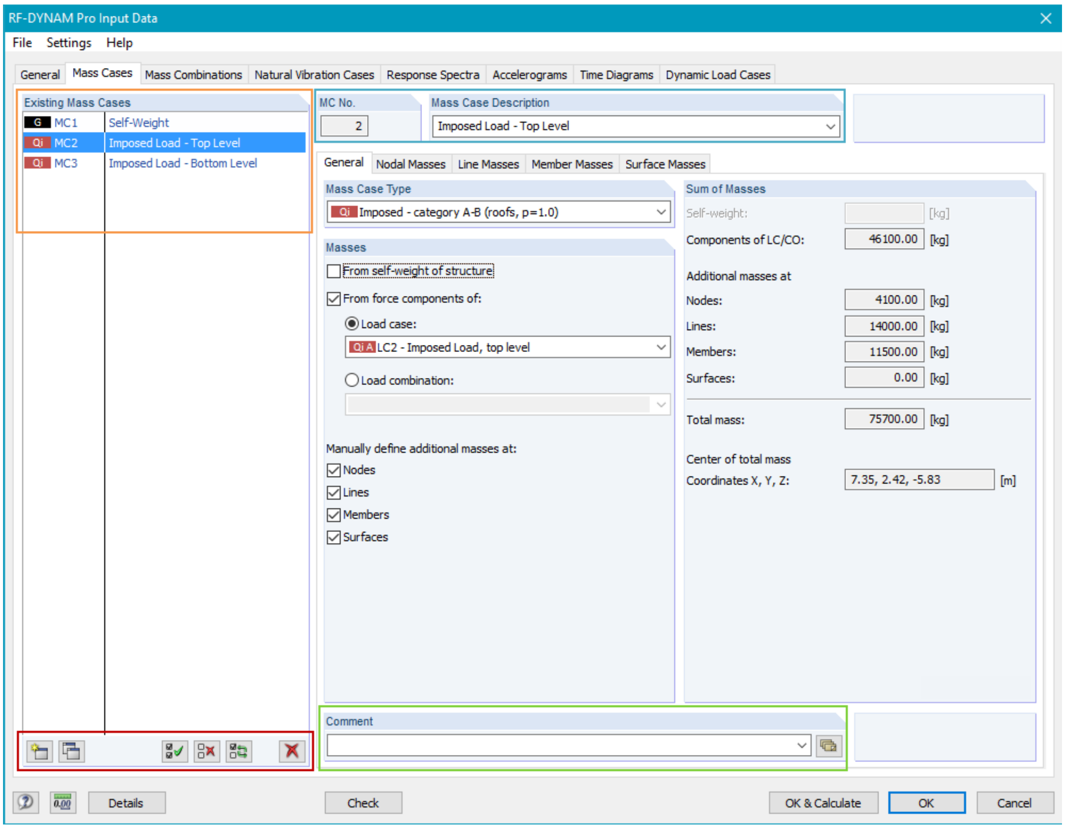
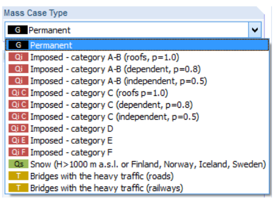



Figure 2.2: Module Window *Mass Cases* with *General* tab – *MC2* is selected and all four options to define additional masses are selected to show the sub-tabs

The mass case number is set automatically and cannot be edited. If a mass case is deleted later, the numbers do not change. You can enter a description manually, or you can choose one from the drop-down list.

### 2.2.1 Mass Case Type

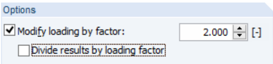



Choose one of the mass case types from the drop-down list. This is especially important when you use *Mass Combinations* (see Section 2.3). Combination factors are then pre-set in accordance with **EN 1990 CEN** [4] and **EN 1998-1 CEN** [1].

### 2.2.2 Masses

Select the *From self-weight of structure* check box when you want to include the self-weight independent of any load case defined in RFEM. This option is shown in Figure 2.2.

You can import masses from a load case or load combination defined in RFEM by selecting the *From force components* check box. Choose the relevant *Load case* or *Load combination* from the lists. Load types like forces and moments will be imported. Additionnally, the types *Pipe content - full* and *Pipe content - partial* can be converted into masses.




The factor to modify loads of load cases, which can be set in the calculation parameters of RFEM (**Edit** → **Load Cases and Combinations**), is not applied in RF-DYNAM Pro. To increase the masses by a factor you have to employ *Mass Combinations* described in Section 2.3.

When you import a load case that contains self-weight of the model, make sure that the *From self-weight of structure* check box in this tab is cleared. Otherwise you will double the self-weight of the model.

In Section 2.2.4, global settings in RF-DYNAM Pro are described that influence the mass import from *Load Cases*.

In addition, or as an alternative to the previously described options to import masses, it is possible to define nodal, line, member, or surface masses. Depending on the check boxes you select, additional sub-tabs will appear as shown in Figure 2.2. The settings within those sub-tabs are detailed in Section 2.2.5 and Section 2.2.6.

### 2.2.3 Sum of Masses

On the right hand side of the window, the sums of masses are shown to double-check the input of self-weight, imported masses from load cases or combinations, and additional masses. The total mass and the resulting centre of mass of the selected *Mass Case* are indicated as well.

Possibly neglected masses (see Section 2.2.4) are not reflected in the sum of mass. Here, full masses are presented.

### 2.2.4 Global Settings for Mass Import in *Details*


The *Details* dialogue box controls the global settings which influence the import of masses. In particular, it is the type of mass conversion and the option to neglect masses. Those settings are global, i.e. they act on all defined *Mass Cases* in RF-DYNAM Pro. For further details on *Details* dialogue box, see Section 2.12.

#### Mass Conversion Type

You can determine whether *Full loads* or only *Z-components* of the forces in both Z-directions or only in the direction of gravity are imported. The available options are shown in Figure 2.3.

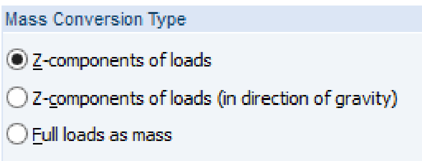
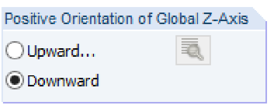



Figure 2.3: Section *Mass Conversion Type* in *Details* dialogue box

The third option imports full loads regardless their direction. This is the only way to import masses from horizontal earth pressure.

When different loads are acting on one FE-node, the load resultant is built first. It is then imported to RF-DYNAM Pro.




The direction of gravity is defined in the direction of the global Z-axis when the positive orientation is defined downwards. It is defined against the direction of the global Z-axis when the positive orientation is defined upwards. The settings shown on the left can be found in the **Edit** → **Model Data** → **General Data** menu of RFEM.

## Neglect Masses

It is possible to neglect masses from parts of the model. For this, select the *Neglect masses* option as seen in Figure 2.4.

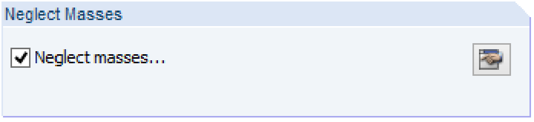



Figure 2.4: Section *Neglect Masses* in *Details* dialogue box

With the button, you open the *Neglect Masses* dialogue box (see Figure 2.5). There you can define which masses of nodes, members, lines and surfaces are to be neglected.

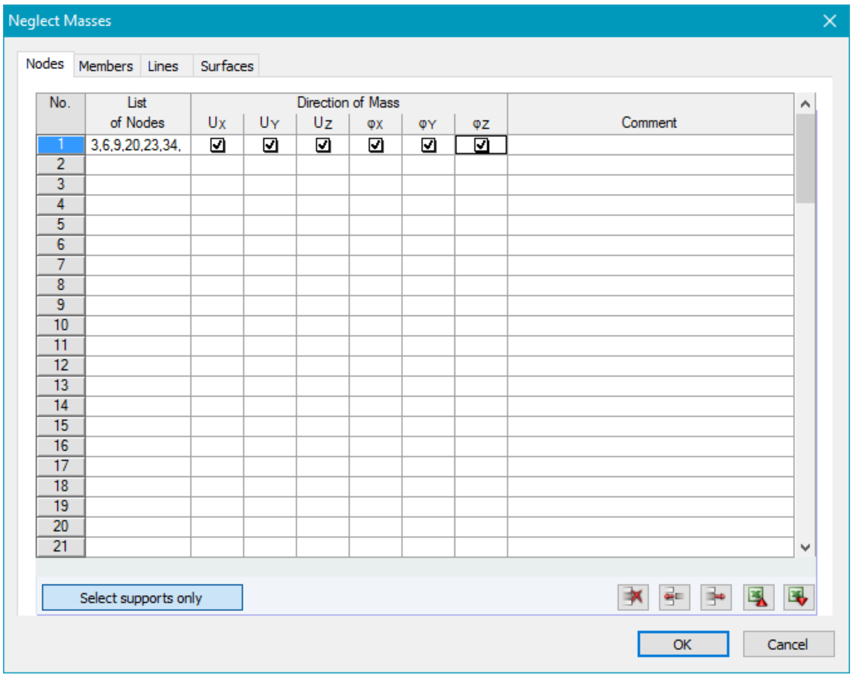



Figure 2.5: Dialogue box *Neglect Masses* for masses on nodes, members, lines, and surfaces

You can enter a list of nodes (members, lines and surfaces) manually or pick them graphically by using the button. The filter button [Select supports only] turns blue when activated; it is useful for the selection of supports. If this filter is active, you can select the whole model, but only the supported nodes (members, lines or surfaces) will be selected.

You can decide which directions of masses are to be neglected by selecting the relevant check boxes for displacements $u_X$, $u_Y$ and $u_Z$ and rotations $\varphi_X$, $\varphi_Y$ and $\varphi_Z$.

In the *Sum of Masses* section of the *Mass Case* tab (see Figure 2.2), the neglected masses are not visible. There always the full masses are shown. The neglected masses on the single FE-nodes can be checked in the result table *5.6 Masses on Mesh Points* (see Section 4.1).

You can find an example of the neglecting masses feature in the Knowledge Base - 001222.

## 2.2.5 Additional Nodal Masses

The *Nodal Masses* tab is shown in Figure 2.6. It is available when the *Manually define additional masses at nodes* option has been selected in the *General* tab.

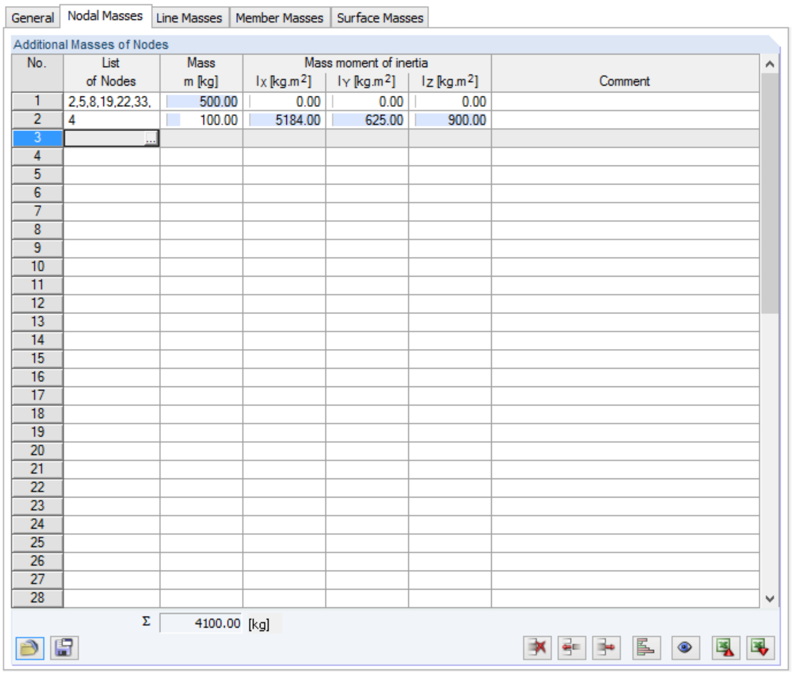



Figure 2.6: Module Window *Nodal Masses*

You can enter a list of nodes manually or use the [...] function to select the relevant nodes in the work window.

The masses *m* [kg] can be defined manually. They act in the directions defined in the *Natural Vibration Cases* (see Section 2.4.3).

In addition, the *Mass moment of inertia* $I_X$, $I_Y$, and $I_Z$ can be defined to model more complex mass points (i.e. rotation of a machine can approximately be considered).

The buttons beneath the table provide common table functions as described in the RFEM manual, Section 11.5.

The table entry can be stored in a library and can be opened whenever needed. The [Save] button opens a dialogue box where you can enter a file name.

## 2.2.6 Additional Line, Member and Surface Masses

In addition to nodal masses, you can define line, member or surface masses manually. The corresponding sub-tabs appear when the check boxes for additional masses at *Lines*, *Members* or *Surfaces* have been selected in the *Mass Cases* tab (see Figure 2.2). The table to enter additional line masses is shown in Figure 2.7.

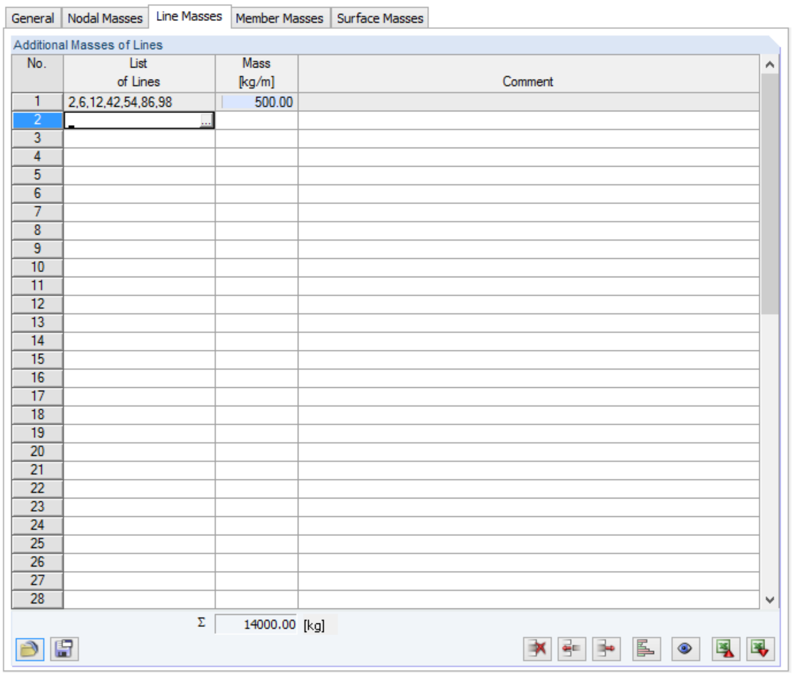



Figure 2.7: Module Window *Line Masses*

As the tables to enter member and surface masses are very similar, they are not explicitly shown here.

A list of lines (members or surfaces, respectively) can be entered manually or by using the function which makes it possible to select the objects graphically in the work window.

Line and member masses are provided in [kg] per unit length. Surface masses are given in [kg] per unit area. They can be defined manually and act in the directions defined in the *Natural Vibration Cases* (see Section 2.4.3).

## 2.3 Mass Combinations

Mass cases can be combined to mass combinations. This is done in analogy to load cases and load combinations in RFEM as described in the RFEM manual in Section 5.5.1. The *Mass Combinations* tab only exists when the corresponding check box has been selected in the *General* tab (see Figure 2.1). When you open the *Mass Combinations* tab for the first time, a mass combination is pre-set and the existing mass cases are listed with the combination factors set by default.

The module window with mass combination *MCO1* containing self-weight and imposed load mass cases is illustrated in Figure 2.8.

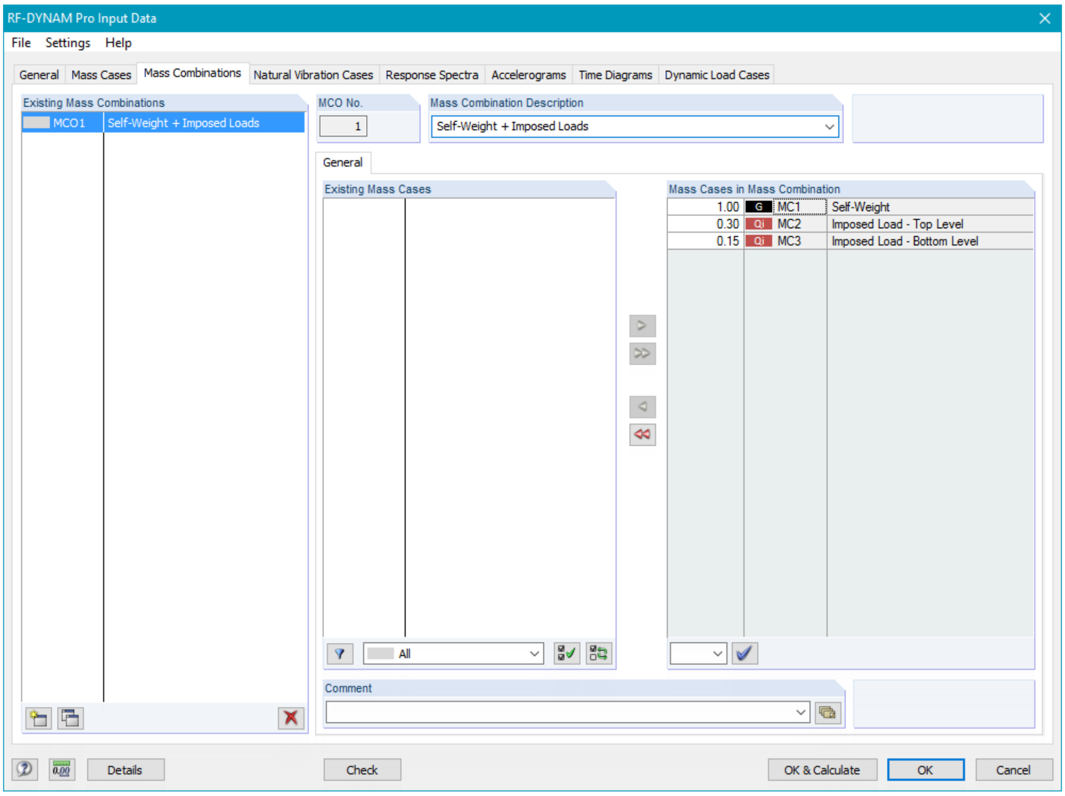
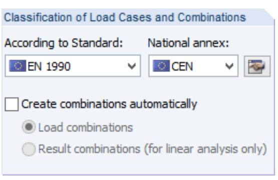



Figure 2.8: Module Window *Mass Combinations* with all existing mass cases selected for mass combination

You can add selected or all mass cases to a mass combination by using the buttons shown on the left. By doing so, the mass cases move from the left to the right list. Combination factors are pre-set automatically by RF-DYNAM Pro, but can be changed manually.

Similarly, you can remove single or all mass cases from a mass combination by using the buttons shown on the left.




The combination factors are pre-set in accordance to EN 1990 [4] and EN 1998-1 [1]. Those factors can be adjusted manually by entering a value or choosing a value from the drop-down list. As regulated in [1] Section 3.2.4, additional masses beside the self-weight have to be considered to calculate inertia effects.

$$\sum G_{k,j} \text{ “+” } \sum \psi_{E,i} \cdot Q_{k,i} \qquad (2.1)$$

where $G_{k,j}$ are the permanent loads and $Q_{k,i}$ any imposed load. $\psi_{E,i}$ are the combination factors for the imposed loads defined as

$$\psi_{E,i} = \varphi \cdot \psi_{2,i} \qquad (2.2)$$

where $\psi_{2,i}$ are regulated in EN 1990 Table A.1.1 and $\varphi$ are provided in EN 1998-1 Section 4.2.4.

The combination factors may be regulated differently in each National Annex of EN 1998-1 or other international building standards.

## 2.4 Natural Vibration Cases

The *Natural Vibration Cases* tab is the centrepiece of the **RF-DYNAM Pro - Natural Vibrations** module. It is essential for the response spectrum analysis and the time history analysis based on modal decomposition (modal analysis).

In this tab, you set how many eigenvalues you want to calculate, define which masses are used and in which direction they act. You also set the eigenvalue solver, the type of mass matrix and define how the mode shapes are scaled. This is also the place where you define stiffness modifications or import axial forces as an initial condition.

The *General* tab of the *Natural Vibration Case* window is shown in Figure 2.9.

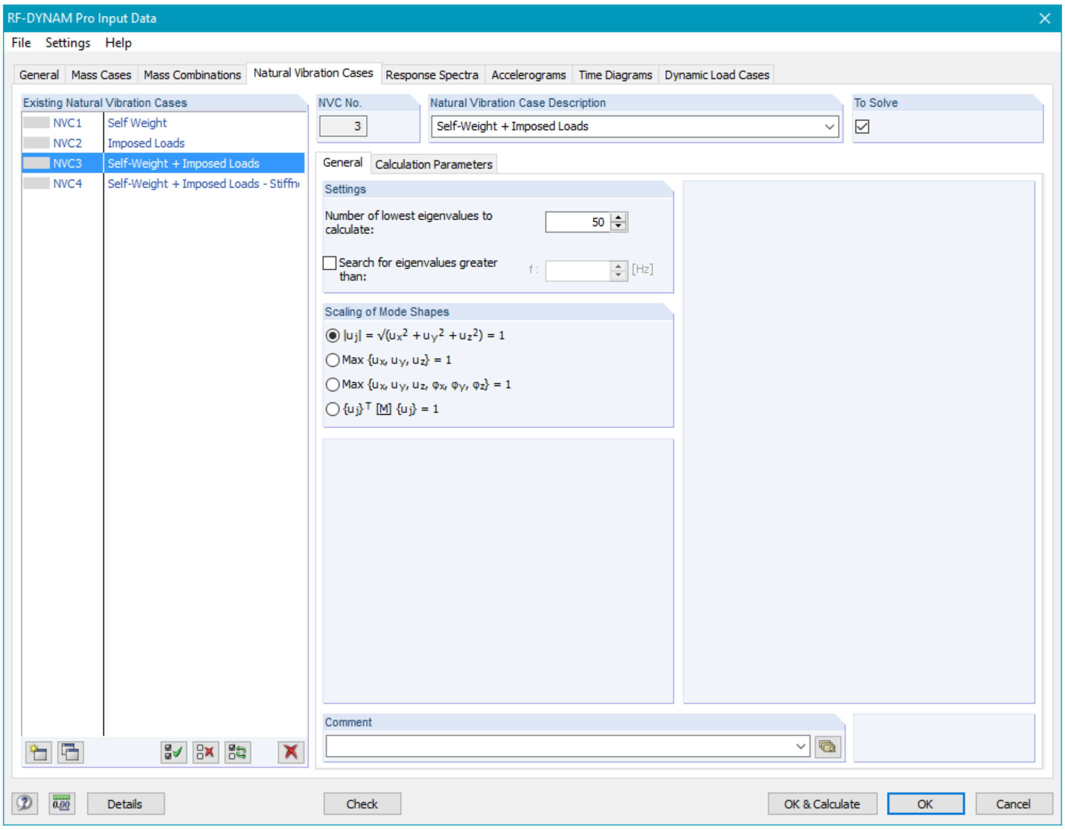
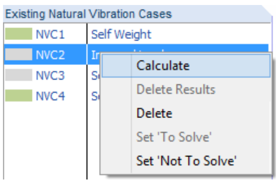


Figure 2.9: Module Window *Natural Vibration Cases* with *General* tab

For each natural vibration case *NVC*, you can set various calculation parameters as shown in Figure 2.10.

The number of the natural vibration case is set automatically and cannot be edited. When a case is deleted later, the numbers do not change. You can decide whether the specific *NVC* is to be calculated or not by selecting or clearing its *To Solve* check box.



You can calculate each *NVC* separately by using the shortcut menu and apply *Calculate*. The colour of a *NVC* is grey when no results of this natural vibration case are available. It turns green as soon as the calculation is finished and results are available. For more information, see Section 3.3.

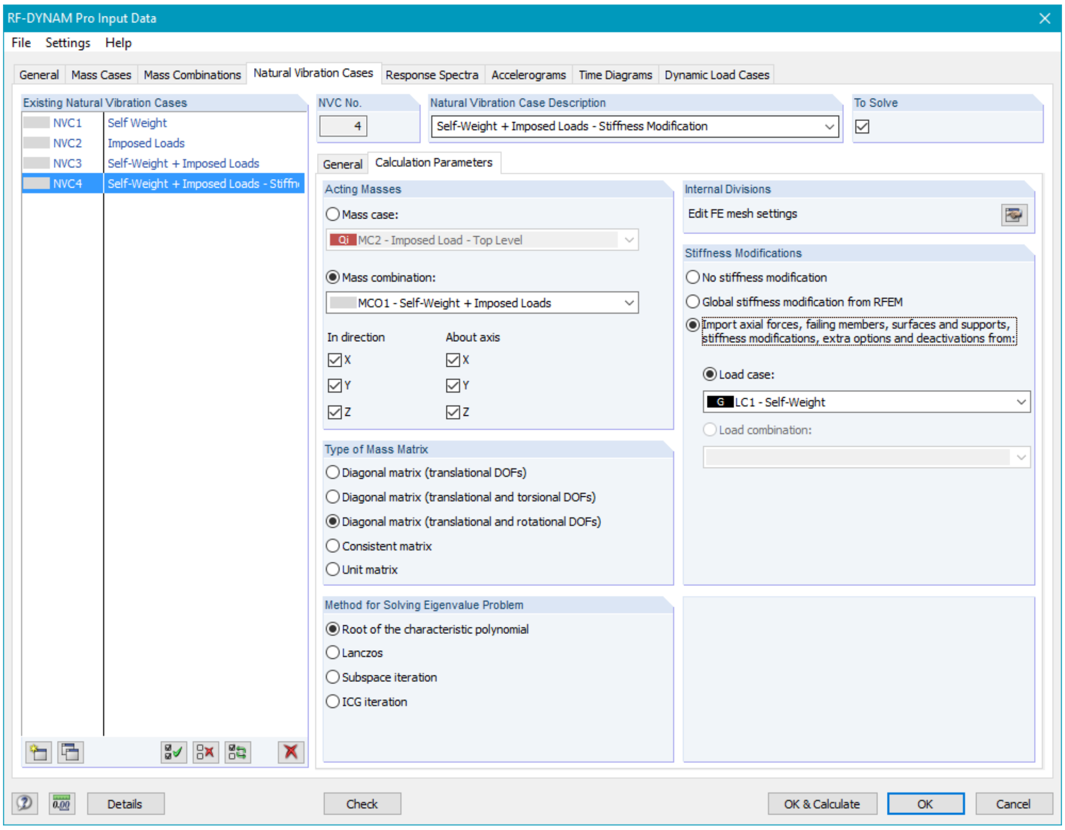
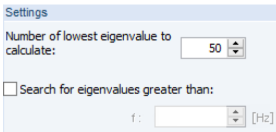


Figure 2.10: Module Window *Natural Vibration Cases* with *Calculation Parameters* tab

## 2.4.1 Number of Eigenvalues

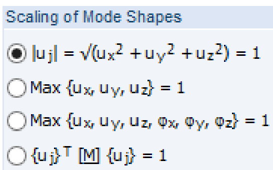



In the *General* tab (see Figure 2.9), you define the number of lowest eigenvalues to be calculated. The maximum number of eigenvalues is limited to 9,999 in RF-DYNAM Pro, but is also limited by the structural system. The number of available eigenvalues is equal to the degrees of freedom (number of free mass points multiplied by the number of directions in which the masses are acting).

It is possible to calculate eigenvalues only above a certain value of natural frequency *f* to reduce the number of produced results.

Be careful with this option and study the lowest eigenvalues of the system first. To evaluate the importance of each eigenvalue, the *Effective Modal Mass Factors* are useful (see Section 4.1.4).

## 2.4.2 Scaling of Mode Shapes



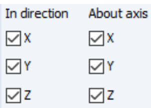

The mode shapes can be scaled to any arbitrary value. The first three options are all a good choice for a satisfying illustration of the mode shapes. The option $|u_j| = \sqrt{u_x^2 + u_y^2 + u_z^2} = 1$ scales the magnitude of the mode shape vector $u_i$ (only translational parts) to 1. The option $Max\{u_X,u_Y,u_Z\} = 1$ chooses the maximum translational part of the mode shape vector and sets it to 1. The option $Max\{u_X,u_Y,u_Z,\varphi_X,\varphi_Y,\varphi_Z\} = 1$ considers the complete mode shape vector including the rotational parts, chooses the maximum and sets this to 1. For all those three options, the scaling is done separately for each eigenvalue i.

The option $\{u_j\}^T[M]\{u_j\} = 1$ is always used internally for time history or response spectrum analyses, regardless the choices set here. The modal masses $m_j$ are 1 kg for each eigenvalue when using this scaling option (see Section 4.1).

No matter which scaling option is chosen: the translational mode shapes $u_X$, $u_Y$ and $u_Z$ are dimensionless and the rotational mode shapes $\varphi_X$, $\varphi_Y$ and $\varphi_Z$ are provided in $[1/m]$. The resulting mode shapes are discussed in Section 4.1.2.

## 2.4.3 Acting Masses

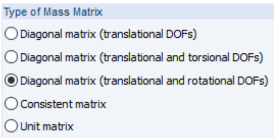
For each natural vibration case (*NVC*), a *Mass case* or *Mass combination* can be imported. Select it from the respective list in the *Calculation Parameters* sub-tab (see Figure 2.10).




You have to define the direction in which the masses are acting. The masses act in the global translational X, Y, or Z-direction when you activate the corresponding check boxes, or rotationally about the global X, Y, and Z-axes.

Those settings change your mass matrix: You will get different resulting mode shapes and frequencies. To perform a planar calculation of your structure, consider only masses acting in one global direction at a time. The planar simplification requires a regular structure. For a three-dimensional analysis, consider masses acting in all global directions.

## 2.4.4 Type of Mass Matrix




Five different types of mass matrices are available in the *Calculation Parameters* tab as shown in Figure 2.10.

### Diagonal matrix

When the type of the mass matrix **M** is chosen to be diagonal, the masses are lumped to the FE-nodes. The entries in the matrix are the lumped masses in the translational directions X, Y, and Z. Masses rotating about the X, Y or Z-axis are considered depending on the type of diagonal matrix. There are three different types of diagonal mass matrices available:

**- Diagonal matrix (translational DOFs)**

Only masses in the translational directions X, Y, and Z are considered. The check boxes *about axis* are not available. The diagonal mass matrix **M** is structured as follows:

$$\mathbf{M} = diag\left(M_{1,X}, M_{1,Y}, M_{1,Z}, M_{2,X}, \cdots, M_{n,j}, \cdots\right) \tag{2.3}$$

where n = 1... FE-nodes and j = X, Y and Z directions

**- Diagonal matrix (translational and torsional DOFs)**

The masses in the translational directions X, Y, and Z as well as the masses acting about the longitudinal axis of a member or surface are considered, if the check boxes *about axis* have been selected. The direction that is taken into account depends on the local longitudinal directions of the members and surfaces. The diagonal mass matrix **M** on a specific FE-point with mass $m$ is structured as follows:

$$\mathbf{M} = m \cdot diag\left(1,1,1,\ Y^2 + Z^2, X^2 + Z^2, X^2 + Y^2\right) \tag{2.4}$$

where X, Y, or Z are the distances to the centre of total mass provided in the *Mass Case* tab described in Section 2.2. The manually defined mass moments of inertia $I_X$, $I_Y$ and $I_Z$ as provided in the *Nodal Mass* table (see Section 2.2.5) are also considered when acting in longitudinal direction.

**- Diagonal matrix (translational and rotational DOFs)** This type of mass matrix considers all degrees of freedom. The masses in the translational directions X, Y, and Z and rotational directions X, Y, and Z are taken into account. The diagonal mass matrix **M** on a specific FE-point with mass $m$ is structured as follows:

$$\mathbf{M} = m \cdot diag\left(1,1,1,\ (Y^2 + Z^2) + I_X,\ (X^2 + Z^2) + I_Y,\ (X^2 + Y^2) + I_Z\right) \tag{2.5}$$

where X, Y, or Z are the distances to the centre of total mass (provided in the *Mass Case* tab described in Section 2.2), and $I_X$, $I_Y$, or $I_Z$ are the mass moments of inertia (defined in the *Nodal Mass* table described in Section 2.2.5).

### Consistent Mass Matrix

The consistent mass matrix is a full mass matrix of the finite elements. Thus, the masses are not simply lumped to the FE-nodes. Shape functions are used instead for a more realistic distribution of the masses within the FE-elements. With this mass matrix, non-diagonal entries in the matrix are considered, which means that mass rotation in general is taken into account. The structure of the consistent mass matrix is as follows (neglecting the shape functions here for simplicity):

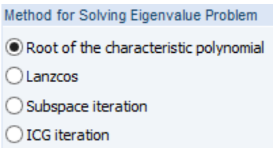
$$\mathbf{M} = m \cdot \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & Y^2 + Z^2 & -XY & -XZ \\ 0 & 0 & 0 & -XY & X^2 + Z^2 & -YZ \\ 0 & 0 & 0 & -XZ & -YZ & X^2 + Y^2 \end{bmatrix} \tag{2.6}$$

where the mass $m$ and the distances X, Y, or Z refer to the centre of total mass given in the *Mass Case* tab described in Section 2.2.

### Unit Matrix

The unit matrix overwrites all the previously defined masses. A unit matrix is a consistent matrix with diagonal members of 1 kg. By using the *Unit matrix*, the mass at each FE-point is set to 1 kg. Translations and rotations of the masses are considered. This mathematical approach should only be used for numerical analyses. More details about matrix types and especially the use of the unit matrix are provided by Barth and Rustler [5].

## 2.4.5 Eigenvalue Solver




There are different methods available to solve the eigenvalue problem. The choice is dependent on the size of the structural system considered and is more a question of performance than of accuracy. The methods are all suitable to determine accurate eigenvalues.

### Root of the characteristic polynomial

This is the analytic solution of an eigenvalue problem, see for example [6] or [7]. It is solved using a direct method. The main advantage of this method is the precision of higher eigenvalues and that all eigenvalues of a model can be determined. For large structures this method can be quite slow.

### Lanczos

The method by Lanczos is an iterative method to determine the $p$ lowest eigenvalues and corresponding mode shapes of large models. In most cases, this algorithm allows to reach a quick convergence. It is possible to calculate $n$-$1$ eigenvalues ($n$ = degrees of freedom of the system). For further details see Bathe [6].

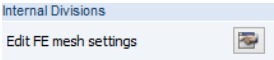
### Subspace Iteration

The subspace iteration is appropriate for large FE-models where you want to calculate only a few eigenvalues. All required eigenvalues are determined in one step. The computer memory limits the number of eigenvalues that can be calculated by this method. For further details see Bathe [6].

### ICG Iteration

This method is also suitable for large models where only a few eigenvalues are required. But here the eigenvalues are calculated successively. Therefore, the number of required eigenvalues is proportional to the computing time. Theoretically, all eigenvalues of a model can be calculated.

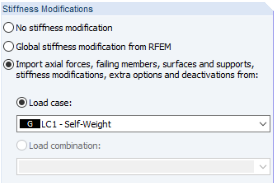
### 2.4.6 FE Mesh Settings




The button shown on the left links directly to the FE mesh settings of the main RFEM program. The *FE Mesh Settings* dialogue box is also available under **Calculation** → **FE Mesh Settings** in RFEM. Further details about the FE mesh and the parameters that can be adjusted are explained in the RFEM manual, Section 7.2.

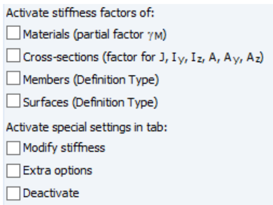
Members that are not integrated into surfaces are not divided into FE-nodes by default. When a diagonal mass matrix is used, the mass of the member is lumped to the two end nodes. When those end nodes are supported, you might not be able to calculate eigenvalues. In this case, you need to activate the member division in the FE mesh settings.

### 2.4.7 Stiffness Modifications as Initial Conditions



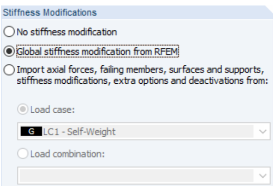

Stiffness modifications can be imported from a specific *Load case* or *Load combination*. In detail this means the following:

- Geometric stiffness matrix due to the influence of axial forces is added to the stiffness matrix of the structure. This can be employed to approximately consider the P-$\Delta$effect for the calculation of frequencies, or to evaluate the influence of pre-stressing cables. Theory and example when and how to consider the P-$\Delta$effect can be found in FAQ 002316.
- Failed members, surfaces and supports due to the defined non-linearities in RFEM are imported as initial conditions. Therefore, failed parts of the model are not available for the calculation of eigenvalues. An example how this feature can be used to study the influence of tension members in RF-DYNAM Pro is given in FAQ 002237.
- Stiffness factors for materials, cross-sections, members and surfaces are applied also in the eigenvalue analysis when those are activated in the *Calculation Parameters* of the LC or CO.
- The factors set in the *Modify Stiffness* tab of the LC or CO will be applied in RF-DYNAM Pro.
- The *Extra Options* settings of the LC or CO can be used to import the stiffness from the add-on module RF-CONCRETE.
- Import of deactivated members, surfaces and solids. This modifies the structural system that is analysed in *RF-DYNAM Pro*.




Calculation parameters of LC or CO in RFEM

The calculation parameters of a load case or load combination that are relevant for RF-DYNAM Pro are shown on the left hand side. The check boxes *Materials*, *Cross-sections*, *Members*, and *Surfaces* activate the modifications made in the material, cross-section, member, and surface dialogue boxes in the main program RFEM (see RFEM manual, Sections 4.3, 4.13, 4.17, and 4.4). Further details on the calculation parameters of load cases are given in the RFEM manual, Section 7.3.




*Global stiffness modifications* can be imported from RFEM independently of any *LC* or *CO*. Those global modifications import the material partial factor $\gamma_M$, cross-section, member and surface modifications that are changed in RFEM (see RFEM manual in Sections 4.3, 4.13, 4.17, and 4.4).

In RF-DYNAM Pro, those stiffness modifications are initial conditions. The calculation within RF-DYNAM Pro - Natural Vibrations is fully linear. You can activate the initial conditions as shown in Figure 2.10.

When you want to consider axial forces only, make sure that no stiffness modifications are activated in the calculation parameters of the imported *LC* or *CO*.

When you want to study deactivated members only, make sure you create a load case containing no loads but the stiffness modifications you are interested in.

A figure describing the complexity of this feature is provided in the Knowledge Base - 001023.

## 2.5 Response Spectra

A response spectrum is a plot of the maximum peak response to a specified input illustrated usually versus the natural period of single degree-of-freedom (SDOF) oscillators. It is produced by calculating the response to a specific input (i.e. average of several earthquake motions) for a family of SDOF oscillators each having a different natural period but the same damping value. There are computational advantages in using the response spectra method and it is a very common method described in building standards. But you should be aware that it is only an approximate method that calculates maximum internal forces of your system. For further details on the response spectrum analysis, see for example Wilson [8] and Tedesco [9].

RF-DYNAM Pro offers the multi-modal and multi-point (only with *RF-DYNAM Pro - Forced Vibrations*) response spectrum method. Ready to use response spectra curves are available in building standards, many of those are implemented in RF-DYNAM Pro. RF-DYNAM Pro can generate a response spectrum from any given accelerogram (only available with *RF-DYNAM Pro - Forced Vibrations*).

The *Response Spectra* tab is only available when the *Response spectra* or the *Equivalent loads* options have been selected in the *General* tab shown in Figure 2.1. This tab belongs either to the add-on module *RF-DYNAM Pro - Forced Vibrations* or to the module *RF-DYNAM Pro - Equivalent Loads*.

In RF-DYNAM Pro, you have three options to enter response spectra: according to a building standard, user-defined, or generated from an accelerogram. The *Response Spectra* tab is illustrated in Figure 2.11.

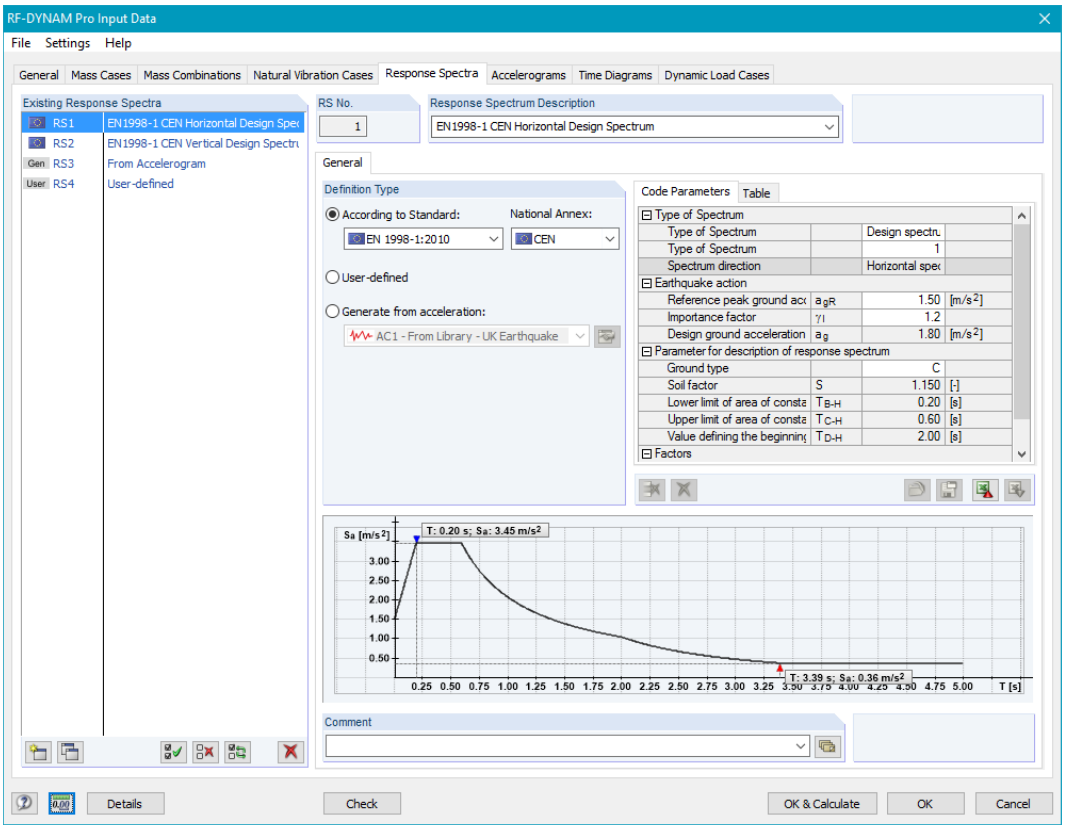



Figure 2.11: Module Window *Response Spectra* with *General* tab – option *According to Standard* selected, showing *Code Parameters*

### 2.5.1 Response Spectra According to Building Standards

The parameters according to a building standard can be entered to generate a response spectrum. The list of available building standards is provided in Figure 2.12.

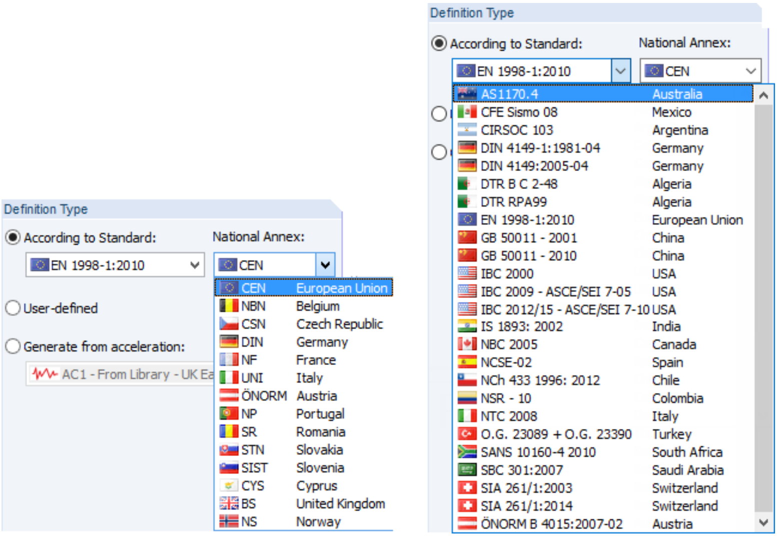


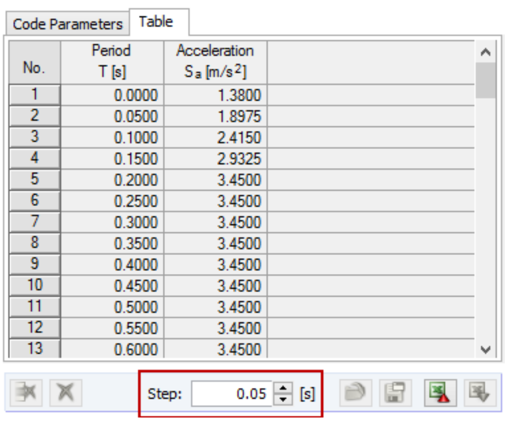

Figure 2.12: List of implemented international and European building standards that regulate the earthquake design of structures and provide formulas for response spectra

Select a *Standard* from the drop-down list; the *Code Parameters* and their default values (see Figure 2.11) change depending on the chosen standard. The parameters can be selected from drop-down lists or entered manually depending on the type of parameter and depending on the chosen building standard.

Many standards distinguish between a linear design spectrum employing the ductility factor $q$ and an elastic response spectrum employing a damping correction factor $\eta$. Note the drop-down menu to choose between these types of spectra. Please note also the drop-down menu to distinguish between horizontal and vertical response spectra definitions. The cells that are not editable are determined by another parameter already set. For example in EN 1998-1 CEN [1], the ground classes *A* to *E* determine the parameters $T_B$, $T_C$ and $T_D$; those parameters cannot be edited manually.

When you change the ground type to *Other*, you can adjust the parameters $T_B$, $T_C$ and $T_D$ manually for the response spectra in accordance to EN 1998-1 [1].

The resulting response spectrum is illustrated in the graphic seen in Figure 2.11. The tool tip of your mouse shows information about the displayed values. The values of the generated response spectra are listed in the *Table* tab as shown in Figure 2.13.




| No. | Period T [s] | Acceleration $S_a$ [m/s²] |
|---|---|---|
| 1 | 0.0000 | 1.3800 |
| 2 | 0.0500 | 1.8975 |
| 3 | 0.1000 | 2.4150 |
| 4 | 0.1500 | 2.9325 |
| 5 | 0.2000 | 3.4500 |
| 6 | 0.2500 | 3.4500 |
| 7 | 0.3000 | 3.4500 |
| 8 | 0.3500 | 3.4500 |
| 9 | 0.4000 | 3.4500 |
| 10 | 0.4500 | 3.4500 |
| 11 | 0.5000 | 3.4500 |
| 12 | 0.5500 | 3.4500 |
| 13 | 0.6000 | 3.4500 |

Figure 2.13: Tabulated values of generated response spectrum

The values can be exported to Excel. The time *Step* should be adjusted before exporting the data.

### 2.5.2 User-Defined Response Spectra

Any kind of response spectrum can be defined by entering the period $T$ and corresponding accelerations $S_a$ [m/s$^2$] in the *Table* shown in Figure 2.14.

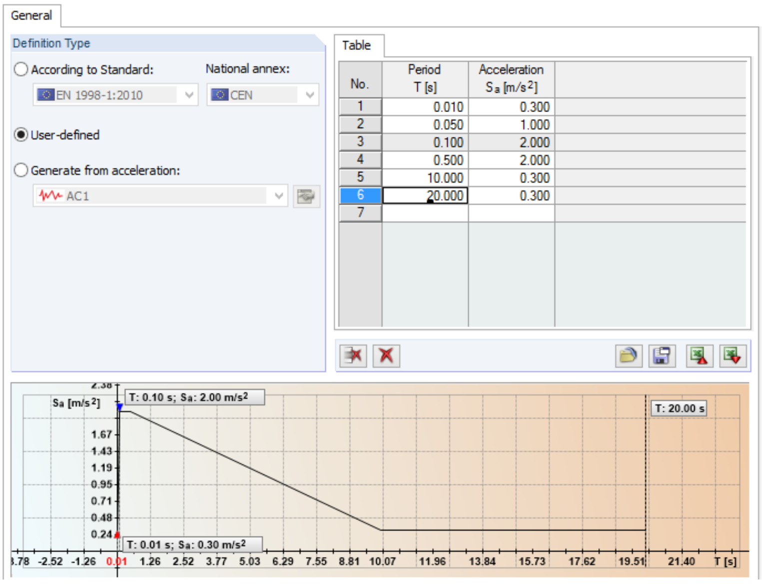



Figure 2.14: User-defined entry of response spectrum by defining *Period T [s]* and *Acceleration* $S_a$ *[m/s$^2$]*

User-defined response spectra can be stored in a library and can be opened whenever needed. The [Save] button opens a dialogue box to enter a file name.

You can export your user-defined response spectra to Excel or import response spectra from Excel by means of the respective buttons.

### 2.5.3 Response Spectra Generated from Accelerograms

The response spectra can be automatically generated from a given accelerogram.

This option is only available when *Accelerations* have been selected in the *General* tab shown in Figure 2.1. This option belongs to the add-on module *RF-DYNAM Pro - Forced Vibrations*.

The available options to generate response spectra from an acceleration are shown in Figure 2.15.

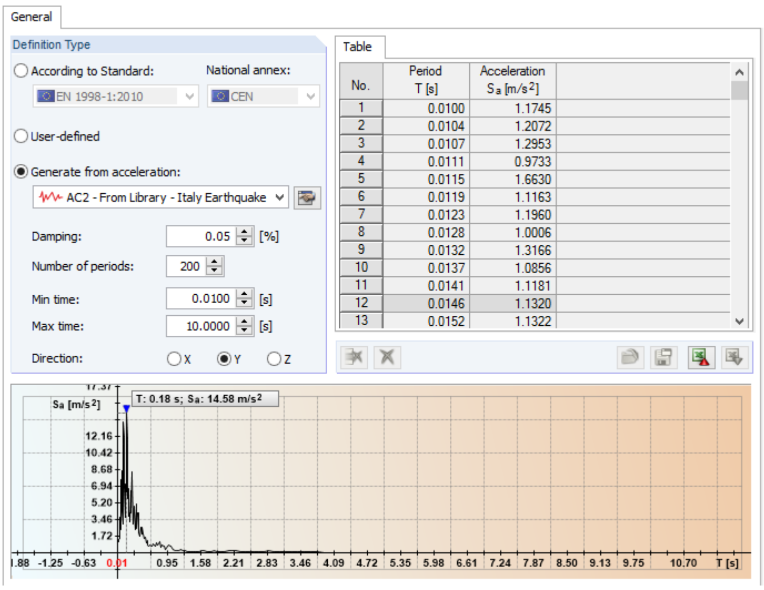
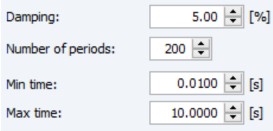



Figure 2.15: Generated response spectrum from acceleration with graphics and tabular values

The [Edit] button links to the *Accelerogram* tab described in Section 2.6 where you can define acceleration-time diagrams or load accelerograms from a library of earthquake recordings. When accelerograms are already defined, you can select the relevant entry from the drop-down menu.

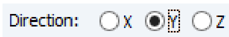



You can adjust the parameters as seen on the left. The generated response spectrum is illustrated in the graphic below, and the values of period $T$ and acceleration $S_a$ are listed in the table (see Figure 2.15). Both will be updated as soon as you change one of the parameters.

The viscous *Damping d* [%] is the damping of the SDOF oscillator family for which the maximum system responses are calculated. The larger the damping is set, the fewer accelerations will be generated.

The *Number of periods* determines the number of steps between the minimum and maximum time (period), and determines therewith the number of data points generated.

The *Min time* is the period of the first SDOF oscillator considered for the calculation. You see the results in the first row in the table shown in Figure 2.15. The *Max time* is the period of the last SDOF oscillator considered for the calculation. You see the results in the last row in the table shown in Figure 2.15.




The response spectra generated might be different in each direction as the accelerograms might be different in each direction. You can change the displayed response spectra in the graphic and table by using the radio buttons shown on the left.

You can export the generated response spectra to MS Excel.

## 2.6 Accelerograms

Accelerograms are acceleration-time diagrams usually recorded from previous earthquakes. In *RF-DYNAM Pro* accelerograms can be used to generate response spectra or to perform a time history analysis. In both cases the system can be excited on all or some supports.

The *Accelerogram* tab is shown in Figure 2.16.

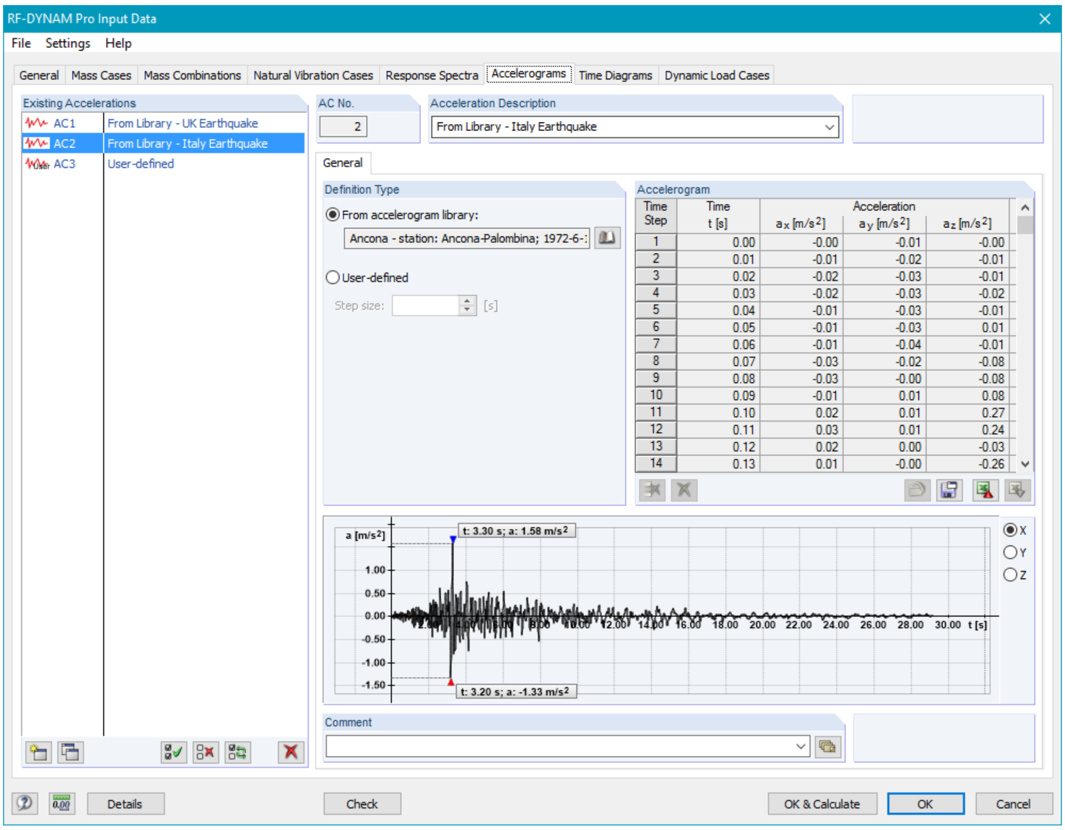



Figure 2.16: Module window *Accelerograms* with *General* tab

The *Accelerogram* tab is only available when *Accelerations* have been selected in the *General* tab shown in Figure 2.1. This tab belongs to the add-on modules *RF-DYNAM Pro - Forced Vibrations* and *RF-DYNAM Pro - Nonlinear Time History*.

In RF-DYNAM Pro, you can load accelerograms from a library or enter them manually.

## 2.6.1 Library of Earthquake Recordings

RF-DYNAM Pro provides a library with a large number of existing and measured accelerograms. By now, more than 1018 accelerograms have been collected and stored in this library. In addition, user-defined accelerograms can be saved in this library.

The library can be accessed with the button shown on the left. The library is shown in Figure 2.17.

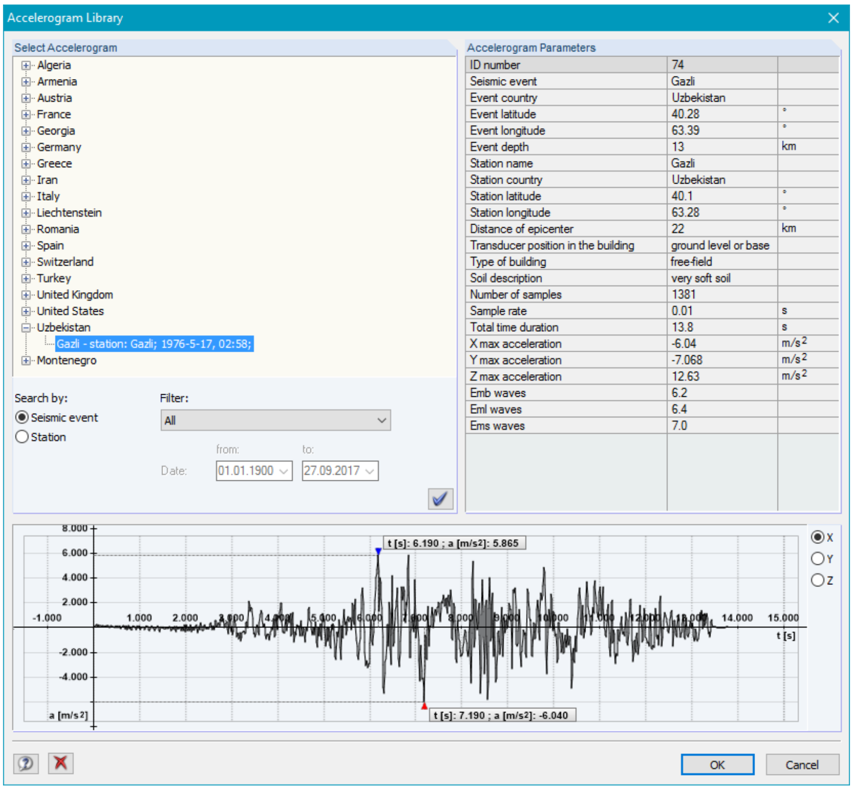
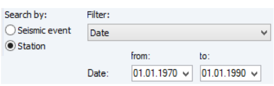



Figure 2.17: Library containing a large number of earthquake recordings

You can *Select the Accelerogram* by country in the tree structure.




It is possible to sort the entries by *Seismic event* or *Station*. You can choose more filter criteria from the drop down menu. For example, it is possible to limit the accelerograms to a particular period of time.

Each accelerogram is identified by its parameters. It is characterized by, for example, the country, latitude and longitude and by the distance to the epicentre. Those data are provided on the right-hand side of the library shown in Figure 2.17.


The chosen accelerogram is displayed in the graphic at the bottom of the dialogue box shown in Figure 2.16. Most of the accelerograms differ in the three directions X, Y, and Z. You can switch between those directions in the graphic.

Accelerograms loaded from the library can be saved with a different file name. The [Save] button opens a dialogue box to enter a file name. Alternatively you can export the accelerograms to Excel.

## 2.6.2 User-Defined Acceleration-Time Diagrams

Any kind of acceleration-time diagram can be defined by entering the required values in the *Accelerogram* table shown in Figure 2.18.

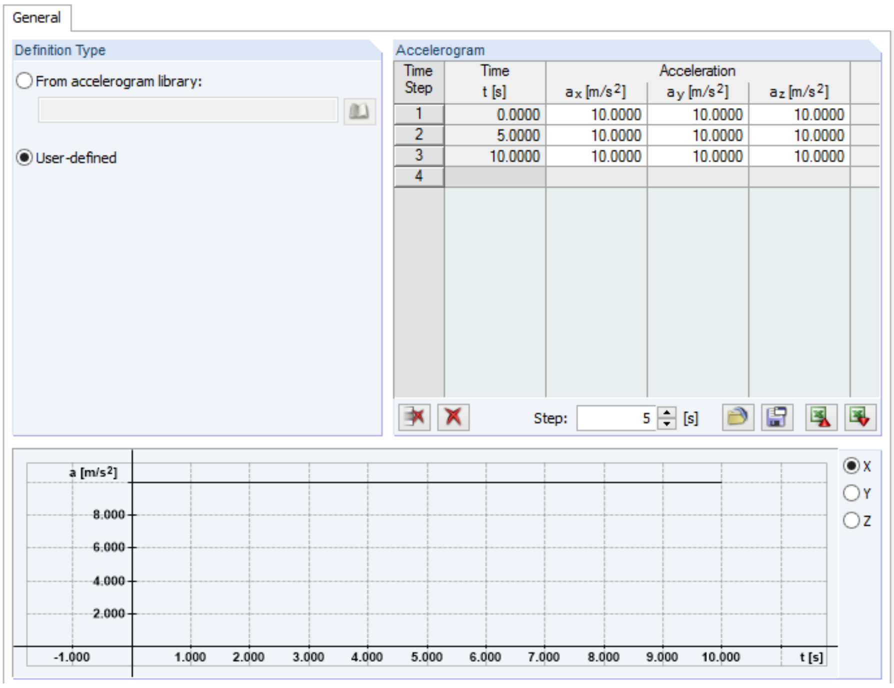
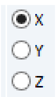



Figure 2.18: User-defined entry of acceleration-time diagram

User-defined accelerograms can be stored in a library and can be opened whenever needed. The [Save] button opens a dialogue box to enter a file name.

You can export your user-defined accelerogram to Excel or import an acceleration-time diagram from Excel.




The defined accelerogram is displayed in the graphic (see Figure 2.16). You can switch between the X, Y, and Z-directions in the graphic.

## 2.7 Time Diagrams

Time diagrams can be defined *Transient*, *Periodic*, or entered as *Function*. They excite the system at a specific position. The load position is defined in static load cases (*LC*) where any type of load can be entered. The static load cases (*LC*) are connected to the time diagrams (*TD*) in the *Dynamic Load Cases* (see Section 2.10.2), and the multiplier *k* is used to determine the final value of the excitation force.

The time diagram tab is only available when *Time diagrams* have been selected in the *General* tab shown in Figure 2.1. This tab belongs to the add-on modules *RF-DYNAM Pro - Forced Vibrations* and *RF-DYNAM Pro - Nonlinear Time History*.

### 2.7.1 Transient Time Diagrams

The *Time Diagram* tab is illustrated in Figure 2.19 with an example of a transient time diagram. The values can be entered in the table. The resulting time diagram is illustrated in the graphic at the bottom. For each time step only one value of *k* shall exist.

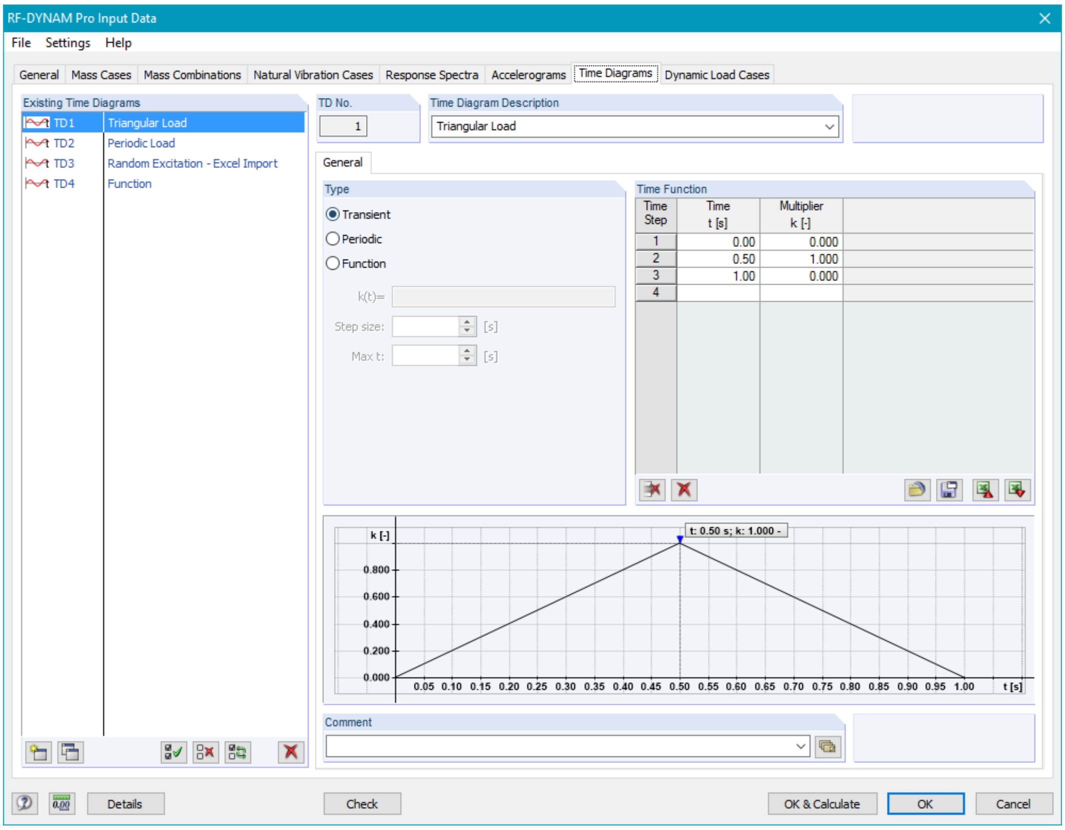



Figure 2.19: Module Window *Time Diagrams* showing *Transient* time diagrams

The table entries can be stored in a library and can be opened whenever needed. Importing and exporting from and to Excel is also possible.

## 2.7.2 Periodic Excitation

To enter periodic functions, the *Angular Frequency* $\omega$ [rad/s], the *Shift* $\varphi$ [rad], and the *Multiplier* $k$ have to be provided in tabular form. This is shown in Figure 2.20.

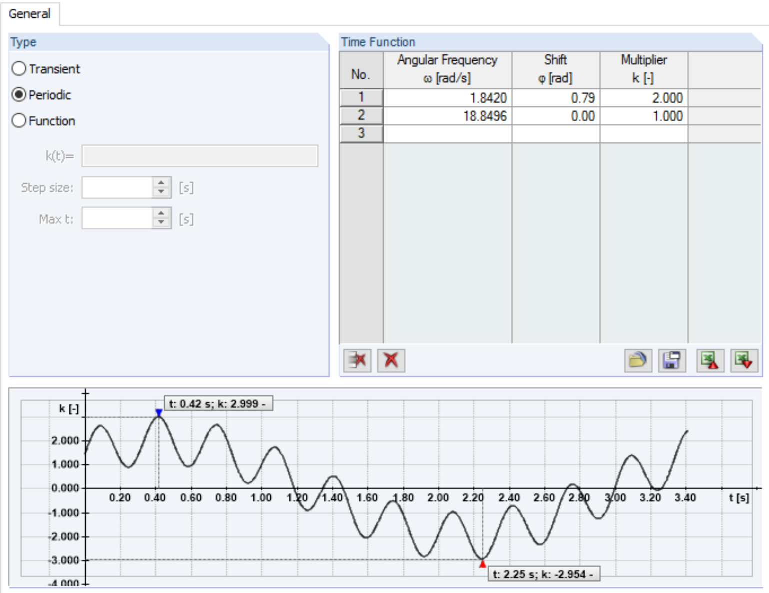



Figure 2.20: Module Window *Time Diagrams* showing *Periodic* time diagrams

Several harmonic functions can be overlain by filling several rows in the table. The periodic functions are defined as follows:

$$f(t) = k_1 \cdot sin\,(\omega_1 \cdot t + \varphi_1) + k_2 \cdot sin\,(\omega_2 \cdot t + \varphi_2) + \ldots \tag{2.7}$$

The usual [Save], [Open], [Import] and [Export] functions are available.

## 2.7.3 Functions

You can enter the *Function k(t)* directly in the text box to define a time diagram. The parameter *t* is reserved for the time. The resulting time diagram is displayed in the graphic as shown in Figure 2.21. The generated tabulated values are listed on the right hand side.

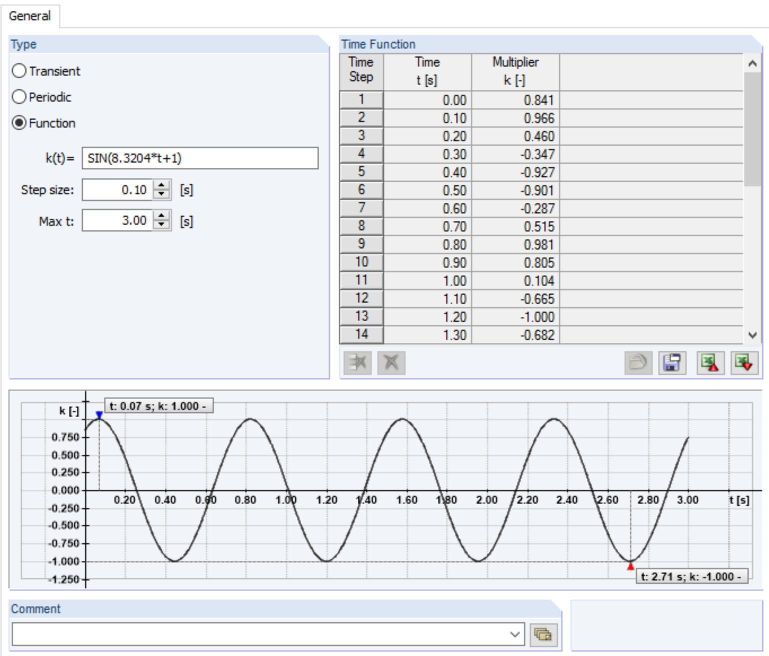



Figure 2.21: Module Window *Time Diagrams* showing *Function* time diagrams

The value *Max t* sets the maximum time for calculation; the time diagram $k(t) = 0$ for all $t > t_{max}$. The *Step size* is used for the tabulated values in the table and for the automatic determination of the time step for calculation (see Section 2.10.5).

All operators and functions that are available in RFEM can be also used in RF-DYNAM Pro. You can use the parameters that you have defined in the main program RFEM. To avoid unit conversion, it is recommended to employ dimensionless parameters in RF-DYNAM Pro. For more information about the parametric input, consult Section 11.6 of the RFEM manual.

The tabulated values can be saved to the library or exported to Excel.

A complex example of how the time diagram functions can be used is shown in the DLUBAL webinar Time History Analysis in RF-/DYNAM Pro - Walking and Running Across Pedestrian Bridge.

## 2.8 Dynamic Load Cases

Dynamic load cases (*DLC*) combine the input that has been made so far and define the calculation parameters for the analysis. Four different types of dynamic load cases are available: *Response spectrum analysis*, linear and nonlinear *Time history analysis of accelerograms*, linear and nonlinear *Time history analysis of time diagrams*, and *Equivalent static force analysis*.

The response spectrum analysis and the linear time history analysis belong to the add-on module *RF-DYNAM Pro - Forced Vibrations*, the nonlinear time history analysis belongs to the add-on module *RF-DYNAM Pro - Nonlinear Time History*, and the equivalent static force analysis belongs to the add-on module *RF-DYNAM Pro - Equivalent Loads*.

The *General* tab of the *Dynamic Load Cases* tab is shown in Figure 2.22.

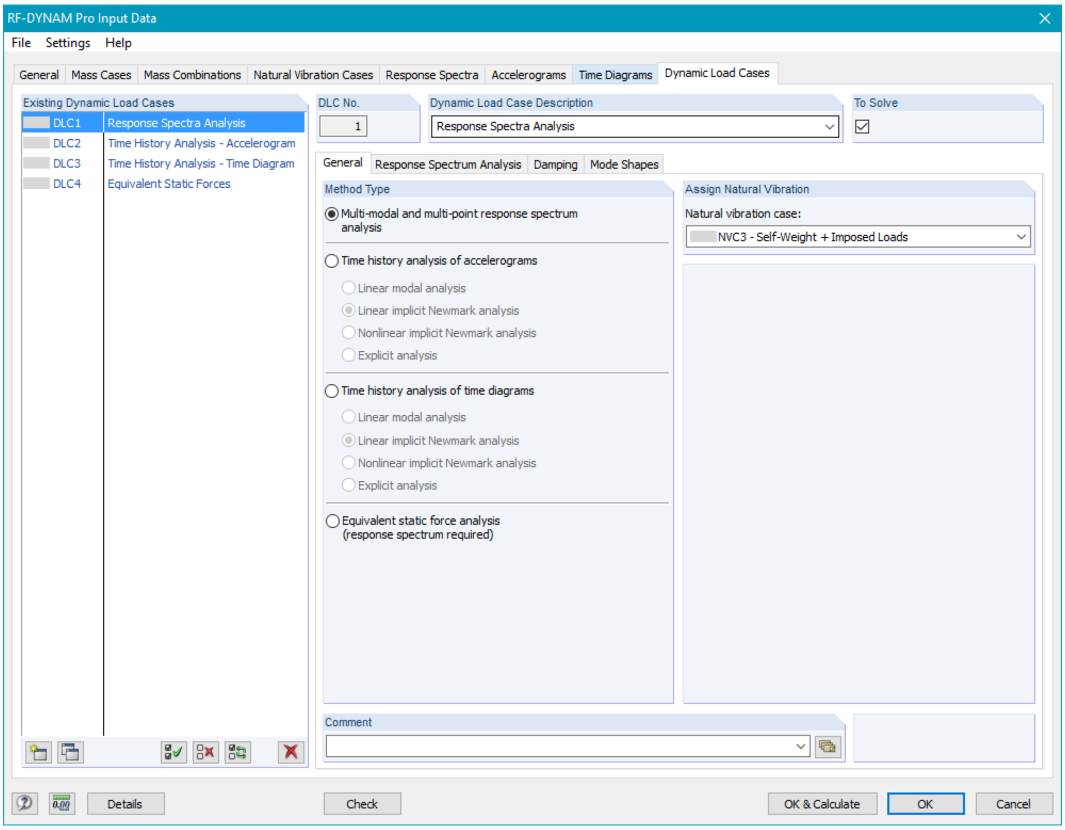
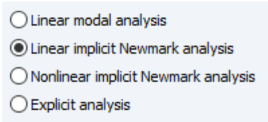
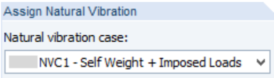



Figure 2.22: Module Window *Dynamic Load Cases* with *General* tab

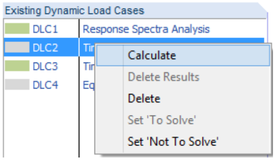
Those options are only available when the specific add-on modules have been activated in the *General* tab of RF-DYNAM Pro (see Figure 2.1).




For the linear time history analysis, two solvers are available – the *modal analysis* which is based on modal decomposition and requires a natural vibration analysis, and the *implicit Newmark solver* which is a direct solver that does not require eigenvalues. For the nonlinear time history analysis, two solvers are available as well – the *implicit Newmark solver* and an *explicit solver*. Details on the time history solvers are provided in Section 2.10.3 and Section 2.10.4.




You have to assign a specific *Natural vibration case (NVC)* (discussed in Section 2.4) to the dynamic load case for a response spectrum analysis, a modal analysis or an equivalent static force analysis.




The dynamic load case (*DLC)*) number is set automatically and cannot be edited. When a case is deleted later, the numbers do not change. You can decide whether the specific *DLC* is to be solved or not by selecting or clearing the *To Solve* option. You can also access this option in the shortcut menu by right-clicking the corresponding *DLC* number.

You can calculate each dynamic load case separately by using its shortcut menu and applying *Calculate*. The colour of a *DLC* is grey when no results of this dynamic load case are available yet. It turns green as soon as the calculation has finished. For more information, see Section 3.3.

Load cases (*LC*) and/or result combinations (*RC*) are generated in each dynamic load case when the check boxes in the *To Generate* section have been selected (see *Response Spectra Analysis* tab in Figure 2.23, *Time History Analysis* tab in Figure 2.26 and Figure 2.27, and *Equivalent Force Analysis* tab in Figure 2.32). Those load cases and result combinations are generated automatically and are overwritten when the RF-DYNAM Pro calculation is performed again. The descriptions of the load cases and result combinations clearly refer to the originating *DLC*s.

You can enter the number of the first generated load case or result combination. RF-DYNAM Pro does not overwrite existing load cases or result combinations (static or generated from other add-on modules). It chooses the first unused *LC* or *RC* number available. Load cases and result combinations from RF-DYNAM Pro are deleted when the results of the corresponding dynamic load case (*DLC*) are deleted.

It is possible to activate the export options when the calculation is finished. A re-calculation is hereby not required.

In the following sections of this manual, the *Response Spectrum Analysis*, the *Time History Analysis* and the *Equivalent Static Force Analysis* are discussed in detail.

## 2.9 Dynamic Load Cases - Response Spectrum Analysis

A multi-modal and multi-point response spectra analysis is performed when the corresponding radio button in Figure 2.22 is selected; this is only available in *RF-DYNAM Pro - Forced Vibrations*. The definition of response spectra was discussed in Section 2.5. Mode shapes from the assigned natural vibration case (*NVC*) can be selected for the analysis. The *SRSS* or the *CQC* rule is available for modal combination. The calculation is done within the module *RF-DYNAM Pro - Forced Vibrations* and is therefore completely linear. Stiffness modifications defined in the *NVCs* (Section 2.4.7) are used for the analysis. The final results of the response spectrum analysis are exported to result combinations (*RC*).

The sub-tabs when performing a response spectra analysis are *General*, *Response Spectrum Analysis*, *Damping*, and *Mode Shapes*. The *Response Spectrum Analysis* tab is illustrated in Figure 2.23.

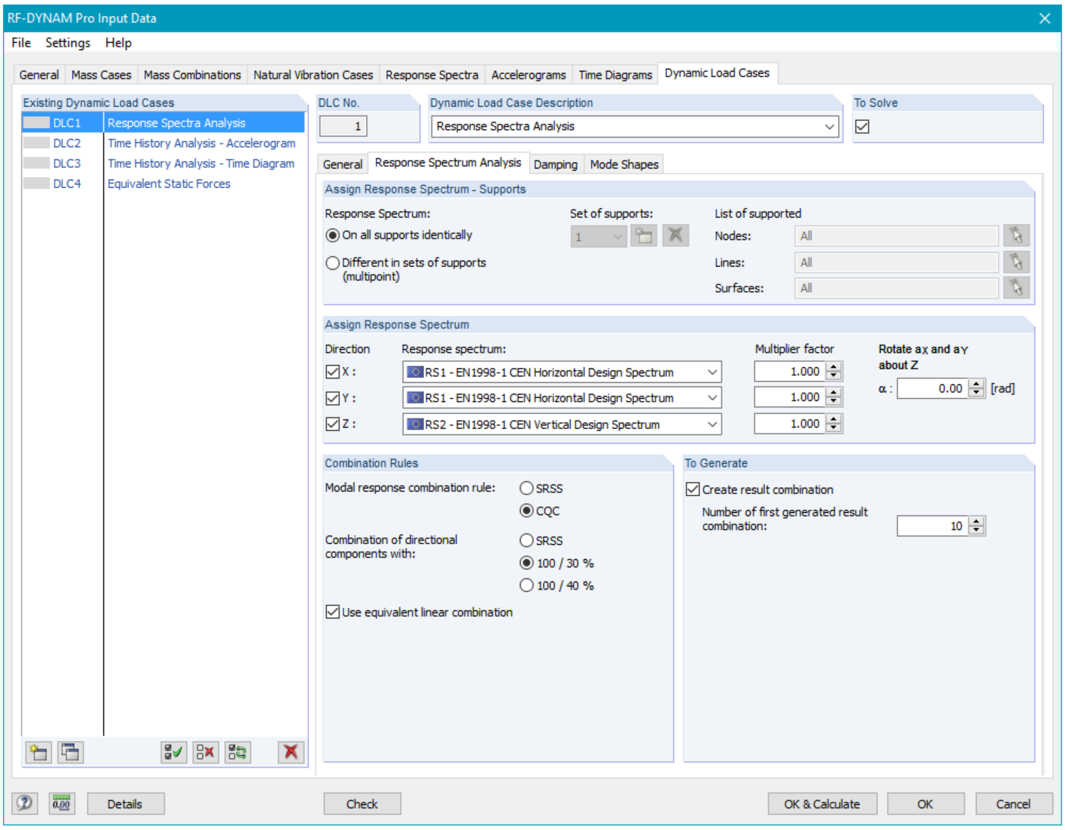
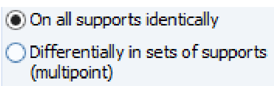



Figure 2.23: Module Window *Dynamic Load Cases* with *Response Spectrum Analysis* tab

### 2.9.1 Assign Supports



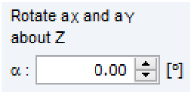

The supports that are excited by the response spectrum have to be assigned. You can either excite all supports with identical response spectra, or you can use the multi-point option and create several sets of supports.


You can create several *Sets of supports* by clicking . Switch in between the available sets by using the drop-down menu. Use the button to delete a specific set. For each set, a list of supported nodes, lines or surfaces must be provided. It can be entered manually or selected in the work window via the button.

For background on the multi-point excitation, the reader is referred to the literature [8 and 10].

### 2.9.2 Assign Response Spectrum

A specific response spectrum can be assigned in each direction. At least one of the directions must have been selected. The relevant *Response spectrum* can be chosen from the drop-down menu. The *Multiplier factor* can be adjusted independently for each direction.




The excitation direction can be rotated in the XY-plane about the Z-axis. For example, when your response spectrum shall excite the structure 45 ° rotated about the Z-axis, activate only the X-direction with your response spectrum and enter $\alpha = 45$ °.

For each set of supports, the response spectrum in each specific direction must be activated separately. This allows for different response spectra for each set of supports.

### 2.9.3 Combination of Modal Responses

In these settings you define how the responses resulting from different eigenvalues of the structure are combined. The modal combination is the first step of dynamic combination.


The modal response can be combined using the *Square Root of the Sum of the Squares* (*SRSS*) rule or the *Complete Quadratic Combination* (*CQC*) rule. Both of those quadratic combinations can be applied in the standard form or modified as *equivalent linear combination*.

The standard form of *SRSS* combines maximum results. The signs are lost as seen in the formula below:

$$R_{SRSS} = \sqrt{R_1^2 + R_2^2 + \ldots + R_p^2} \tag{2.8}$$

The combined results $R_{SRSS}$ result from the modal responses $R_p$ from $p$ modes of the structure.

In RF-DYNAM Pro, a modified form of the *SRSS* rule is available in order to calculate corresponding results, *i.e.* corresponding internal forces. Compared to the standard form of *SRSS* rule, those corresponding results are usually much smaller and the signs are correct in relation to the controlling force. The *SRSS* rule is applied as equivalent linear combination [11] as follows:

$$R_{SRSS} = \sum_{i=1}^{p} f_i \cdot R_i \;\text{ with }\; f_i = \frac{R_i}{\sqrt{\sum\limits_{j=1}^{p} R_j^2}} \tag{2.9}$$

By this formula the result combinations are consistent in itself.

The *SRSS* rule is only allowed for systems where adjacent natural periods $\mathrm{T_i} < \mathrm{T_j}$ differ more than 10%, so when the following statement is true:

$$\frac{T_i}{T_j} < 0.9 \tag{2.10}$$

In all other cases, the *CQC* rule must be applied. The *CQC* rule is defined as follows:

$$R_{CQC} = \sqrt{\sum_{i=1}^{p}\sum_{j=1}^{p} R_i \varepsilon_{ij} R_j} \tag{2.11}$$

where the correlation coefficient $\varepsilon$ is

$$\varepsilon_{ij} = \frac{8 \cdot \sqrt{D_i D_j}\left(D_i + rD_j\right) r^{3/2}}{(1-r^2)^2 + 4D_iD_jr\,(1+r^2) + 4\left(D_i^2 + D_j^2\right) r^2} \;\text{ with }\; r = \frac{\omega_j}{\omega_i} \tag{2.12}$$

The correlation coefficient $\varepsilon$ simplifies when the viscose damping value $D$ is equal for all modes to the following:

$$\varepsilon_{ij} = \frac{8 \cdot D^2(1+r)r^{3/2}}{(1-r^2)^2 + 4D^2r(1+r)^2} \tag{2.13}$$

The damping value $D_i$ that is required for the calculation of the correlation coefficient $\varepsilon_{ij}$ is defined in the *Damping* tab that appears as soon as you have selected the *CQC* rule. The settings for damping are described below.

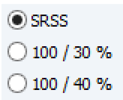
Analogously to the *SRSS* rule, the *CQC* rule can be applied as equivalent linear combination [11]. The *CQC* rule, still employing $\varepsilon_{ij}$ as defined in Equation 2.12, is modified as follows:

$$R_{CQC} = \sum_{i=1}^{p} f_i \cdot R_i \quad \text{with} \quad f_i = \frac{\sum_{j=1}^{p} \varepsilon_{ij} R_j}{\sqrt{\sum_{i=1}^{p}\sum_{j=1}^{p} R_i \varepsilon_{ij} R_j}} \tag{2.14}$$

Further information and mathematical derivations of the combination rules are provided in [12–14].

### 2.9.4 Combination of Directional Components

In the settings you also define how the responses resulting from different excitation directions are combined. The directional combination is the second step of dynamic combination.




The internal forces resulting from different excitation directions can be combined quadratically with the *SRSS* rule, or using the 100% / 30% (40%) rule. The *SRSS* rule is applied as defined in Equation 2.8 but now i = 1..p are the excitation directions X, Y and Z. The *SRSS* rule for the directional combination can also be realised as equivalent linear combination applying Equation 2.9.

### 2.9.5 Export Result Combinations

When you select *Create result combinations*, RF-DYNAM Pro automatically generates and overwrites result combinations during the calculation. The *RC*s are tied to the results, which means that they are deleted as soon as the results of the *DLC* are deleted (see Section 3.3).

One *RC* is generated for the *SRSS* rule, but a maximum of three result combinations is generated when you choose the 100% / 30% (40%) rule.

More information about the exported *Result Combinations* is provided in Section 4.2.

### 2.9.6 Damping for CQC rule

The *Damping* tab is only available when the *CQC* rule is chosen for the combination of mode shapes (defined in Equation 2.11 and Equation 2.12).

For the CQC rule, the Lehr's damping values $D_i$ are needed. Those can be defined equally or differently for each mode of the system. The Rayleigh damping with the coefficients $\alpha$ and $\beta$ is also available, but those values are internally converted to the Lehr's damping.

The *Damping* tab is shown in Figure 2.24.

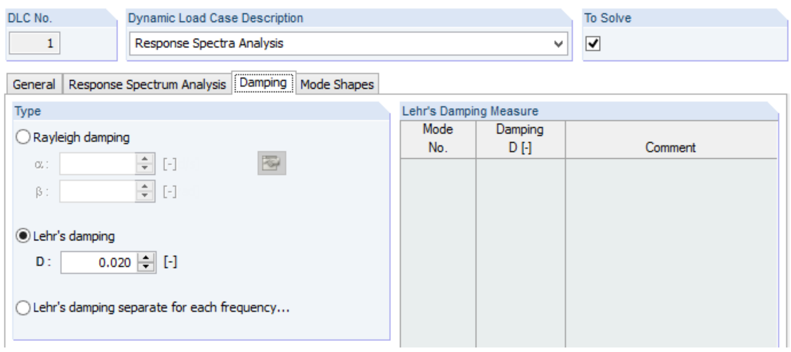
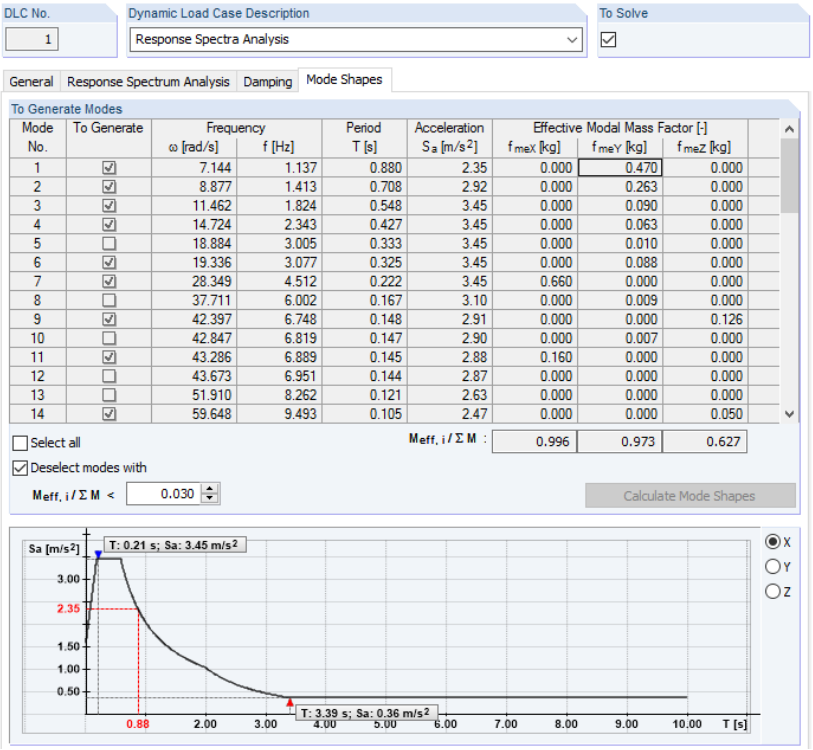



Figure 2.24: Module Window *Dynamic Load Cases* with *Damping* tab

For the conversion from Rayleigh coefficients $\alpha$ and $\beta$ to the Lehr's damping $D_i$ it is referred to Section 2.10.9.

## 2.9.7 Mode Shape Selection

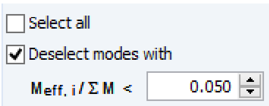
In the *Mode Shapes* tab illustrated in Figure 2.25, the natural frequencies ω and *f* as well as the periods *T* are listed with the corresponding accelerations $S_a$ of the response spectrum and the effective modal mass factors $f_{me}$ in the translational directions. The assigned response spectrum is illustrated in the graphic. Corresponding values are displayed in red when you have selected a row in the table as shown in Figure 2.25.




| Mode No. | To Generate | Frequency ω [rad/s] | Frequency f [Hz] | Period T [s] | Acceleration $S_a$ [m/s²] | Effective Modal Mass Factor [-] $f_{meX}$ [kg] | $f_{meY}$ [kg] | $f_{meZ}$ [kg] |
|---|---|---|---|---|---|---|---|---|
| 1 | ☑ | 7.144 | 1.137 | 0.880 | 2.35 | 0.000 | 0.470 | 0.000 |
| 2 | ☑ | 8.877 | 1.413 | 0.708 | 2.92 | 0.000 | 0.263 | 0.000 |
| 3 | ☑ | 11.462 | 1.824 | 0.548 | 3.45 | 0.000 | 0.090 | 0.000 |
| 4 | ☑ | 14.724 | 2.343 | 0.427 | 3.45 | 0.000 | 0.063 | 0.000 |
| 5 | ☐ | 18.884 | 3.005 | 0.333 | 3.45 | 0.000 | 0.010 | 0.000 |
| 6 | ☑ | 19.336 | 3.077 | 0.325 | 3.45 | 0.000 | 0.088 | 0.000 |
| 7 | ☑ | 28.349 | 4.512 | 0.222 | 3.45 | 0.660 | 0.000 | 0.000 |
| 8 | ☐ | 37.711 | 6.002 | 0.167 | 3.10 | 0.000 | 0.009 | 0.000 |
| 9 | ☑ | 42.397 | 6.748 | 0.148 | 2.91 | 0.000 | 0.000 | 0.126 |
| 10 | ☐ | 42.847 | 6.819 | 0.147 | 2.90 | 0.000 | 0.007 | 0.000 |
| 11 | ☑ | 43.286 | 6.889 | 0.145 | 2.88 | 0.160 | 0.000 | 0.000 |
| 12 | ☐ | 43.673 | 6.951 | 0.144 | 2.87 | 0.000 | 0.000 | 0.000 |
| 13 | ☐ | 51.910 | 8.262 | 0.121 | 2.63 | 0.000 | 0.000 | 0.000 |
| 14 | ☑ | 59.648 | 9.493 | 0.105 | 2.47 | 0.000 | 0.000 | 0.050 |
| | | | | | Meff, i / Σ M : | 0.996 | 0.973 | 0.627 |

Figure 2.25: Module Window *Dynamic Load Cases* with *Modes Shapes* tab – first mode selected in table


When no calculation has been performed yet, use the button [Calculate Mode Shapes] to calculate the frequencies internally without closing the module.




In the table, you can select modes that shall be used for the response spectra analysis. All modes will be selected when the *Select all* option is activated. In this case, the selection cannot be changed manually. By using the *Deselect modes with* option, you can deactivate modes with an effective modal mass factor below a specific value. When both check boxes are clear, you can select the modes manually in the *To Generate* column.

The sums of the effective modal mass factors $f_{me}$ are shown at the bottom of the table. According to EN 1998-1 Section 4.3.3.3 [1], the effective modal mass factors of all modes taken into account shall be at least 90%. When this cannot be achieved, all modes with a factor above 5% shall be taken into account. Further details about the effective modal mass factors and how those are calculated are provided in Section 4.1.


You can switch between the excitation directions X, Y and Z in the graphic. Note that also the values $S_a$ in the table update depending on this choice.

The values $S_a$ in the table and the diagram reflect the multiplier factors applied to each direction (see *Response Spectrum Analysis* tab in Figure 2.23). Note that – if you have rotated your assigned response spectrum around the angle $\alpha$ – the directions X and Y are local directions in the *Mode Shapes* tab.

When you use the multi-point option and have defined several sets of supports in the *Response Spectrum Analysis* tab (see Figure 2.23), the values of $S_a$ in the table and the diagram in the *Mode Shapes* tab change depending on the selected set of support in the drop-down menu.

## 2.10 Dynamic Load Cases - Time History Analysis

A time history analysis is performed when either the *Time history analysis of accelerograms* or the *Time history analysis of time diagrams* has been selected in the *General* sub-tab (see Figure 2.22). The excitations and the results of a time history analysis are both data versus time.

The corresponding sub-tabs when performing a time history analysis are *General*, *Time History Analysis*, *Calculation Parameters*, and *Damping*. The *Time History Analysis* tab is illustrated in Figure 2.26 and Figure 2.27. Note the difference in the user interface depending on the type of excitation: Accelerograms excite the structure at the supports. The time history analysis of accelerograms is discussed in Section 2.10.1. Time diagrams, however, can excite the structure everywhere. The type of load and location of excitation is defined by load cases (*LC*) in RFEM. The time history analysis of time diagrams is discussed in Section 2.10.2.

The time history analysis can be performed linearly, neglecting geometrically and structural non-linearities with the add-on module *RF-DYNAM Pro - Forced Vibrations*. Two linear solvers are available – the *Linear modal analysis* and the *Linear implicit Newmark analysis*. They are described in Section 2.10.3.

A nonlinear time history analysis is available with the add-on module *RF-DYNAM Pro - Nonlinear Time History*. A large strain deformation analysis is performed under consideration of the defined nonlinearities. Two nonlinear solvers are available – the *Nonlinear implicit Newmark analysis* and the *Explicit analysis*. They are described in Section 2.10.4.

Time steps for the analysis are discussed in Section 2.10.5. The mass matrix settings are described in Section 2.10.8 and the structural damping options in Section 2.10.9.

### 2.10.1 Time History Analysis of Accelerograms

Time history analysis of accelerograms is used to design the structure for the event of an earthquake. Accelerograms excite the structure at its supports.

$$\mathbf{M}\,\ddot{\mathbf{u}} + \mathbf{D}\,\dot{\mathbf{u}} + \mathbf{K}\,\mathbf{u} = -\mathbf{M}\,\ddot{u}_g \tag{2.15}$$

in which **M** is the mass matrix, **D** is the damping matrix and **K** is the static stiffness matrix. The time dependent vectors $\mathbf{u}$, $\dot{\mathbf{u}}$ and $\ddot{\mathbf{u}}$ are the absolute displacements, velocities and accelerations. $\ddot{u}_g$ is the ground acceleration which can be different in the different directions.

The definition of accelerograms was discussed in Section 2.6. In RF-DYNAM Pro, all supports can be excited simultaneously with the same accelerogram. The multi-point option makes it possible to excite single supports by different accelerograms (only available for linear solvers).

When nonlinearities play a role in the model, the time history of accelerograms might be favoured compared to an response spectrum analysis.

The *Time History Analysis* sub-tab is illustrated in Figure 2.26.

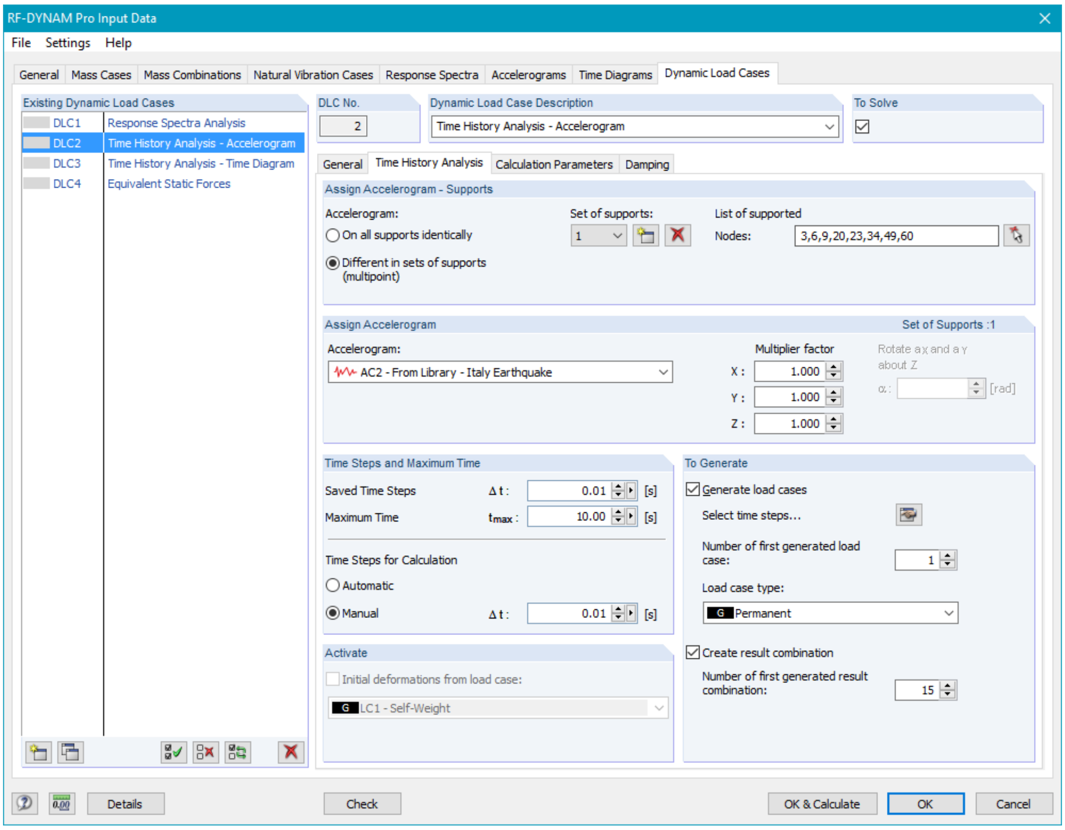
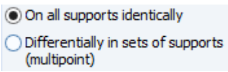
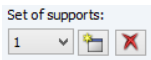



Figure 2.26: Module Window *Dynamic Load Cases* with *Time History Analysis* tab of *Accelerograms*

## Assign Supports




The supports that are excited by the accelerogram have to be assigned. When you consider all supports to be identically excited, then those supports can be nodal, line or surface supports. The multi-point option is available for nodal supports only at the moment. Supports defined with a spring constant cannot be used for a time history analysis of accelerograms at the moment. The multi-point option is only available for a linear time history analysis.

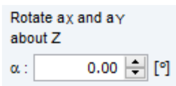



You can create several sets of supports by clicking the button. You can switch between the sets by using the drop-down menu. Use the button to delete a specific set. For each set, a list of supported nodes must be provided. It can be entered manually or selected graphically by using the function.

For background on the multi-point excitation, the reader is referred to the literature [8 and 10].

## Assign Accelerogram

Choose the available accelerograms from the drop-down menu. You can apply different multiplier factors in each direction.




The excitation direction can be rotated in the XY-plane about the Z-axis. For example, when your accelerogram shall excite the structure 45 ° rotated about the Z-axis, activate only the X-direction with your response spectrum chosen from the drop-down menu and enter $\alpha = 45$ °.

You can assign a specific accelerogram to each set of supports.

## 2.10.2 Time History Analysis of Time Diagrams

With a time history analysis of time diagrams, any point of the system can be excited in any direction.

$$\mathbf{M}\,\ddot{\mathbf{u}} + \mathbf{D}\,\dot{\mathbf{u}} + \mathbf{K}\,\mathbf{u} = p(t) \tag{2.16}$$

in which **M** is the mass matrix, **D** is the damping matrix and **K** is the static stiffness matrix. The time dependent vectors **u**, $\dot{\mathbf{u}}$ and $\ddot{\mathbf{u}}$ are the absolute displacements, velocities and accelerations. $p(t)$ is the external excitation.

The definition of time diagrams was discussed in Section 2.7. In the *Dynamic Load Cases*, those time diagrams are combined with load cases (*LC*) to define the type, the location and direction of the excitation.

The *Time History Analysis* sub-tab is illustrated in Figure 2.27.

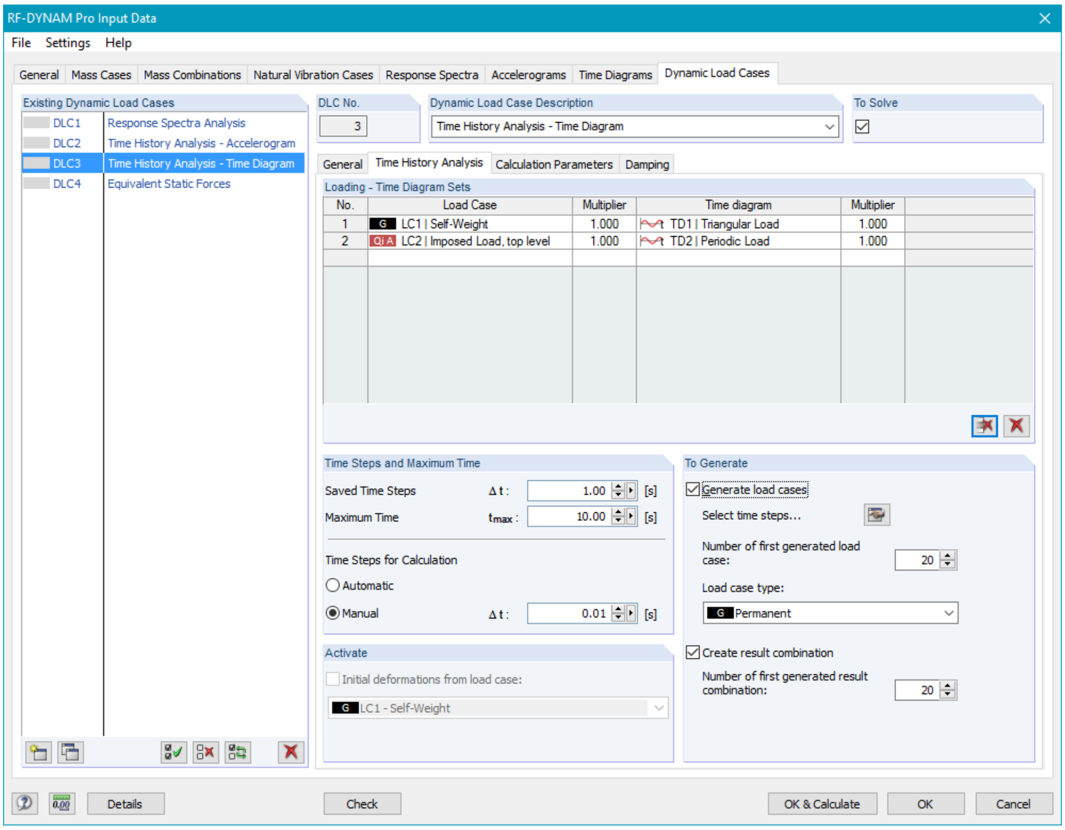



Figure 2.27: Module Window *Dynamic Load Cases* with *Time History Analysis* tab of *Time Diagrams*

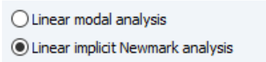
## Loading - Time Diagram Sets

Load cases defined in the main program RFEM are to be combined with *Time Diagrams* (see Section 2.7) to *Loading - Time Diagram Sets*. In the table, choose a load case by activating the list via and allocate a time diagram to define the excitation of your structure. You can apply factors for both the load case and the time diagram.

Static load cases are mandatory when performing a time history analysis of time diagrams. The static load case defines the magnitude, direction and positions of the excitation. Nodal, line, surface, free or generated loads can be combined with a function varying over time.

You can combine many of those *Loading - Time Diagram Sets* by filling more rows in the table as shown in Figure 2.27. This is required to simulate an excitation that is varying in time but also changing its position, e.g. a pedestrian walking across a bridge as demonstrated in the webinar Time History Analysis in RF-/DYNAM Pro - Walking and Running Across Pedestrian Bridge.

### 2.10.3 Linear Solvers: Modal Analysis versus Implicit Newmark



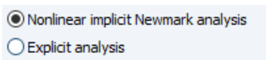

In the *General* sub-tab (see Figure 2.22), you can choose between two linear time history solvers – the *Linear modal analysis* and the *Linear implicit Newmark solver*. The linear analysis is geometrically linear (small deformations) and ignores or replaces all defined non-linearities in the model, *i.e.* cables are replaced by trusses and failure of a support is ignored. The linear solvers belong to the add-on module *RF-DYNAM Pro - Forced Vibrations*.

The **modal analysis** uses a decoupled system based on the eigenvalues and mode shapes of the structure, determined in the assigned natural vibration case (*NVC*). The multi-degree-of-freedom (MDOF) system is transformed into many single-degree-of-freedom-systems (SDOF) (diagonalized mass and stiffness matrix). A certain amount of eigenvalues is required to ensure accuracy. The solution of the decoupled system is then found with an implicit (*Newmark*) solver. The mass matrix settings and stiffness modifications are taken over from the *NVC* assigned. Initial deformations can be applied. Once the eigenvalues are determined, the modal analysis provided in *RF-DYNAM Pro - Forced Vibrations* is slightly faster than the linear implicit Newmark analysis.

The **linear implicit Newmark solver** is a direct time integration scheme that requires sufficient small time steps to achieve accurate results. There is no natural vibration analysis required. Settings for the mass matrix are defined in the *Calculation Parameters* of the *DLC*s. Stiffness modifications and initial deformations are not available with this type of solver. For more theoretical background, see e.g. [10]. The choice of time step is discussed in Section 2.10.5.

For the direct time history solver, dashpot elements can be used. This member type is available in RFEM where you define the viscous damping coefficient *c* as well as the axial stiffness constant (spring element) which is then acting parallel (Kelvin-Voigt-Model).

### 2.10.4 Nonlinear Solvers: Implicit Newmark versus Explicit




In the *General* sub-tab (see Figure 2.22), you can choose between two nonlinear time history solvers – the *Nonlinear implicit Newmark analysis* and the *Explicit analysis*. Both nonlinear solvers perform a large strain deformation analysis and consider all nonlinearities that are available in the main program RFEM. The nonlinear solvers belong to the add-on module *RF-DYNAM Pro - Nonlinear Time History*.

The **explicit anaylsis**, or method of central differences, is suitable for short duration excitations and rapidly changing nonlinearities in the structure. The method is 'explicit' because unknown values depend only on the time *i*, not on the unknown response at time *i + 1*. The explicit integration rule works well in combination with a diagonal mass matrix and with the restriction in damping matrix $\mathbf{C} = \alpha\mathbf{M}$. The method is conditionally stable. A bounded solution is obtained only when the time increment $\Delta t$ is less than the stable time increment $\Delta t_{\text{stable}}$. The stability limit can be defined in terms of the highest eigenvalue in the model $\omega_{\text{max}}$ and the fraction of critical damping $D$ in the highest mode.

$$\Delta t_{\text{stable}} \leq \frac{2}{\omega_{\text{max}}} \left( \sqrt{1 + D^2} - D \right) \tag{2.17}$$

In practice, the stable time increment is estimated by

$$\Delta t_{\text{stable}} = \frac{L^e}{c_d} \tag{2.18}$$

where $L^e$ is the finite element length and $c_d$ the dilatational wave speed that is for linear elastic material (with Poisson's ratio equal to zero) equal to

$$c_d = \sqrt{\frac{E}{\rho}} \tag{2.19}$$

where E is the Young's modulus and $\rho$ the material density.

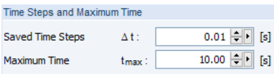
This estimation provides a smaller time step than the stability limit in Equation 2.17. Note also that many effects are not included in the time step estimation and that for accuracy reasons an even smaller time step $\Delta t$ may be required. RFEM uses a fixed time incrementation where the fixed time step can be the initial stable time increment or a user-defined value.

Implicit schemes solve unknown values at time $i+1$ based on the values at time $i$ and $i+1$. Thus, nonlinear equations must be solved, iterations and convergence checks are required. The **implicit Newmark analysis** is unconditionally stable, there is no upper stability bound in time step $\Delta t$. Still, a reasonable small time increment is required to achieve accurate results. The time step depends on the excitation, the frequency of the structure and the complexity of nonlinearities in the structure. The implicit solver needs "time steps typically one or two orders of magnitude larger than the stability limit of simple explicit schemes" [15]. There are no restrictions regarding the mass matrix and Rayleigh damping when using the implicit Newmark solver.

The choice between the two nonlinear solvers depends mainly on the required time step but also on the type of excitation. For both solvers, a time step convergence study must be done. For further theoretical background, see [10 and 15–17].

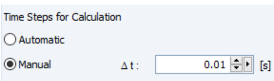
For both nonlinear solvers, dashpots element can be used. This member type is available in RFEM where you define the viscous damping coefficient $c$ as well as the axial stiffness constant (spring element) which is then acting parallel (Kelvin-Voigt-Model). The spring can be defined with slippage and/or as diagram. Those nonlinear features of dashpot elements are only applicable to the nonlinear solvers.

### 2.10.5 Time Steps and Maximum Time




You need to define the time steps to be saved and the maximum time of your time history analysis. Results are only available for the time steps that are saved, and also the dynamic envelope is built from those saved time steps.

Saving less time steps reduces the file size, shortens the calculation time and improves the performance of the post-processing. However, a certain amount of result values is required to avoid skipping the maxima and to ensure a smooth graph in the *Time Course Monitor* (see Section 4.3).




Regardless of the time steps that are finally saved, you need to define the time steps used for the calculation. A suitable choice of time step is essential for a successful time history analysis. It is a compromise between calculation time and accuracy. A highly nonlinear calculation requires far smaller time steps than a linear analysis. For the explicit solver, there exists a stable time increment to obtain a solution.

For linear time history analysis, the following advice can be given [18]:

- Considering the accelerogram (see Section 2.6) and transient time diagram (see Section 2.7), the shortest length of the discrete excitation shall be split into at least seven time steps.
- The highest frequency $f$ of the structure (see Section 4.1) that is relevant for the time history response shall be used to calculate the time step with $\Delta \mathrm{t} \leq 1/(20\,\mathrm{f})$. Analogously, the largest frequency of the excitation (see Section 2.7) shall be checked with $\Delta \mathrm{t} \leq \pi \,/\, (10\,\omega)$.

You can define the time step for calculation manually, or RF-DYNAM Pro determines it automatically.

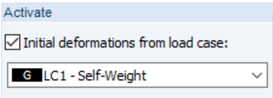
For the *modal analysis* and the linear and nonlinear *implicit Newmark solvers*, RF-DYNAM Pro chooses the time steps for calculation automatically according to the assigned excitation. For accelerograms, transient and function time diagrams, the automatic time step is determined using the time step in between the data points, $\Delta \mathrm{t} = \mathrm{Min}\{\mathrm{t}_{i+1} - \mathrm{t}_i\}/7$. For periodic time diagrams, the automatic time step is determined with $\Delta \mathrm{t} = \pi/(10\,\omega)$ using the largest excitation frequency.

Note that this automatically determined time step is only in accordance to the excitation. Other values, such as the frequency of the structure and non-linearities, are not taken into account, but they do influence the required time step. For the implicit solvers, it is therefore recommended to define the time step for calculation manually.

The *explicit solver* determines the automatic time increment in accordance to the stability limit as discussed in Equations 2.17 and 2.18. Usually, this time step is sufficient also in terms of accuracy. Nevertheless, effects like e.g. rapid changes in stiffness are not included in the time step estimation. For highly nonlinear systems, an even smaller time step $\Delta t$ may be required for accuracy reasons.

A time step convergence study should be performed in any case. It is essential for nonlinear time history analyses.

### 2.10.6 Initial Deformation




When you select *Initial deformations from load case*, you can import initial conditions directly from a load case. Those are the conditions at time step $t = 0$ s that are released shortly after.

In RF-DYNAM Pro, the initial deformations are only available when the *modal analysis* has been selected. For the direct solvers the functions *Initial condition* and *Stationary state* are available. These functions can be selected in the *Calculation Parameters* tab and are described in Section 2.10.8.

### 2.10.7 Export of Load Cases and Result Combinations

You can export load cases for single time steps by activating the *Generate load cases* check box. Select the *Load case type* from the drop down list below. The exported load cases will not contain any loads: they contain the results at the specific time step.

In the list shown in Figure 2.28, you can *Select time steps* that are to be exported.

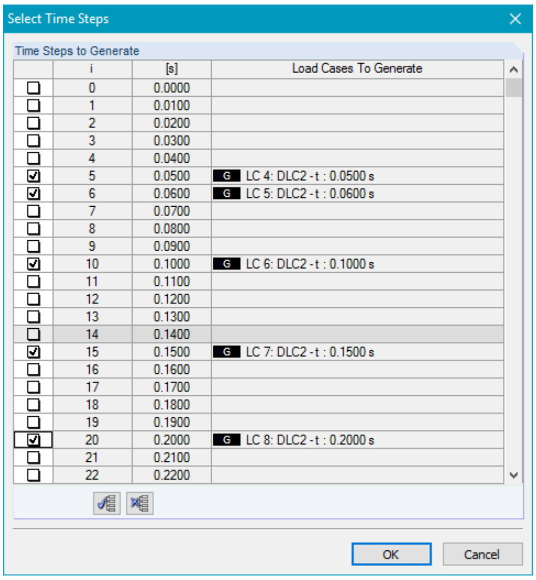



Figure 2.28: Dialogue box *Select Time Steps* for export

A result combination (*RC*) as a result envelope with maximum and minimum values of all saved time steps is generated when you select the *Create result combination* option.

More information on the exported load cases and result combination is provided in Section 4.3.

## 2.10.8 Calculation Parameters

The calculation parameters of time history *DLC*s are only available for direct solvers, such as the *linear* and *nonlinear implicit Newmark solver* and the *explicit solver*. Here, you define the mass matrix and the directions of acting masses. The *Calculation Parameters* sub-tab is shown in Figure 2.29.

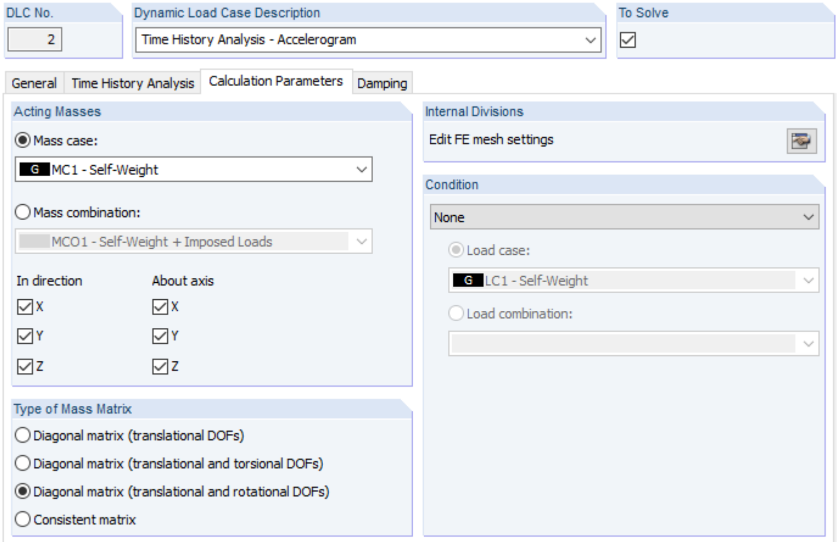
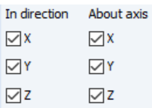



Figure 2.29: Module Window *Dynamic Load Cases* with *Calculation Parameters* tab (direct solvers only)

In case of the *Modal Analysis*, the settings of acting masses, mass matrix and FE mesh settings are taken over from the assigned *NVC*.

### Acting Masses

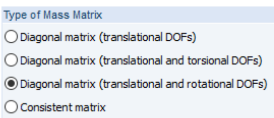
You can import a specific *Mass cases* or *Mass combination* into each *DLC*. Those have been defined as described in Section 2.2 and Section 2.3. Select the relevant mass case or combination from the drop-down menu.




You have to define the direction in which the masses are acting. The masses act in the global translational X, Y, or Z-directions when you select the corresponding check boxes. They act rotationally about the global X, Y, and Z-axes when the corresponding check boxes have been activated. Those settings change your mass matrix.

When the explicit solver is used, masses have to act on each FE-node and in all directions.

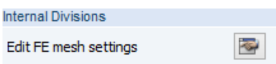
### Type of Mass Matrix




There are four types of mass matrices available in this section of the *Calculation Parameters* sub-tab (see Figure 2.29). The different types of mass matrices are discussed in Section 2.4.4.

The explicit solver is restricted to the *Diagonal matrix (translational and rotational DOFs)*.

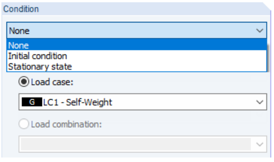
### FE Mesh Settings




The button links directly to the FE mesh settings of the main program RFEM. The *FE Mesh Settings* dialogue box is also available in the **Calculation** → **FE Mesh Settings** menu of RFEM. Further details on the FE mesh and the parameters that can be adjusted are described in the RFEM manual, Section 7.2.

Members that are not integrated into surfaces are not divided into FE-nodes by default. When a diagonal mass matrix is used, the mass of a member is lumped to the two end nodes of the member. For a more refined distribution of the mass, you need to activate the member division in the FE mesh settings.

## Initial Condition and Stationary State




The functions *Initial condition* and *Stationary state* are available. With the option *Initial condition* all conditions from the selected load case or load combination at time step $t = 0$ s are imported and then released shortly after. The function *Stationary state* considers all conditions over the entire time.

In addition to the imported deformations and loads used to establish an equilibrium state at the beginning of the calculation, all stiffness modifications made in the settings of the load cases and load combinations are taken into account. These have already been explained in Section 2.4.7. Furthermore the states of nonlinearities are imported in the nonlinear solvers.

It is possible to choose between load cases and load combinations. For the linear solver, only load cases or load combinations can be used which are calculated according to a geometrically linear analysis. Load cases and load combinations which are calculated according to a large deformation or postcritical analysis can only be used for one of the two nonlinear solvers, however.

## 2.10.9 Structural Damping

In RF-DYNAM Pro, structural viscose damping is available. It can be defined by the Rayleigh coefficients $\alpha$ and $\beta$ or the Lehr's damping values $D_i$. The Lehr's damping value can be equal or different for each mode of the system.

The *Damping* sub-tab is shown in Figure 2.30.

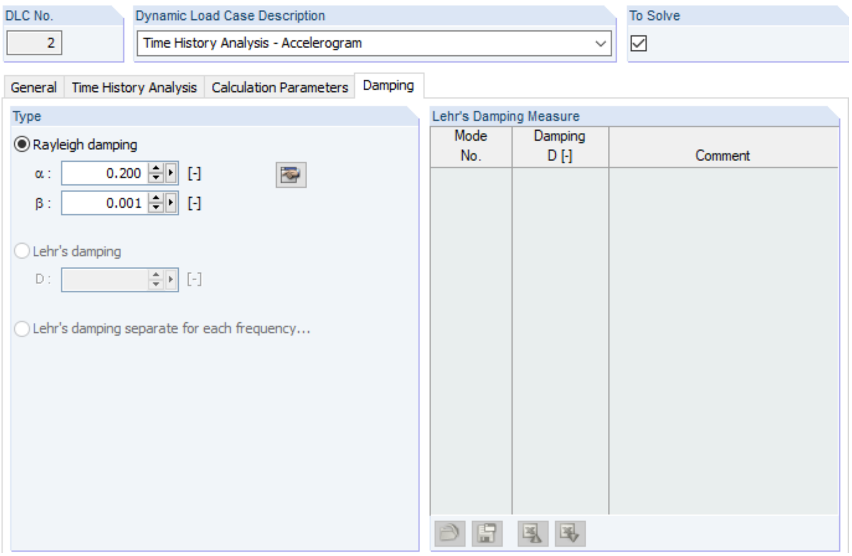



Figure 2.30: Module Window *Dynamic Load Cases* with *Damping* tab

With the *Rayleigh damping* coefficients, the damping matrix **C** is defined as

$$\mathbf{C} = \alpha\mathbf{M} + \beta\mathbf{K} \tag{2.20}$$

using the factors $\alpha$ and $\beta$. The Rayleigh damping coefficients are used for the linear and nonlinear *implicit Newmark* and the *explicit solver*. The damping matrix **C** does not need to be a diagonal matrix for the direct time history solvers. For more information about the Rayleigh damping, see for example [18].

The Lehr's damping values $D_i$ are used for the *modal analysis*. The damping matrix **C** needs to be a diagonal matrix. The Lehr's damping values are defined for each single mode *i* as a factor between the existing and the critical damping as follows:

$$D_i = \frac{c_i}{2 \cdot m_i \cdot \omega_i} \tag{2.21}$$

where $c_i$ are the entries in the diagonal damping matrix, $m_i$ the modal masses, and $\omega_i$ the angular frequencies of the system.

The Rayleigh coefficients and the Lehr's damping are connected as follows:

$$D_i = \frac{1}{2}\left(\frac{\alpha}{\omega_i} + \beta\,\omega_i\right) \tag{2.22}$$

This equation is visualised in Figure 2.31 using the Rayleigh coefficients $\alpha = 0.2$ and $\beta = 1 \times 10^{-3}$. It gets clear that for one set of Rayleigh coefficients different values of Lehr's damping result for each natural frequency of the structure.

RF-DYNAM Pro internally converts the Rayleigh damping coefficients $\alpha$ and $\beta$ into the Lehr's damping values $D_i$ if the *modal analysis* is selected. In this case, the solution is unique.

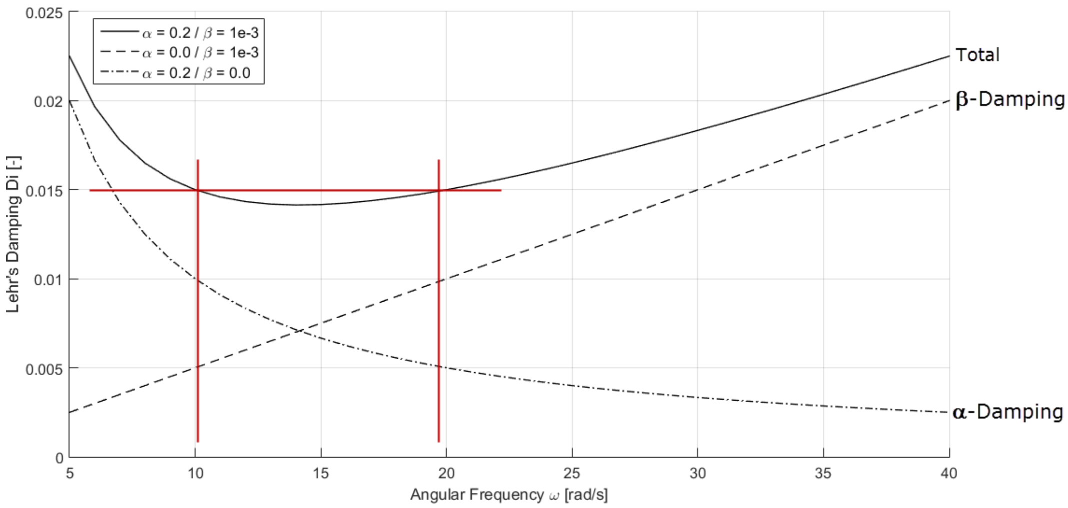



Figure 2.31: Relation between Rayleigh coefficients $\alpha$ and $\beta$ and the Lehr's damping *D* using the examples:
1 – Total: $\alpha = 0.2$ and $\beta = 1 \times 10^{-3}$
2 – $\beta$-Damping: $\alpha = 0.0$ and $\beta = 1 \times 10^{-3}$
3 – $\alpha$-Damping: $\alpha = 0.2$ and $\beta = 0.0$
Illustrated are the Lehr's damping values resulting for each angular frequency $\omega$ of the structure.

For the *implicit* and *explicit* solver, the Rayleigh coefficients have to be specified. You can convert Lehr's damping values $D_i$ to Rayleigh coefficients $\alpha$ and $\beta$ by clicking the button: in a new dialog box, define the most dominant frequencies of your model in combination with the required Lehr's damping values.

The conversion from Lehr's damping values to Rayleigh coefficients is illustrated in the example in Section 5.2.

## 2.11 Dynamic Load Cases - Equivalent Load Analysis

The equivalent load analysis is performed when the corresponding radio button in Figure 2.22 is selected; it belongs to the add-on module *RF-DYNAM Pro - Equivalent Loads*. The equivalent load analysis in *RF-DYNAM Pro* is based on the multi-modal response spectra analysis.

The main differences to the *Response Spectra Analysis* in the add-on module *RF-DYNAM Pro - Forced Vibrations* described in Section 2.9 are listed below:

- Load cases with equivalent loads are exported to RFEM separately for each mode and each excitation direction.
- The calculation of load cases is done in the main program RFEM. Thus, nonlinearities are considered by default, but stiffness modifications defined in the *NVCs* are not taken over.
- Accidental torsional actions can be considered automatically.
- Base shear forces can be easily evaluated separately for each mode.
- Result combinations are produced separately for each excitation direction (combined modal responses with *SRSS* or *CQC*) and for the combination of results from different excitation directions (*SRSS*, 100% / 30% (40%)).
- A signed result option is available that provides unique *RCs* using the signs of the dominant mode shape.
- The results are reproducible step by step.
- All supports are excited identically (no multi-point option).

The corresponding sub-tabs when performing an equivalent load analysis are *General*, *Equivalent Force Analysis*, and *Mode Shapes*. The *Equivalent Force Analysis* sub-tab is illustrated in Figure 2.32.

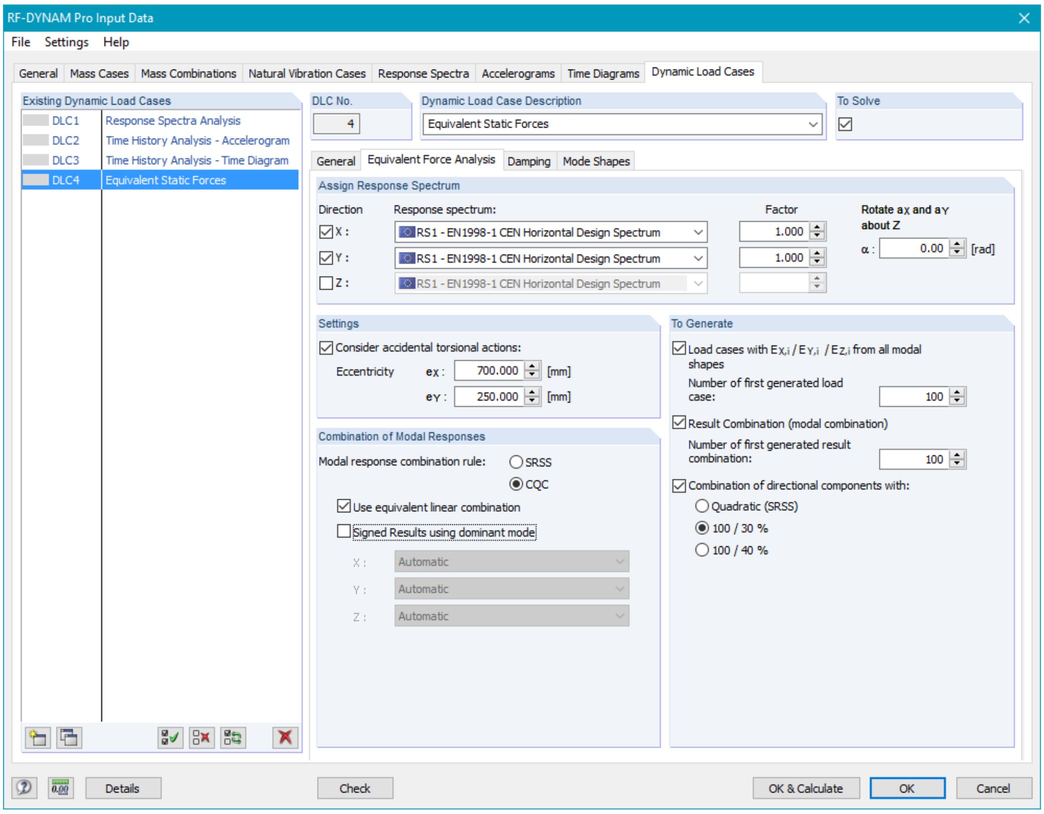
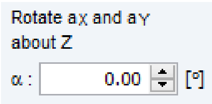



Figure 2.32: Module Window *Dynamic Load Cases* with *Equivalent Force Analysis* tab

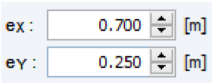
## 2.11.1 Assign Response Spectrum

In each direction, a specific *Response spectrum* can be assigned. At least one of the directions must be selected. The response spectrum (*RS*) can to be chosen from the drop-down menu. It is possible to adjust the multiplier *Factor* independently for each direction.




The excitation direction can be rotated in the XY-plane about the Z-axis. For example, when the response spectrum shall excite the structure 45 ° rotated about the Z-axis, activate only the X-direction with your response spectrum chosen from the drop-down menu and enter $\alpha = 45$ °.

## 2.11.2 Accidental Torsional Actions

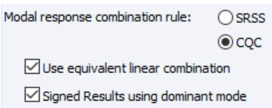



RF-DYNAM Pro considers accidental torsion actions automatically when you have selected the corresponding check box and the eccentricities $e_X$ and $e_Y$ have been defined. This is regulated in EN 1998-1 Sections 4.3.2 and 4.3.3.3.3 [1], for example. The eccentricities $e_X$ and $e_Y$ define how far the centre of mass is displaced, to account for uncertainties in the location of masses. For the calculation of the equivalent loads and the torsional moments, see Section 4.4.1 and Section 4.4.2.

When you export load cases with generated equivalent loads, two load cases will be generated for each mode shape and each direction – one with positive torsional moments, and one with negative torsional moments. RF-DYNAM Pro creates an alternative combination of those two load cases. More information on how modal combinations are applied when accidental torsional actions are activated are reported in the Knowledge Base - 001118.

## 2.11.3 Combination of Modal Responses

In the dialogue box section *Combination of Modal Responses*, you define how the responses resulting from different eigenvalues of the structure are to be combined. The modal combination is the first step of dynamic combination.




The modal response can be combined using the *Square Root of the Sum of the Squares* (*SRSS*) rule or the *Complete Quadratic Combination* (*CQC*) rule. Both of those quadratic combinations can be applied in the standard form or modified as equivalent linear combination. The option *Signed Results* provides modal combinations with unique signs using the dominant mode shape of the structure.

The standard form of *SRSS* combines maximum results; the signs are lost. The formula is provided below:

$$R_{SRSS} = \sqrt{R_1^2 + R_2^2 + \ldots + R_p^2} \tag{2.23}$$

The combined results $R_{SRSS}$ result from the modal responses $R_p$ from $p$ modes of the structure.

RF-DYNAM Pro includes a modified form of the *SRSS* rule in order to calculate corresponding results, *i.e.* corresponding internal forces. Compared to the standard form of the *SRSS* rule, those corresponding results are usually much smaller and the signs are correct in relation to the controlling force. The *SRSS* rule is applied as equivalent linear combination [11] as follows:

$$R_{SRSS} = \sum_{i=1}^{p} f_i \cdot R_i \qquad \text{where} \qquad f_i = \frac{R_i}{\sqrt{\sum\limits_{j=1}^{p} R_j^2}} \tag{2.24}$$

By this formula the result combinations are consistent in itself.

The *SRSS* rule is only allowed for systems where adjacent natural periods $T_i < T_j$ differ more than 10%, *i.e.* when the following statement is true:

$$\frac{T_i}{T_j} < 0.9 \tag{2.25}$$

In all other cases, the *CQC* rule must be applied. The *CQC* rule is defined as follows:

$$R_{CQC} = \sqrt{\sum_{i=1}^{p} \sum_{j=1}^{p} R_i \varepsilon_{ij} R_j} \tag{2.26}$$

where the correlation coefficient $\varepsilon$ is

$$\varepsilon_{ij} = \frac{8 \cdot \sqrt{D_i D_j}\left(D_i + rD_j\right) r^{3/2}}{(1-r^2)^2 + 4D_i D_j r\,(1+r^2) + 4\left(D_i^2 + D_j^2\right) r^2} \qquad \text{where} \quad r = \frac{\omega_j}{\omega_i} \tag{2.27}$$

If the viscose damping value $D$ is equal for all modes, the correlation coefficient $\varepsilon$ simplifies to:

$$\varepsilon_{ij} = \frac{8 \cdot D^2(1+r)r^{3/2}}{(1-r^2)^2 + 4D^2 r(1+r)^2} \tag{2.28}$$

The damping value $D_i$ that is required for the calculation of the correlation coefficient $\varepsilon_{ij}$ is defined in the *Damping* tab. It appears as soon as you have selected the *CQC* rule. The settings for damping are described below.

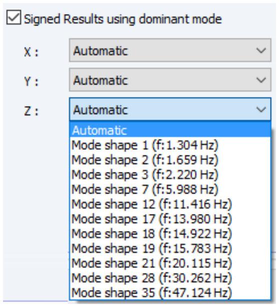
Analogously to the *SRSS* rule, the *CQC* rule can be applied as equivalent linear combination [11]. The *CQC* rule, still employing $\varepsilon_{ij}$ as defined in Equation 2.27, is modified as follows:

$$R_{CQC} = \sum_{i=1}^{p} f_i \cdot R_i \qquad \text{where} \quad f_i = \frac{\sum_{j=1}^{p} \varepsilon_{ij} R_j}{\sqrt{\sum_{i=1}^{p} \sum_{j=1}^{p} R_i \varepsilon_{ij} R_j}} \tag{2.29}$$

You can find more information on the option *Use equivalent linear combination* and an example in the Knowledge Base - 001098.

When the *CQC* rule is applied, the exported load cases contain information on the corresponding angular frequency $\omega$ and the applied damping $D_i$.

You can find more information on the *CQC* rule and an example in the Knowledge Base - 001263. Independently of RF-DYNAM Pro, the *CQC* rule is available in the main program RFEM. This is described in the Knowledge Base - 001259.

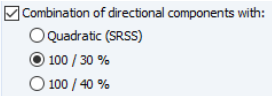



With the usual quadratic combinations like the *SRSS* and *CQC* rule, all maxima are positive and all minima are negative in the produced result envelope. The signs of results are lost and the results combinations are not unique. To overcome this side effect of quadratic combinations and to produce a more realistic representation of internal forces, the so-called *Signed Results* option is available in RF-DYNAM Pro. When the structure has a dominant eigenvalue, the signs of the corresponding load case can be applied to the results in the result combination. This leads to unique result combinations (*RC*). When you choose *Automatic* in the drop-down list, RF-DYNAM Pro selects the dominant mode shape automatically by means of the largest effective modal mass factor. You can also select any other eigenvalue manually.

You can find more information on the *Signed Results* option and an example in the Knowledge Base - 001278.

Further information and mathematical derivations of the combination rules are provided in e.g. [12–14].

## 2.11.4 Combination of Directional Components




The internal forces resulting from different excitation directions can be combined quadratically with the *SRSS* rule, or by using the 100% / 30% (40%) rule. The *SRSS* rule is applied as defined in Equation 2.23, but now $i = 1..p$ are the excitation directions X, Y, and Z. The *SRSS* rule for the directional combination can also be realised as equivalent linear combination applying Equation 2.24.

## 2.11.5 Export of Load Cases and Result Combinations

When you select the *Load cases* check box, load cases are exported to RFEM separately for each mode and separately for each excitation direction. The equivalent loads are calculated in according to Equation 4.9, Equation 4.10, and Equation 4.11.

When you select *Result combination (modal combination)* and *Combination of directional components*, several result combinations are automatically generated.

- *RC*s containing the modal response combination (*SRSS* or *CQC*) separately for each direction.
- *RC*s combining the results in the different excitation directions. One *RC* is generated for the *SRSS* rule, but a maximum of three *RC*s are generated for the 100% / U[30 percent] (40%) rule.

More information on the exported load cases and result combinations is provided in Section 4.4.

## 2.11.6 Damping for CQC Rule

The *Damping* tab is only available when the *CQC* rule has been selected for the combination of mode shapes (defined in Equation 2.26 and Equation 2.27).

For the CQC rule, the Lehr's damping values $D_i$ are needed. Those can be defined equally or differently for each mode of the system. The Rayleigh damping with the coefficients $\alpha$ and $\beta$ is also available, but those values are internally converted to the Lehr's damping. The *Damping* tab is the same as for the *Response Spectrum Analysis*. It is shown in Figure 2.24 in Section 2.9.6.

For the conversion of the Rayleigh coefficients $\alpha$ and $\beta$ to the Lehr's damping $D_i$, see Section 2.10.9.

## 2.11.7 Mode Shape Selection

RF-DYNAM Pro - Equivalent Loads provides a multi-modal response spectra analysis. You can select as many modes as important for the analysis. The *Mode Shape* tab and its settings are discussed in detail in Section 2.9.7 and are the same for the equivalent load analysis.

## 2.12 Global *Details* Settings


The *Details* dialogue box can be accessed by pressing the [Details] button. You can also open it via **Settings** → **Details**. In this dialogue box, you can set global parameters that are valid for the overall dynamic calculation and independent of any defined case within the module.

The *Details* dialogue box is shown in Figure 2.33.

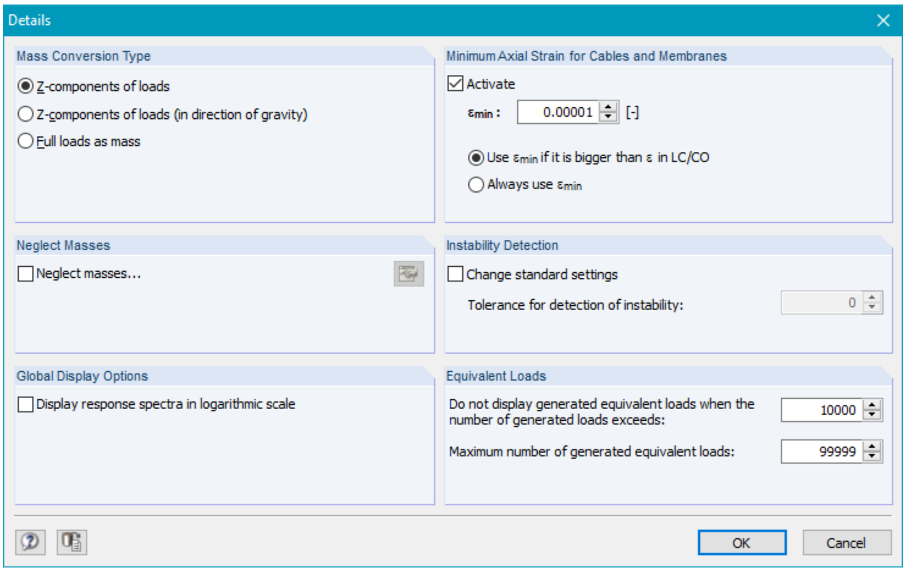



Figure 2.33: *Details* dialogue box with global parameters concerning RF-DYNAM Pro

### Mass Conversion Type

In Section 2.2, the mass import from load cases is explained. Section 2.2.4 provides details on the different types of mass conversions.

### Neglect Masses

Masses on nodes, lines, members, and surfaces can be neglected in the dynamic analysis. In Section 2.2.4, details on the *Neglect Masses* feature are provided.

### Display Response Spectra

You can change the display of the response spectra as seen in Figure 2.11 and Figure 2.25. The default display uses a linear x-axis, but it is possible to change it to a logarithmic scale.

### Minimum Axial Strain for Cables and Membranes

These special types of members and surfaces require a minimum axial strain. If the limit is set too small, the eigenvalues achieved are not realistic and only local mode shapes are determined. The default value is applicable for most cases. You can find more details on cables in the RFEM manual, Section 4.17.

### Instability Detection

The stability of a system can be analysed. For it, the default value in RF-DYNAM Pro is set to 0.01, which means a very sensitive detection of instability and an early break-off limit. Please see RFEM manual, Section 7.3.3 for further details.

### Equivalent Loads

As discussed in Section 2.11, load cases are exported containing the generated equivalent loads. Loads can only be viewed in the graphic when less than 5,000 loads are produced. You can change this value here, but note that the process of displaying more than 5,000 loads is slow.

The maximum number of loads that RFEM can produce and export to load cases is 99,999. As equivalent loads are generated on each FE-node, you can only get a full set of equivalent loads when your structure has less than 99,999 FE-nodes. When your structure has more FE-nodes, the smallest equivalent loads are neglected, and only 99,999 loads will be exported. You can adjust the maximum number of generated equivalent loads in the *Details* dialogue box.

## 2.13 Units and Decimal Places


You can access the *Units and Decimal Places* dialogue box with the button shown on the left. The dialogue box is shown in Figure 2.34.

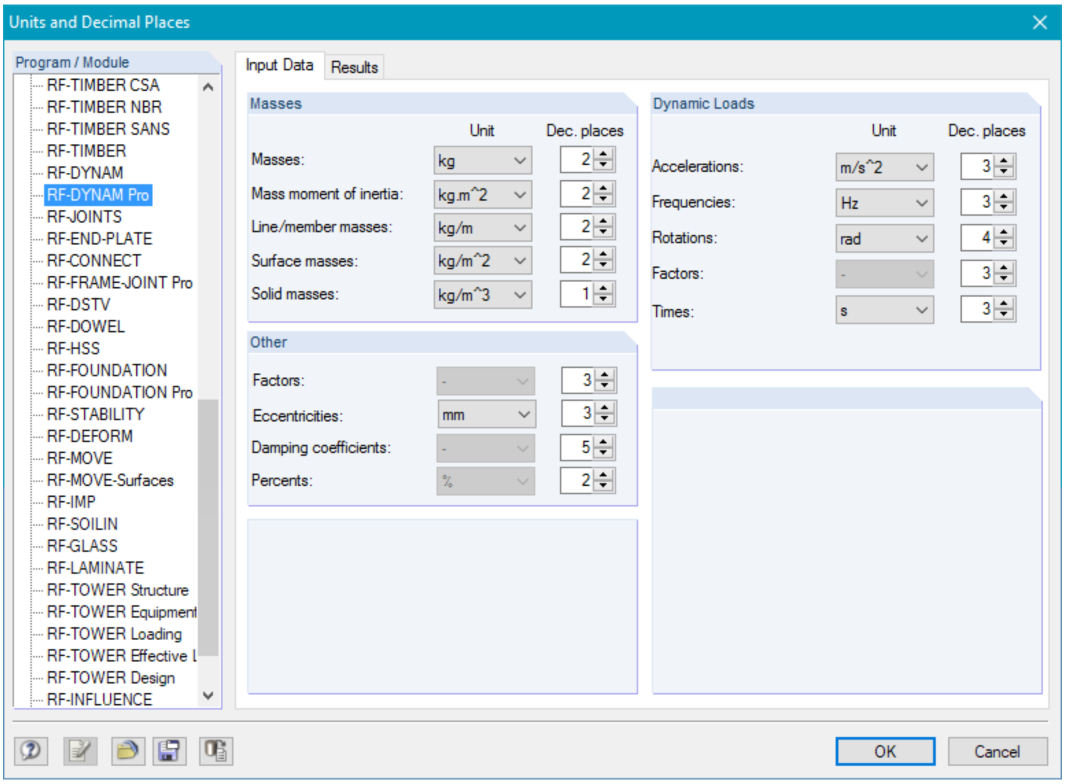



Figure 2.34: Dialogue box *Units and Decimal Places* showing settings of RF-DYNAM Pro *Input Data*

Select the add-on module *RF-DYNAM Pro* in the list of modules. The *Input Data* tab is opened. There you can choose the units from the drop-down menus and adjust the decimal places.

# 3 Calculation

## 3.1 Check


Before starting the calculation, you can check the input data without closing the add-on module. Click [Check] in the bottom part of the module.

The module shows the *Input Data Verification* dialogue box displaying any warning and error messages. When no errors have been found, the message *No consistency errors found* is displayed.

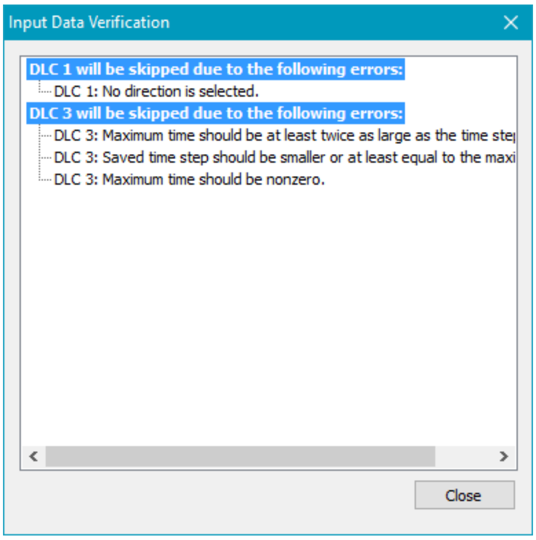
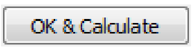


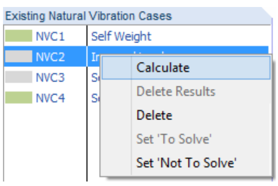

Figure 3.1: Dialogue box *Input Data Verification* showing messages

When you click [OK & Calculate], the check is also performed. If an error is detected, the *Input Data Verification* dialogue box is shown and you can decide whether you want to perform or cancel the calculation.

## 3.2 Start Calculation




To start the calculation, click the [OK & Calculate] button. RF-DYNAM Pro then calculates all cases (*NVC*s and *DLC*s) where no results are available and which have been selected to be solved (*To Solve* check box). The add-on module closes once the calculation has finished.




To start the calculation of a singlec natural vibration case (*NVC*) or dynamic load case (*DLC*), you can right-click the specific case and select *Calculate*. Only the selected case will be calculated by RF-DYNAM Pro. When the calculation has finished, the module stays open. The colour of the selected case changes from grey to green. When you calculate a *DLC*, the assigned *NVC* will be calculated as well.

You can also start the calculation in the RFEM user interface. To calculate the RF-DYNAM Pro case directly, select it from the list in the RFEM toolbar shown in Figure 3.2. Click the [Show Results] button to perform the calculation. All cases (*NVC*s and *DLC*s) will be calculated where no results are available and which have been selected to be solved (*To Solve* check box).


Figure 3.2: Direct calculation of *RF-DYNAM Pro* case in RFEM

The *To Calculate* dialogue box (**Calculate** → **To Calculate**) lists the add-on module cases as well as load cases and load combinations. This is shown in Figure 3.3.

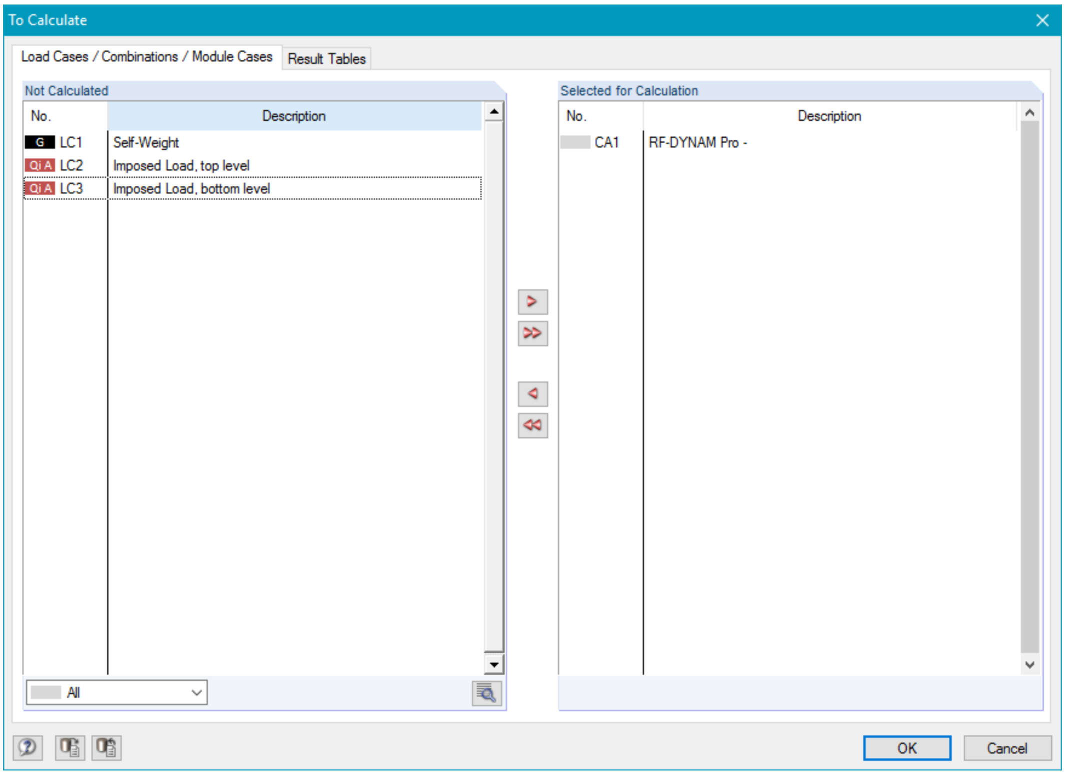
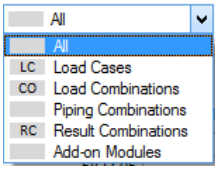



Figure 3.3: Dialogue box *To Calculate*




You can filter the available cases with the drop-down menu shown on the left.

To transfer the selected *RF-DYNAM Pro* case to the list on the right, use the or button. Click [OK] to start the calculation. All cases (*NVCs* and *DLCs*) will be calculated where no results are available and which have been selected to be solved (*To Solve* check box).

When you have started the calculation, you can observe the analysis process in a separate dialogue box shown in Figure 3.4.

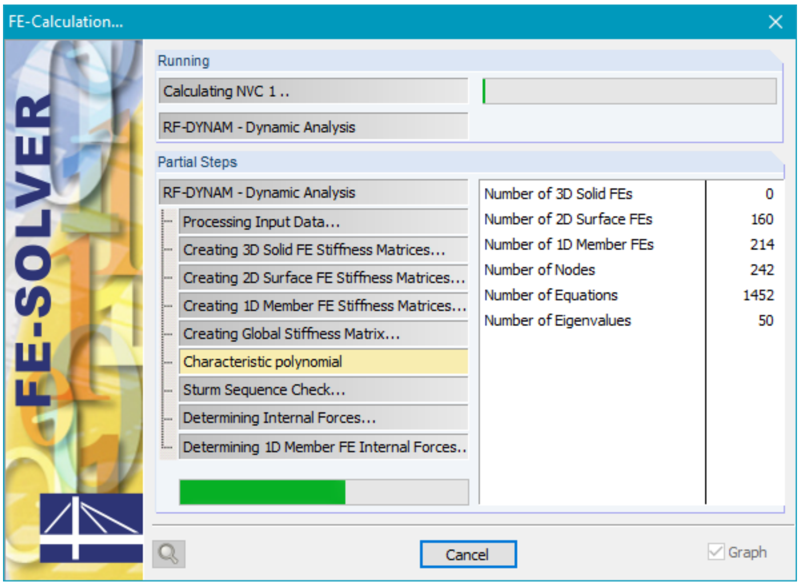
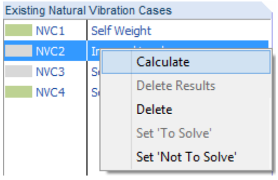



Figure 3.4: Process of RF-DYNAM Pro calculation

## 3.3 Availability of Results




The process to delete and calculate cases in RF-DYNAM Pro is differentiated: Cases like natural vibration cases (*NVCs*) and dynamic load cases (*DLCs*) can be calculated separately, and also the results can be deleted separately by using the shortcut menu (right-click the specific case). The shortcut menu also offers to delete the case, set the case *To Solve* or *Not to solve*.

The colour of a case is grey when no results are available. The color of a case is green when results are available; then also the exported *LC*s and *RC*s do exist. Exported *RC*s and *LC*s are tied to the results, *i.e.* they will be deleted as soon as the results of a case are deleted.

The results of a specific case will be deleted, when

- you use the shortcut menu to delete the results,
- something is changed in the *NVC* or *DLC* within RF-DYNAM Pro,
- assigned mass cases (*MC*), mass combinations (*MCO*), natural vibration cases (*NVC*), response spectra (*RS*), accelerograms (*AC*) or time diagrams (*TD*) change,
- assigned load cases *LC* or load combinations *CO* change.

Results of a specific case will not be deleted, when

- the description of the case is modified,
- comments are made in the *General* tab,
- the scaling of mode shapes is altered in a *NVC*,
- the export settings are changed in a *DLC* (button [OK & Calculate] updates only the exported *LC*s and *RC*s).

# 4 Results

The results of RF-DYNAM Pro are embedded in the main program RFEM. The general interpretation of the results is described in Sections 8 and 9 of the RFEM manual.

Figure 4.1 shows the main program RFEM. The results of a *Natural Vibration Case* are displayed, the first mode shape is illustrated, and the natural frequencies are listed in the table.

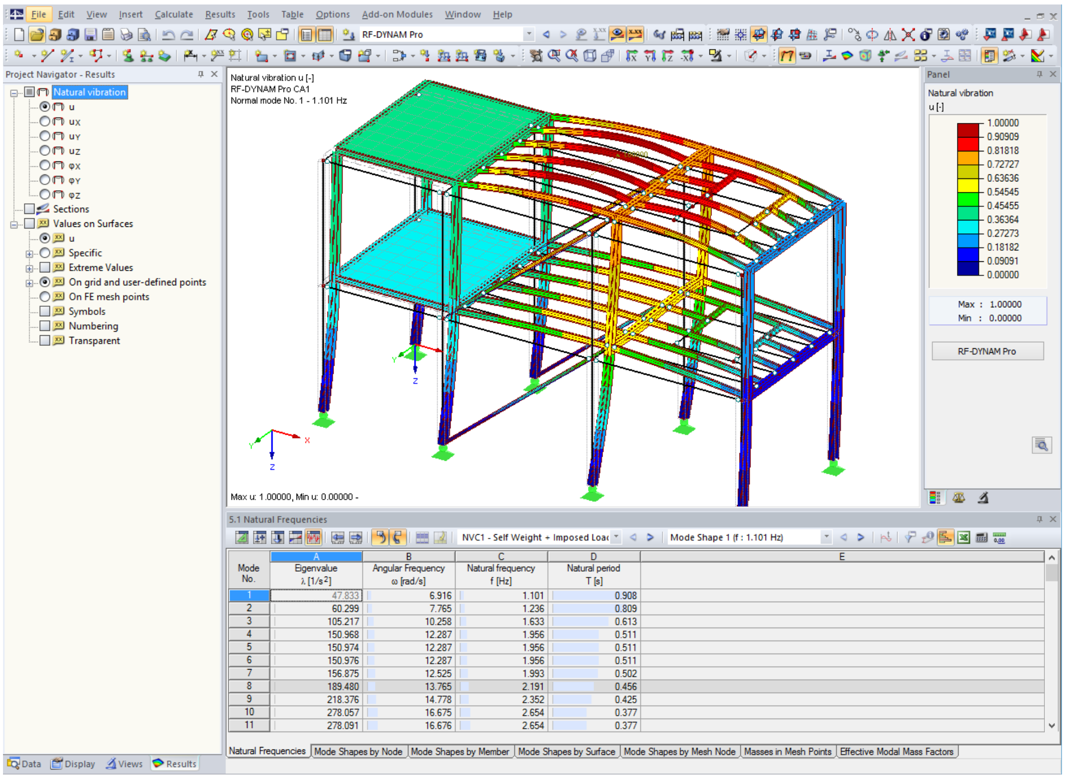


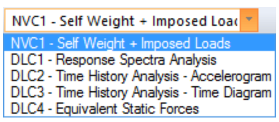

Figure 4.1: Results of a *Natural Vibration Case* in main program RFEM

The *Display* navigator and the *Results* navigator are explained in Section 3.4.3 of the RFEM manual, the *Tables* in Section 3.4.4 and the *Control Panel* in Section 3.4.6. In this RF-DYNAM Pro manual, only the newly available results will be discussed.

You can access all relevant results tables of your dynamic analysis with the [Dynamic Analysis] button.

Each of the results tables discussed in this chapter can be exported to MS Excel. The values within the tables are saved as strings, so only the number of decimal places that are displayed will be exported. If you need a higher accuracy, you have to adjust the decimal places in the *Unit and Decimal Places* dialogue box accessible with the [0.00] button.

All dynamic results tables can also be exported via **File** → **Export**. You can choose to export all dynamic tables or only selected tables as shown in Figure 4.2. The results that are available for each dynamic case are detailed in the following sections. Analogously you can export the data to the *CSV* file format.




With the drop-down menu, you can switch between the available natural vibration cases (*NVC*) and dynamic load cases (*DLC*). You can also switch between the cases using the [◀] and [▶] buttons. The available results tables depend on that selection. This is clarified in the following sections.

The graphic in the work area of RFEM is also updated according to the selected *NVC* or *DLC*. All results which are usually available in RFEM are accessible in the same way for the RF-DYNAM Pro results. The interaction between tables and graphic works as common for RFEM.

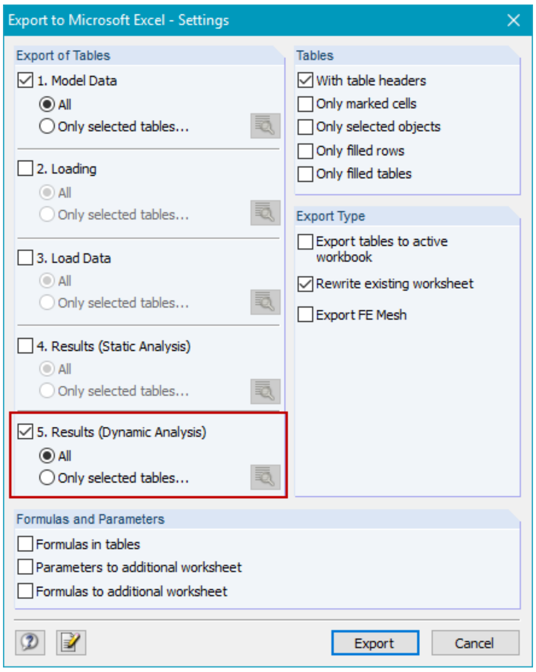
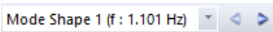



Figure 4.2: Dialogue box *Export to Microsoft Excel - Settings* to export all dynamic tables (*NVC*s and *DLC*s)

## 4.1 Natural Vibration Cases

The results tables that belong to natural vibration cases are available when a *NVC* case has been selected in the drop-down menu. The corresponding table numbers are **5.1 through 5.7**. Those results tables are always available; they belong to the module *RF-DYNAM Pro - Natural Vibrations*. The input data required for natural vibration cases (*NVC*) is discussed in Section 2.4.

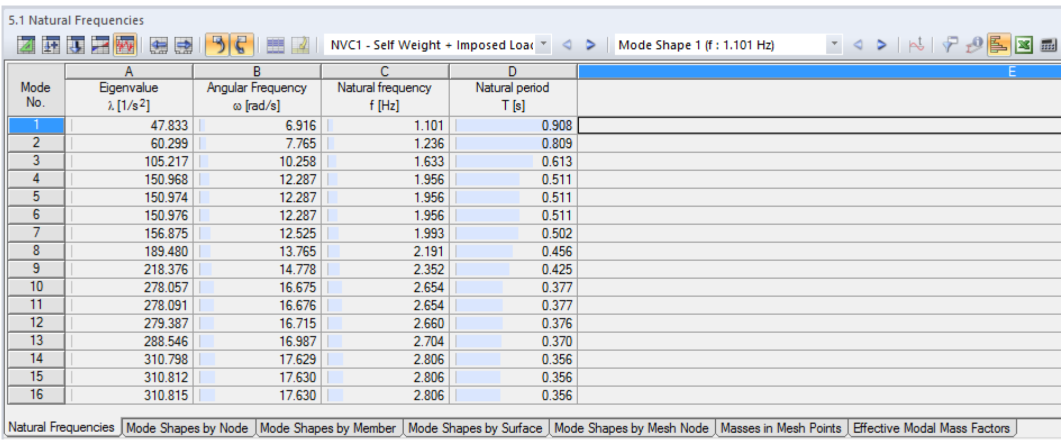



You can switch between the available mode shapes in the work window by using the drop-down menu shown on the left or the and buttons. The maximum deformation displayed in the graphics depends on the scaling option; this is discussed in Section 2.4.2.

The animation of the mode shapes can be activated via the button in the main toolbar (see Figure 4.1).

### 4.1.1 Natural Frequencies

Table 5.1 provides the natural frequencies of the undamped system. The results table is shown in Figure 4.3.




| Mode No. | A Eigenvalue λ [1/s²] | B Angular Frequency ω [rad/s] | C Natural frequency f [Hz] | D Natural period T [s] |
|---|---|---|---|---|
| 1 | 47.833 | 6.916 | 1.101 | 0.908 |
| 2 | 60.299 | 7.765 | 1.236 | 0.809 |
| 3 | 105.217 | 10.258 | 1.633 | 0.613 |
| 4 | 150.968 | 12.287 | 1.956 | 0.511 |
| 5 | 150.974 | 12.287 | 1.956 | 0.511 |
| 6 | 150.976 | 12.287 | 1.956 | 0.511 |
| 7 | 156.875 | 12.525 | 1.993 | 0.502 |
| 8 | 189.480 | 13.765 | 2.191 | 0.456 |
| 9 | 218.376 | 14.778 | 2.352 | 0.425 |
| 10 | 278.057 | 16.675 | 2.654 | 0.377 |
| 11 | 278.091 | 16.676 | 2.654 | 0.377 |
| 12 | 279.387 | 16.715 | 2.660 | 0.376 |
| 13 | 288.546 | 16.987 | 2.704 | 0.370 |
| 14 | 310.798 | 17.629 | 2.806 | 0.356 |
| 15 | 310.812 | 17.630 | 2.806 | 0.356 |
| 16 | 310.815 | 17.630 | 2.806 | 0.356 |

Figure 4.3: Table *5.1 Natural Frequencies* with eigenvalues $\lambda$ [$1/s^2$], angular frequencies $\omega$ [rad/s], natural frequencies f [Hz], and natural period T [s]

The equation of motion of a multi-degree of freedom without damping is solved with the four available eigenvalue solvers discussed in Section 2.4.5. Further theoretical details can be found e.g. in BATHE [6] or TEDESCO [9]. The equation of motion is defined as

$$\mathbf{M\ddot{u}} + \mathbf{Ku} = \mathbf{0} \tag{4.1}$$

where **M** is the mass matrix discussed in Section 2.4.4, **K** the stiffness matrix. **u** are the mode shapes containing translational and rotational parts:

$$\mathbf{u} = (u_X, u_Y, u_Z, \varphi_X, \varphi_Y, \varphi_Z)^T \tag{4.2}$$

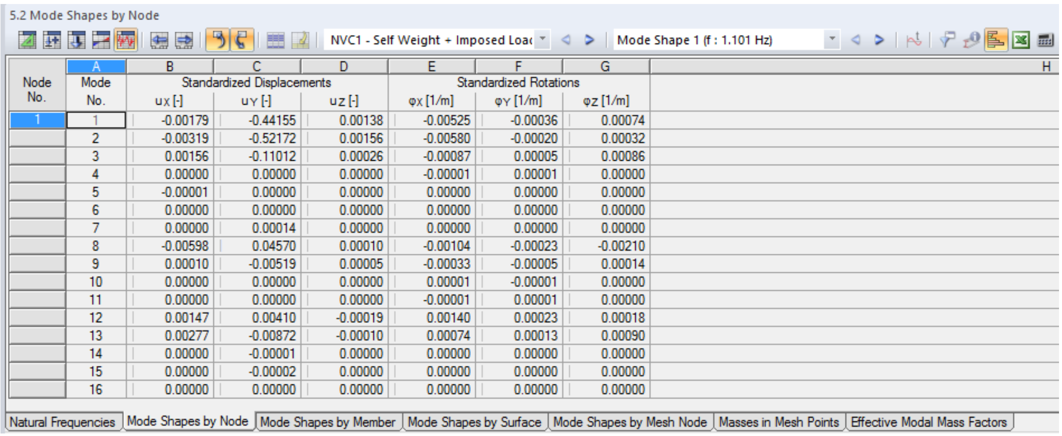
The eigenvalue $\lambda$ $[1/s^2]$ is connected to the angular frequency $\omega$ $[1/s]$ with $\lambda_i = \omega_i^2$. The natural frequency $f$ [Hz] is then derived with $f = \omega/2\pi$, and the natural period $T$ [s] is the reciprocal of the frequency obtained with $T = 1/f$.

For a multi-degree of freedom (MDOF) system, several eigenvalues $\lambda_i$ and corresponding mode shapes $\mathbf{u_i}$ exist for each mode $i$.

## 4.1.2 Mode Shapes


Each frequency of the system has a corresponding mode shape. These mode shapes are illustrated graphically in the work area of RFEM. You can use the drop-down menu to switch between the mode shapes; this was discussed above.

All mode shapes are tabulated in Tables 5.2, 5.3, 5.4 and 5.5. The difference in those tables are the way how the values are sorted. In Table 5.2, the standardised displacements $u_X$, $u_Y$, and $u_Z$ and rotations $\varphi_X$, $\varphi_Y$, and $\varphi_Z$ are sorted by nodes. This table is illustrated in Figure 4.4.


5.2 Mode Shapes by Node

NVC1 - Self Weight + Imposed Loac | Mode Shape 1 (f : 1.101 Hz)

| Node No. | Mode No. | Standardized Displacements uX [-] | uY [-] | uZ [-] | Standardized Rotations φX [1/m] | φY [1/m] | φZ [1/m] |
|---|---|---|---|---|---|---|---|
| 1 | 1 | -0.00179 | -0.44155 | 0.00138 | -0.00525 | -0.00036 | 0.00074 |
| | 2 | -0.00319 | -0.52172 | 0.00156 | -0.00580 | -0.00020 | 0.00032 |
| | 3 | 0.00156 | -0.11012 | 0.00026 | -0.00087 | 0.00005 | 0.00086 |
| | 4 | 0.00000 | 0.00000 | 0.00000 | -0.00001 | 0.00001 | 0.00000 |
| | 5 | -0.00001 | 0.00000 | 0.00000 | 0.00000 | 0.00000 | 0.00000 |
| | 6 | 0.00000 | 0.00000 | 0.00000 | 0.00000 | 0.00000 | 0.00000 |
| | 7 | 0.00000 | 0.00014 | 0.00000 | 0.00000 | 0.00000 | 0.00000 |
| | 8 | -0.00598 | 0.04570 | 0.00010 | -0.00104 | -0.00023 | -0.00210 |
| | 9 | 0.00010 | -0.00519 | 0.00005 | -0.00033 | -0.00005 | 0.00014 |
| | 10 | 0.00000 | 0.00000 | 0.00000 | 0.00001 | -0.00001 | 0.00000 |
| | 11 | 0.00000 | 0.00000 | 0.00000 | -0.00001 | 0.00001 | 0.00000 |
| | 12 | 0.00147 | 0.00410 | -0.00019 | 0.00140 | 0.00023 | 0.00018 |
| | 13 | 0.00277 | -0.00872 | -0.00010 | 0.00074 | 0.00013 | 0.00090 |
| | 14 | 0.00000 | -0.00001 | 0.00000 | 0.00000 | 0.00000 | 0.00000 |
| | 15 | 0.00000 | -0.00002 | 0.00000 | 0.00000 | 0.00000 | 0.00000 |
| | 16 | 0.00000 | 0.00000 | 0.00000 | 0.00000 | 0.00000 | 0.00000 |

Natural Frequencies | Mode Shapes by Node | Mode Shapes by Member | Mode Shapes by Surface | Mode Shapes by Mesh Node | Masses in Mesh Points | Effective Modal Mass Factors

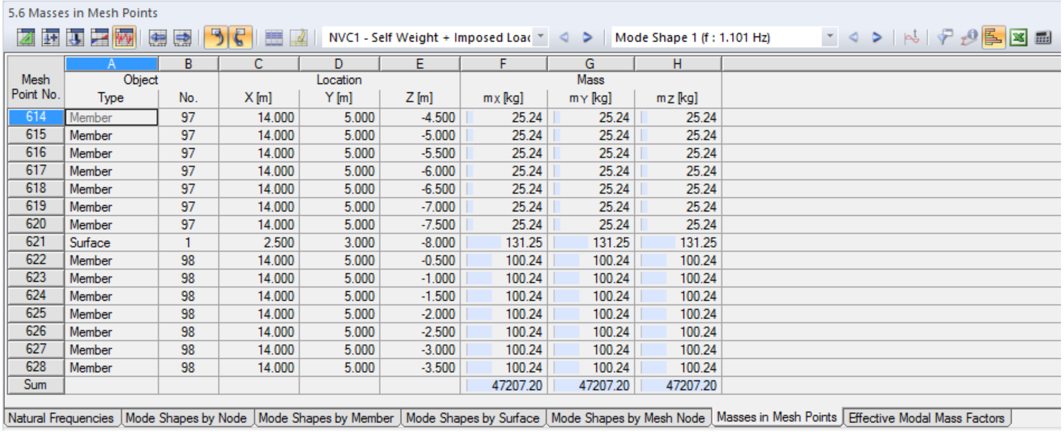
Figure 4.4: Table *5.2 Mode Shapes by Node* with standardised displacements $u_X$, $u_Y$, and $u_Z$ and rotations $\varphi_X$, $\varphi_Y$, and $\varphi_Z$

Table 5.3 lists the values sorted by members, Table 5.4 sorted by surfaces, and Table 5.5 sorted by mesh points. If there are no surfaces in the model, Table 5.4 does not exist.

The maximum values of $u$ and $\varphi$ are dependent on the selected scaling option discussed in Section 2.4.2. The standardised displacements $u$ are dimensionless, and the standardized rotations $\varphi$ are given in $[1/m]$. Those units result from the scaling procedure.

### 4.1.3 Masses in Mesh Points

The calculation of the masses in mesh points depends on the imported masses (see Section 2.2), on the settings in the *Details* dialogue box (see Section 2.12), and on the finally assigned mass case (*MC*) or mass combination *(MCO* (see Section 2.4.3). In Table 5.6, the masses in the translational directions $m_X$, $m_Y$, and $m_Z$ are provided. The sums of the masses are given at the bottom of the table. Table 5.6 is shown in Figure 4.5. The coordinates of the mesh points in the global coordinate system are listed as well.



5.6 Masses in Mesh Points

NVC1 - Self Weight + Imposed Loa | Mode Shape 1 (f : 1.101 Hz)

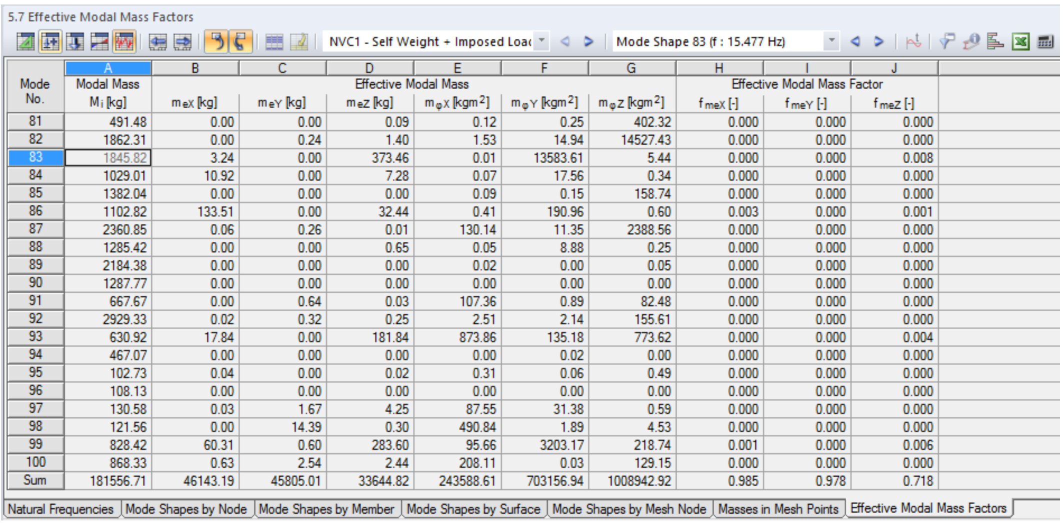
| | A | B | C | D | E | F | G | H |
|---|---|---|---|---|---|---|---|---|
| Mesh | Object | | Location | | | Mass | | |
| Point No. | Type | No. | X [m] | Y [m] | Z [m] | mX [kg] | mY [kg] | mZ [kg] |
| 614 | Member | 97 | 14.000 | 5.000 | -4.500 | 25.24 | 25.24 | 25.24 |
| 615 | Member | 97 | 14.000 | 5.000 | -5.000 | 25.24 | 25.24 | 25.24 |
| 616 | Member | 97 | 14.000 | 5.000 | -5.500 | 25.24 | 25.24 | 25.24 |
| 617 | Member | 97 | 14.000 | 5.000 | -6.000 | 25.24 | 25.24 | 25.24 |
| 618 | Member | 97 | 14.000 | 5.000 | -6.500 | 25.24 | 25.24 | 25.24 |
| 619 | Member | 97 | 14.000 | 5.000 | -7.000 | 25.24 | 25.24 | 25.24 |
| 620 | Member | 97 | 14.000 | 5.000 | -7.500 | 25.24 | 25.24 | 25.24 |
| 621 | Surface | 1 | 2.500 | 3.000 | -8.000 | 131.25 | 131.25 | 131.25 |
| 622 | Member | 98 | 14.000 | 5.000 | -0.500 | 100.24 | 100.24 | 100.24 |
| 623 | Member | 98 | 14.000 | 5.000 | -1.000 | 100.24 | 100.24 | 100.24 |
| 624 | Member | 98 | 14.000 | 5.000 | -1.500 | 100.24 | 100.24 | 100.24 |
| 625 | Member | 98 | 14.000 | 5.000 | -2.000 | 100.24 | 100.24 | 100.24 |
| 626 | Member | 98 | 14.000 | 5.000 | -2.500 | 100.24 | 100.24 | 100.24 |
| 627 | Member | 98 | 14.000 | 5.000 | -3.000 | 100.24 | 100.24 | 100.24 |
| 628 | Member | 98 | 14.000 | 5.000 | -3.500 | 100.24 | 100.24 | 100.24 |
| Sum | | | | | | 47207.20 | 47207.20 | 47207.20 |

Natural Frequencies | Mode Shapes by Node | Mode Shapes by Member | Mode Shapes by Surface | Mode Shapes by Mesh Node | Masses in Mesh Points | Effective Modal Mass Factors

Figure 4.5: Table *5.6 Masses in Mesh Points* with translational directions $m_X$, $m_Y$, and $m_Z$ for each mesh point and sums of masses

### 4.1.4 Effective Modal Mass Factors

Table 5.7 provides the modal masses $\mathbf{M_i}$, the effective modal masses $m_e$, and the effective modal mass factors $f_{me}$. It is illustrated in Figure 4.6. The effective modal masses describe how much mass is activated by each eigenvalue of the system in each direction.



5.7 Effective Modal Mass Factors

NVC1 - Self Weight + Imposed Loa | Mode Shape 83 (f : 15.477 Hz)

| | A | B | C | D | E | F | G | H | I | J |
|---|---|---|---|---|---|---|---|---|---|---|
| Mode | Modal Mass | Effective Modal Mass | | | | | | Effective Modal Mass Factor | | |
| No. | Mi [kg] | meX [kg] | meY [kg] | meZ [kg] | mφX [kgm²] | mφY [kgm²] | mφZ [kgm²] | fmeX [-] | fmeY [-] | fmeZ [-] |
| 81 | 491.48 | 0.00 | 0.00 | 0.09 | 0.12 | 0.25 | 402.32 | 0.000 | 0.000 | 0.000 |
| 82 | 1862.31 | 0.00 | 0.24 | 1.40 | 1.53 | 14.94 | 14527.43 | 0.000 | 0.000 | 0.000 |
| 83 | 1845.82 | 3.24 | 0.00 | 373.46 | 0.01 | 13583.61 | 5.44 | 0.000 | 0.000 | 0.008 |
| 84 | 1029.01 | 10.92 | 0.00 | 7.28 | 0.07 | 17.56 | 0.34 | 0.000 | 0.000 | 0.000 |
| 85 | 1382.04 | 0.00 | 0.00 | 0.00 | 0.09 | 0.15 | 158.74 | 0.000 | 0.000 | 0.000 |
| 86 | 1102.82 | 133.51 | 0.00 | 32.44 | 0.41 | 190.96 | 0.60 | 0.003 | 0.000 | 0.001 |
| 87 | 2360.85 | 0.06 | 0.26 | 0.01 | 130.14 | 11.35 | 2388.56 | 0.000 | 0.000 | 0.000 |
| 88 | 1285.42 | 0.00 | 0.00 | 0.65 | 0.05 | 8.88 | 0.25 | 0.000 | 0.000 | 0.000 |
| 89 | 2184.38 | 0.00 | 0.00 | 0.00 | 0.02 | 0.00 | 0.05 | 0.000 | 0.000 | 0.000 |
| 90 | 1287.77 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.000 | 0.000 | 0.000 |
| 91 | 667.67 | 0.00 | 0.64 | 0.03 | 107.36 | 0.89 | 82.48 | 0.000 | 0.000 | 0.000 |
| 92 | 2929.33 | 0.02 | 0.32 | 0.25 | 2.51 | 2.14 | 155.61 | 0.000 | 0.000 | 0.000 |
| 93 | 630.92 | 17.84 | 0.00 | 181.84 | 873.86 | 135.18 | 773.62 | 0.000 | 0.000 | 0.004 |
| 94 | 467.07 | 0.00 | 0.00 | 0.00 | 0.00 | 0.02 | 0.00 | 0.000 | 0.000 | 0.000 |
| 95 | 102.73 | 0.04 | 0.00 | 0.02 | 0.31 | 0.06 | 0.49 | 0.000 | 0.000 | 0.000 |
| 96 | 108.13 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.000 | 0.000 | 0.000 |
| 97 | 130.58 | 0.03 | 1.67 | 4.25 | 87.55 | 31.38 | 0.59 | 0.000 | 0.000 | 0.000 |
| 98 | 121.56 | 0.00 | 14.39 | 0.30 | 490.84 | 1.89 | 4.53 | 0.000 | 0.000 | 0.000 |
| 99 | 828.42 | 60.31 | 0.60 | 283.60 | 95.66 | 3203.17 | 218.74 | 0.001 | 0.000 | 0.006 |
| 100 | 868.33 | 0.63 | 2.54 | 2.44 | 208.11 | 0.03 | 129.15 | 0.000 | 0.000 | 0.000 |
| Sum | 181556.71 | 46143.19 | 45805.01 | 33644.82 | 243588.61 | 703156.94 | 1008942.92 | 0.985 | 0.978 | 0.718 |

Natural Frequencies | Mode Shapes by Node | Mode Shapes by Member | Mode Shapes by Surface | Mode Shapes by Mesh Node | Masses in Mesh Points | Effective Modal Mass Factors

Figure 4.6: Table *5.7 Effective Modal Mass Factors* with modal masses $M_i$, effective modal masses $m_e$, and effective modal mass factors $f_{me}$ related to global axes

The modal mass is defined with

$$M_i = \mathbf{u}_i^T \cdot \mathbf{M} \cdot \mathbf{u}_i \tag{4.3}$$

where $\mathbf{u_i}$ is the eigenvector of a single mode *i* as defined in Equation 4.2, and **M** is the mass matrix discussed in Section 2.4.4. The modal mass $\mathbf{M_i}$ is independent of direction. It changes depending on the scaling option chosen for the mode shapes (see Section 2.4.2); when the option $\mathbf{u}_i^T\mathbf{M}\mathbf{u}_i = 1$ kg is chosen, all modal masses are $M_i = 1$ kg.

The effective modal masses $m_{ij}^{eff}$ provide the masses that are accelerated in the j-direction, where $j = 1{,}2{,}3$ for translations and $j = 4{,}5{,}6$ for rotations, separately for each mode *i*. Those masses are independent of the scaling option for mode shapes, and directly related to the participation factors $\Gamma_{i,j}$.

$$\Gamma_{i,j} = \frac{1}{M_i}\mathbf{u}_i^T\mathbf{MT}_j \tag{4.4}$$

where $\mathbf{T}_j$ is the $j^{th}$ column in matrix $\mathbf{T}$

$$\mathbf{T} = \begin{bmatrix} 1 & 0 & 0 & 0 & (Z-Z_0) & -(Y-Y_0) \\ 0 & 1 & 0 & -(Z-Z_0) & 0 & (X-X_0) \\ 0 & 0 & 1 & (Y-Y_0) & -(X-X_0) & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \tag{4.5}$$

X, Y, and Z are the global coordinates of the considered FE-node provided in Table 5.6 (see Figure 4.5), $X_0$, $Y_0$ and $Z_0$ are the centres of total mass provided in the **Mass Cases** → **General** tab in RF-DYNAM Pro (see Section 2.2). This matrix **T** exists for each FE-node. The definition of the participation factors defining also the rotational degrees of freedom is detailed in [19], Section 15.7.5. The participation factor $\Gamma_{ij}$ is dimensionless for translations and has the unit [m] for rotations.

The effective modal masses are defined with

$$m_{ij}^{eff} = M_i \cdot \Gamma_{ij}^2 \tag{4.6}$$

where the effective masses for translations $m_{eX}$, $m_{eY}$, and $m_{eZ}$ are provided in [kg] and for rotations $m_{\varphi X}$, $m_{\varphi Y}$, and $m_{\varphi Z}$ in [kgm$^2$].

The sums of the effective modal masses $\sum m_e$ are provided in Table 5.7 (see Figure 4.6). In the translational directions, those sums are equal to the total sum of the structure $M_{total}$ (see Table 5.6 in Figure 4.5) beside masses that are not activated, *i.e.* masses in fixed supports, and assuming that all eigenvalues of the system are calculated.

The effective modal mass factors $f_{me}$ are required to decide whether or not a specific mode must be taken into account for the response spectra analysis or the equivalent load analysis. **EN 1998-1** [1] states in **Section 4.3.3.3** that "the sum of the effective modal masses for the modes taken into account" needs to be "at least 90% of the total mass of the structure" and that "all modes with effective modal masses greater than 5% of the total mass" have to be taken into account .

The effective modal mass factors $f_{me}$ are defined as follows:

$$f_{me} = \frac{m_e}{\sum m_e} \tag{4.7}$$

Further details on modal analysis can be found in the literature, e.g. Meskouris [20] or Tedesco [9].

### 4.1.5 Export to Excel or CSV

You can export all results tables that have been discussed in this section to MS Excel or save them in the *CSV* format using the **File** → **Export** option. The list of results tables is shown in Figure 4.7.

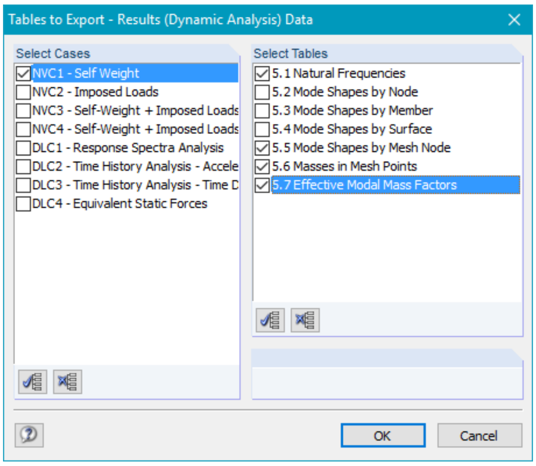
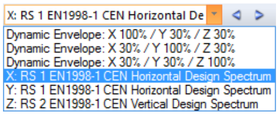



Figure 4.7: Results tables to export for natural vibration cases (*NVC*)

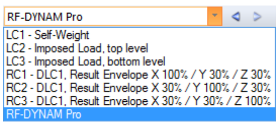
## 4.2 Dynamic Load Cases - Response Spectra Analysis

The results tables that belong to response spectra analysis cases are available when the corresponding dynamic load case (*DLC*) is chosen in the drop-down menu. Tables 5.11 to 5.16 belong to this type of dynamic load cases. The response spectrum analysis belongs to the module *RF-DYNAM Pro - Forced Vibrations*. The input data required for a response spectrum analysis is discussed in Section 2.5 and Section 2.9.




You can switch among the results in the separate excitation directions X, Y and Z as well as among the combined results (SRSS or 100% / 30% rule) with the drop-down menu shown on the left. The tabulated results and also the graphic in the work window of RFEM will be updated according to your choice.

Only in the dynamic Tables 5.11 through 5.16, you get results of both the modal combinations and the directional combinations. The exported *RC*s in RFEM contain only the final result combinations, however, *i.e.* after the directional components have been combined.




When you have selected the export of result combinations as discussed in Section 2.9.5, the *RC*s are available in the drop-down list in the *toolbar* in RFEM shown on the left. You have also access to the *RC*s in the *Data* navigator using **Load Cases and Combinations** → **Result Combinations** or from the *Edit Load Cases and Combinations* dialogue box that can be accessed with the button.

## 4.2.1 Support Forces

The nodal support forces are provided in Table 5.11 which is shown in Figure 4.8. Similarly, the line support forces are listed in Table 5.12. In those tables, the maximum and minimum support forces are listed with the corresponding values.

5.11 Nodes - Support Forces

DLC1 - Response Spectra Analysis | X: RS 1 EN1998-1 CEN Horizontal De

| Node No. | A | B Support Forces | C | D | E Support Moments | F | G |
|---|---|---|---|---|---|---|---|
| | | $P_X$ [N] | $P_Y$ [N] | $P_Z$ [N] | $M_X$ [Nm] | $M_Y$ [Nm] | $M_Z$ [Nm] |
| 3 | max $P_X$ | **694.0** | 21.8 | -2729.8 | 0.00 | 0.00 | 0.04 |
| | min $P_X$ | **-694.0** | -21.8 | 2729.8 | 0.00 | 0.00 | -0.04 |
| | max $P_Y$ | 693.2 | **21.8** | -2726.4 | 0.00 | 0.00 | 0.04 |
| | min $P_Y$ | -693.2 | **-21.8** | 2726.4 | 0.00 | 0.00 | -0.04 |
| | max $P_Z$ | -694.0 | -21.8 | **2729.8** | 0.00 | 0.00 | -0.04 |
| | min $P_Z$ | 694.0 | 21.8 | **-2729.8** | 0.00 | 0.00 | 0.04 |
| | max $M_X$ | 0.0 | 0.0 | 0.0 | **0.00** | 0.00 | 0.00 |
| | min $M_X$ | 0.0 | 0.0 | 0.0 | **0.00** | 0.00 | 0.00 |
| | max $M_Y$ | 0.0 | 0.0 | 0.0 | 0.00 | **0.00** | 0.00 |
| | min $M_Y$ | 0.0 | 0.0 | 0.0 | 0.00 | **0.00** | 0.00 |
| | max $M_Z$ | 693.7 | 21.8 | -2729.0 | 0.00 | 0.00 | **0.04** |
| | min $M_Z$ | -693.7 | -21.8 | 2729.0 | 0.00 | 0.00 | **-0.04** |
| 6 | max $P_X$ | **706.9** | -4.3 | -2834.9 | 0.00 | 0.00 | -0.14 |
| | min $P_X$ | **-706.9** | 4.3 | 2834.9 | 0.00 | 0.00 | 0.14 |

Nodes - Support Forces | Nodes - Deformations | Nodes - Pseudo-Velocities | Nodes - Pseudo-Accelerations | Members - Internal Forces | Surfaces - Internal Forces | Surfaces - Basic Stresses | Surfaces - Basic Strains

Figure 4.8: Table *5.11 Nodes - Support forces* with maximum and minimum results in directions X, Y, and Z

## 4.2.2 Nodal Deformations

The nodal deformations are provided in Table 5.13 as shown in Figure 4.9. The maximum and minimum values are listed for every node.

5.13 Nodes - Deformations

DLC1 - Response Spectra Analysis | X: RS 1 EN1998-1 CEN Horizontal De

| Node No. | A | B Displacements | C | D | E Rotations | F | G |
|---|---|---|---|---|---|---|---|
| | | $u_X$ [mm] | $u_Y$ [mm] | $u_Z$ [mm] | $\varphi_X$ [rad] | $\varphi_Y$ [rad] | $\varphi_Z$ [rad] |
| 1 | max | 4.3 | 0.0 | 0.0 | 0.00 | 0.00 | 0.00 |
| | min | -4.3 | 0.0 | 0.0 | 0.00 | 0.00 | 0.00 |
| 2 | max | 2.5 | 0.0 | 0.0 | 0.00 | 0.00 | 0.00 |
| | min | -2.5 | 0.0 | 0.0 | 0.00 | 0.00 | 0.00 |
| 3 | max | 0.0 | 0.0 | 0.0 | 0.00 | 0.00 | 0.00 |
| | min | 0.0 | 0.0 | 0.0 | 0.00 | 0.00 | 0.00 |
| 4 | max | 4.4 | 0.0 | 0.0 | 0.00 | 0.00 | 0.00 |
| | min | -4.4 | 0.0 | 0.0 | 0.00 | 0.00 | 0.00 |
| 5 | max | 2.5 | 0.0 | 0.0 | 0.00 | 0.00 | 0.00 |
| | min | -2.5 | 0.0 | 0.0 | 0.00 | 0.00 | 0.00 |
| 6 | max | 0.0 | 0.0 | 0.0 | 0.00 | 0.00 | 0.00 |
| | min | 0.0 | 0.0 | 0.0 | 0.00 | 0.00 | 0.00 |
| 7 | max | 4.3 | 0.0 | 0.1 | 0.00 | 0.00 | 0.00 |
| | min | -4.3 | 0.0 | -0.1 | 0.00 | 0.00 | 0.00 |

Nodes - Support Forces | Nodes - Deformations | Nodes - Pseudo-Velocities | Nodes - Pseudo-Accelerations | Members - Internal Forces | Surfaces - Internal Forces | Surfaces - Basic Stresses | Surfaces - Basic Strains

Figure 4.9: Table *5.13 Nodes - Deformations* with maximum and minimum displacements and rotations

## 4.2.3 Pseudo Velocities and Accelerations

Additionally to the standard results, pseudo velocities and accelerations are provided in Table 5.14 shown in Figure 4.10 and Table 5.15 shown in Figure 4.11. The maximum and minimum values are listed for every node.

5.14 Nodes - Pseudo-Velocities

DLC1 - Response Spectra Analysis | X: RS 1 EN1998-1 CEN Horizontal De

| Node No. | | Velocities u'X [mm/s] | u'Y [mm/s] | u'Z [mm/s] | Angular Velocities φ'X [mrad/s] | φ'Y [mrad/s] | φ'Z [mrad/s] |
|---|---|---|---|---|---|---|---|
| 1 | max | 132.22 | 0.83 | 0.34 | 0.72 | 0.37 | 0.41 |
| | min | -132.22 | -0.83 | -0.34 | -0.72 | -0.37 | -0.41 |
| 2 | max | 76.70 | 0.51 | 0.25 | 0.16 | 2.11 | 0.21 |
| | min | -76.70 | -0.51 | -0.25 | -0.16 | -2.11 | -0.21 |
| 3 | max | 0.00 | 0.00 | 0.00 | 0.27 | 30.29 | 0.00 |
| | min | 0.00 | 0.00 | 0.00 | -0.27 | -30.29 | 0.00 |
| 4 | max | 134.55 | 0.94 | 0.35 | 0.69 | 0.41 | 0.54 |
| | min | -134.55 | -0.94 | -0.35 | -0.69 | -0.41 | -0.54 |
| 5 | max | 77.82 | 0.62 | 0.26 | 0.24 | 2.04 | 0.69 |
| | min | -77.82 | -0.62 | -0.26 | -0.24 | -2.04 | -0.69 |
| 6 | max | 0.00 | 0.00 | 0.00 | 0.12 | 30.79 | 0.00 |
| | min | 0.00 | 0.00 | 0.00 | -0.12 | -30.79 | 0.00 |
| 7 | max | 130.37 | 0.99 | 2.95 | 2.27 | 6.86 | 2.75 |
| | min | -130.37 | -0.99 | -2.95 | -2.27 | -6.86 | -2.75 |

Nodes - Support Forces | Nodes - Deformations | Nodes - Pseudo-Velocities | Nodes - Pseudo-Accelerations | Members - Internal Forces | Surfaces - Internal Forces | Surfaces - Basic Stresses | Surfaces - Basic Strains

Figure 4.10: Table *5.14 Nodes - Pseudo-Velocities* with maximum and minimum velocities and angular velocities

5.15 Nodes - Pseudo-Accelerations

DLC1 - Response Spectra Analysis | X: RS 1 EN1998-1 CEN Horizontal De

| Node No. | | Accelerations u"X [m/s²] | u"Y [m/s²] | u"Z [m/s²] | Angular Accelerations φ"X [rad/s²] | φ"Y [rad/s²] | φ"Z [rad/s²] |
|---|---|---|---|---|---|---|---|
| 1 | max | 4.05 | 0.03 | 0.01 | 0.02 | 0.01 | 0.01 |
| | min | -4.05 | -0.03 | -0.01 | -0.02 | -0.01 | -0.01 |
| 2 | max | 2.35 | 0.02 | 0.01 | 0.00 | 0.06 | 0.01 |
| | min | -2.35 | -0.02 | -0.01 | 0.00 | -0.06 | -0.01 |
| 3 | max | 0.00 | 0.00 | 0.00 | 0.01 | 0.93 | 0.00 |
| | min | 0.00 | 0.00 | 0.00 | -0.01 | -0.93 | 0.00 |
| 4 | max | 4.13 | 0.03 | 0.01 | 0.02 | 0.01 | 0.02 |
| | min | -4.13 | -0.03 | -0.01 | -0.02 | -0.01 | -0.02 |
| 5 | max | 2.39 | 0.02 | 0.01 | 0.01 | 0.06 | 0.02 |
| | min | -2.39 | -0.02 | -0.01 | -0.01 | -0.06 | -0.02 |
| 6 | max | 0.00 | 0.00 | 0.00 | 0.00 | 0.94 | 0.00 |
| | min | 0.00 | 0.00 | 0.00 | 0.00 | -0.94 | 0.00 |
| 7 | max | 4.00 | 0.03 | 0.09 | 0.07 | 0.21 | 0.08 |
| | min | -4.00 | -0.03 | -0.09 | -0.07 | -0.21 | -0.08 |

Nodes - Support Forces | Nodes - Deformations | Nodes - Pseudo-Velocities | Nodes - Pseudo-Accelerations | Members - Internal Forces | Surfaces - Internal Forces | Surfaces - Basic Stresses | Surfaces - Basic Strains

Figure 4.11: Table *5.15 Nodes - Pseudo-Accelerations* with maximum and minimum accelerations and angular accelerations

Pseudo velocities and accelerations are not included in the load cases or result combinations that have been exported to the main program RFEM.

## 4.2.4 Member Internal Forces

The internal forces of members are listed in Table 5.16 as shown in Figure 4.12. The maximum and minimum internal forces are provided with all corresponding values.

5.16 Members - Internal Forces

DLC1 - Response Spectra Analysis | X: RS 1 EN1998-1 CEN Horizontal De

| Member No. | Node No. | Location | At Point Of | Normal Force N [N] | Shear Force Vy [N] | Vz [N] | Moments Mx [Nm] | My [Nm] | Mz [Nm] |
|---|---|---|---|---|---|---|---|---|---|
| 1 | 2 | 0.000 | max N | **1035.3** | 1509.6 | 50.9 | 0.12 | -21.28 | 2651.22 |
| | | | min N | **-1035.3** | -1509.6 | -50.9 | -0.12 | 21.28 | -2651.22 |
| | | | max Vy | 1035.1 | **1509.8** | 50.9 | 0.12 | -21.34 | 2651.57 |
| | | | min Vy | -1035.1 | **-1509.8** | -50.9 | -0.12 | 21.34 | -2651.57 |
| | | | max Vz | 1032.3 | 1507.1 | **51.0** | 0.12 | -21.51 | 2646.87 |
| | | | min Vz | -1032.3 | -1507.1 | **-51.0** | -0.12 | 21.51 | -2646.87 |
| | | | max Mx | 1035.1 | 1509.7 | 50.9 | **0.12** | -21.33 | 2651.46 |
| | | | min Mx | -1035.1 | -1509.7 | -50.9 | **-0.12** | 21.33 | -2651.46 |
| | | | max My | -1018.8 | -1489.9 | -50.7 | -0.12 | **21.63** | -2616.66 |
| | | | min My | 1018.8 | 1489.9 | 50.7 | 0.12 | **-21.63** | 2616.66 |
| | | | max Mz | 1035.1 | 1509.8 | 50.9 | 0.12 | -21.34 | **2651.57** |
| | | | min Mz | -1035.1 | -1509.8 | -50.9 | -0.12 | 21.34 | **-2651.57** |
| 1 | 1 | 4.000 | max N | **1033.7** | 937.5 | 52.7 | 0.11 | 185.66 | -2337.69 |
| | | | min N | **-1033.7** | -937.5 | -52.7 | -0.11 | -185.66 | 2337.69 |

Nodes - Support Forces | Nodes - Deformations | Nodes - Pseudo-Velocities | Nodes - Pseudo-Accelerations | Members - Internal Forces | Surfaces - Internal Forces | Surfaces - Basic Stresses | Surfaces - Basic Strains

Figure 4.12: Table *5.16 Members - Internal Forces* with maximum and minimum results N, $V_y$, $V_Z$, $M_X$, $M_Y$, $M_Z$

### 4.2.5 Surface Internal Forces

Table 5.17 provides the surface internal forces as shown in Figure 4.13. The maximum and minimum values together with the coordinates of the *FE-Nodes* are provided in that table.

5.17 Surfaces - Internal Forces

DLC1 - Response Spectra Analysis | X: RS 1 EN1998-1 CEN Horizontal De

| Surface No. | FE Mesh Node | | Point Coordinate X [m] | Y [m] | Z [m] | Moments $m_x$[Nm/m] | $m_y$[Nm/m] | $m_{xy}$[Nm/m] | Shear Forces $v_x$[N/m] | $v_y$[N/m] | Normal Forces $n_x$[N/m] | $n_y$[N/m] | $n_{xy}$[N/m] |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 1 | max | 0.000 | 0.000 | -8.000 | 3410.749 | 179.364 | 1376.558 | 20711.48 | 3835.36 | 47851.08 | 2267.05 | 15351.72 |
| | | min | 0.000 | 0.000 | -8.000 | -3410.749 | -179.364 | -1376.558 | -20711.48 | -3835.36 | -47851.08 | -2267.05 | -15351.72 |
| | 4 | max | 0.000 | 5.000 | -8.000 | 3494.490 | 149.506 | 1378.318 | 21178.28 | 4023.94 | 49223.54 | 1743.35 | 15547.88 |
| | | min | 0.000 | 5.000 | -8.000 | -3494.490 | -149.506 | -1378.318 | -21178.28 | -4023.94 | -49223.54 | -1743.35 | -15547.88 |
| | 7 | max | 4.000 | 0.000 | -8.000 | 5643.206 | 943.374 | 3820.891 | 38021.00 | 9872.25 | 18539.44 | 14545.79 | 8557.06 |
| | | min | 4.000 | 0.000 | -8.000 | -5643.206 | -943.374 | -3820.891 | -38021.00 | -9872.25 | -18539.44 | -14545.79 | -8557.06 |
| | 10 | max | 4.000 | 1.000 | -8.000 | 997.008 | 224.793 | 1006.452 | 4374.81 | 8154.52 | 9139.31 | 28275.37 | 8301.64 |
| | | min | 4.000 | 1.000 | -8.000 | -997.008 | -224.793 | -1006.452 | -4374.81 | -8154.52 | -9139.31 | -28275.37 | -8301.64 |
| | 12 | max | 4.000 | 2.000 | -8.000 | 2492.893 | 105.764 | 618.384 | 9568.30 | 3330.00 | 17529.01 | 39084.54 | 2833.51 |
| | | min | 4.000 | 2.000 | -8.000 | -2492.893 | -105.764 | -618.384 | -9568.30 | -3330.00 | -17529.01 | -39084.54 | -2833.51 |
| | 14 | max | 4.000 | 3.000 | -8.000 | 3313.376 | 148.791 | 624.993 | 12722.82 | 3102.29 | 22731.72 | 42124.74 | 2519.22 |
| | | min | 4.000 | 3.000 | -8.000 | -3313.376 | -148.791 | -624.993 | -12722.82 | -3102.29 | -22731.72 | -42124.74 | -2519.22 |
| | 16 | max | 4.000 | 4.000 | -8.000 | 1009.703 | 229.594 | 1055.992 | 4253.03 | 8654.65 | 9154.55 | 28745.51 | 8963.27 |
| | | min | 4.000 | 4.000 | -8.000 | -1009.703 | -229.594 | -1055.992 | -4253.03 | -8654.65 | -9154.55 | -28745.51 | -8963.27 |

Nodes - Support Forces | Nodes - Deformations | Nodes - Pseudo-Velocities | Nodes - Pseudo-Accelerations | Members - Internal Forces | Surfaces - Internal Forces | Surfaces - Basic Stresses | Surfaces - Basic Strains

Figure 4.13: Table *5.17 Surfaces - Internal Forces* with maximum and minimum results $m_x$, $m_y$, $m_{xy}$, $v_x$, $v_y$, $n_x$, $n_y$, and $n_{xy}$

### 4.2.6 Surface Basic Stresses and Strains

The basic stresses and strains of surfaces are listed in Table 5.18 and Table 5.19. The basic stresses are shown in Figure 4.14.

5.18 Surfaces - Basic Stresses

DLC1 - Response Spectra Analysis | X: RS 1 EN1998-1 CEN Horizontal De

| Surface No. | FE Mesh Node | | Point Coordinate X [m] | Y [m] | Z [m] | Normal and Shear Stresses $\sigma_{X+}$ [kN/cm²] | $\sigma_{Y+}$ [kN/cm²] | $\sigma_{XY+}$ [kN/cm²] | $\sigma_{X-}$ [kN/cm²] | $\sigma_{Y-}$ [kN/cm²] | $\sigma_{XY-}$ [kN/cm²] | $\tau_X$ [kN/cm²] | $\tau_Y$ [kN/cm²] |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 1 | max | 0.000 | 0.000 | -8.000 | 0.050 | 0.003 | 0.023 | 0.110 | 0.006 | 0.042 | 0.059 | -0.006 |
| | | min | 0.000 | 0.000 | -8.000 | -0.050 | -0.003 | -0.023 | -0.110 | -0.006 | -0.042 | -0.059 | 0.006 |
| | 4 | max | 0.000 | 5.000 | -8.000 | 0.051 | 0.002 | 0.023 | 0.113 | 0.005 | 0.042 | 0.060 | -0.006 |
| | | min | 0.000 | 5.000 | -8.000 | -0.051 | -0.002 | -0.023 | -0.113 | -0.005 | -0.042 | -0.060 | 0.006 |
| | 7 | max | 4.000 | 0.000 | -8.000 | 0.144 | 0.031 | 0.095 | 0.121 | 0.013 | 0.084 | 0.198 | -0.023 |
| | | min | 4.000 | 0.000 | -8.000 | -0.144 | -0.031 | -0.095 | -0.121 | -0.013 | -0.084 | -0.198 | 0.023 |
| | 10 | max | 4.000 | 1.000 | -8.000 | 0.029 | 0.023 | 0.018 | 0.018 | 0.012 | 0.029 | 0.045 | 0.007 |
| | | min | 4.000 | 1.000 | -8.000 | -0.029 | -0.023 | -0.018 | -0.018 | -0.012 | -0.029 | -0.045 | -0.007 |
| | 12 | max | 4.000 | 2.000 | -8.000 | 0.069 | 0.022 | 0.013 | 0.047 | 0.026 | 0.016 | 0.073 | 0.019 |
| | | min | 4.000 | 2.000 | -8.000 | -0.069 | -0.022 | -0.013 | -0.047 | -0.026 | -0.016 | -0.073 | -0.019 |
| | 14 | max | 4.000 | 3.000 | -8.000 | 0.092 | 0.023 | 0.013 | 0.063 | 0.030 | 0.016 | 0.094 | 0.021 |
| | | min | 4.000 | 3.000 | -8.000 | -0.092 | -0.023 | -0.013 | -0.063 | -0.030 | -0.016 | -0.094 | -0.021 |
| | 16 | max | 4.000 | 4.000 | -8.000 | 0.029 | 0.023 | 0.019 | 0.018 | 0.013 | 0.030 | 0.046 | 0.007 |
| | | min | 4.000 | 4.000 | -8.000 | -0.029 | -0.023 | -0.019 | -0.018 | -0.013 | -0.030 | -0.046 | -0.007 |

Nodes - Support Forces | Nodes - Deformations | Nodes - Pseudo-Velocities | Nodes - Pseudo-Accelerations | Members - Internal Forces | Surfaces - Internal Forces | Surfaces - Basic Stresses | Surfaces - Basic Strains

Figure 4.14: Table *5.18 Surfaces - Basic Stresses* with maximum and minimum normal and shear stresses

### 4.2.7 Exported Result Combinations

When performing a response spectrum in the add-on module *RF-DYNAM Pro - Forced Vibrations*, result combinations are produced only from the finally combined results. Two steps of combinations are undertaken internally within RF-DYNAM Pro to achieve those results: (1) The modal responses are combined with the *SRSS* or *CQC* rule, and (2) the directional results are combined either with the *SRSS* or with the 100% / 30% (40%) rule. Those final *RC*s will be exported.

The list of generated result combinations can be found in the *Edit Load Cases and Combinations* dialogue box displayed in Figure 4.15.

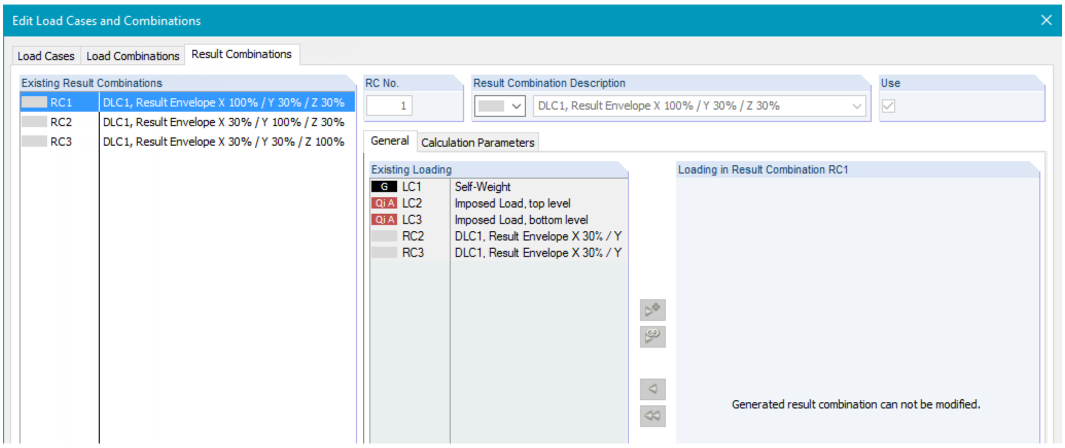



Figure 4.15: Dialogue box *Edit Load Cases and Combinations* with exported *RC*s from response spectrum analysis, containing only result combinations of directional combination

The exported *RC*s cannot be modified, and the calculation parameters cannot be changed.

### 4.2.8 Export to Excel or CSV

You can export all results tables that have been discussed in this section to MS Excel or save them in the *CSV* format using the **File** → **Export** option. The list of results tables is shown in Figure 4.16.

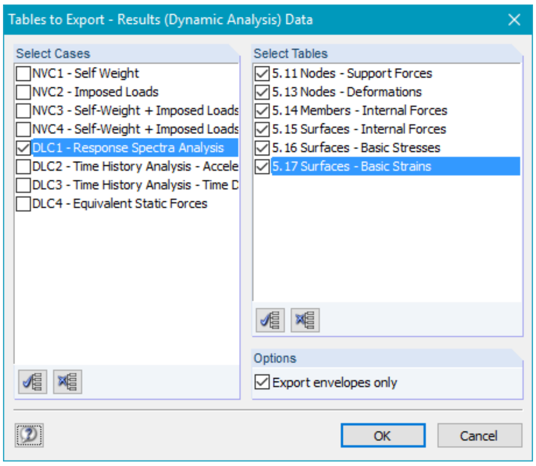
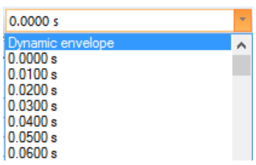



Figure 4.16: Results tables to export for dynamic load cases (*DLC*) with response spectrum analysis

The *Export envelopes only* option is selected by default. Only the results tables that belong to the *Dynamic Envelopes* (*i.e.* X100% / Y30% / Z30%) will be exported. When you clear this check box, the results tables that belong to the single directions will also be exported.

## 4.3 Dynamic Load Cases - Time History Analysis

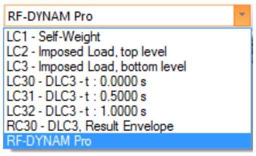
The results tables that belong to *Time History Analysis Cases* are available when the corresponding dynamic load case (*DLC*) is chosen in the drop-down menu. Tables 5.18 to 5.26 belong to this type of dynamic load cases. The linear time history analysis belongs to the module *RF-DYNAM Pro - Forced Vibrations*, the nonlinear time history analysis to the module *RF-DYNAM Pro - Nonlinear Time History*.




The input data required for a time history analysis is discussed in Section 2.10. The definition of accelerograms is described in Section 2.6, the definition of time diagrams in Section 2.7.

Results are available separately for each time step. The *Dynamic envelope* is an envelope of results providing the maximum and minimum results of all time steps. You can switch among the results using the drop-down menu shown on the left. The tabulated results and also the graphic in the work window of RFEM will be updated according to your choice.

The results can also be displayed versus time in the *Time Course Monitor* which is accessible with the button in the menu bar of the table or in the panel. You can activate the animation in the main graphic by using the button in the menu bar of RFEM or in the panel. When you have the *Time Course Monitor* open at the same time, the cursor moves through the plot while the animation is running.

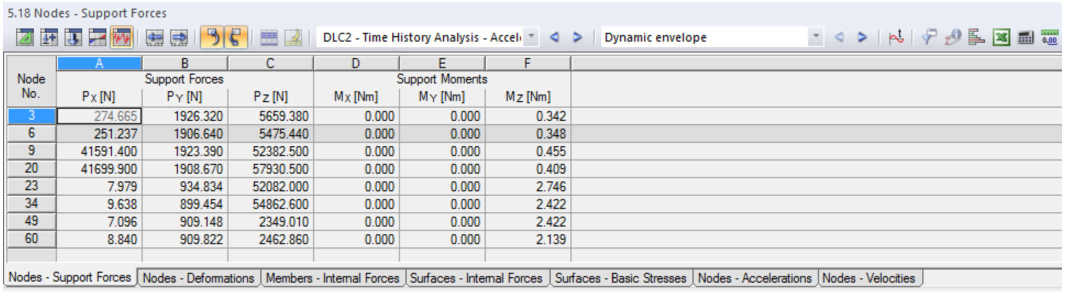



When you have selected the export of load cases and/or result combinations as discussed in Section 2.10.7, the *LC*s and *RC*s are available in the drop-down list in the *toolbar* in RFEM shown on the left. You can also access to the *LC*s in the *Data* navigator using **Load Cases and Combinations** → **Load Cases** or similarly for the *RC*s using **Result Combinations**. Load cases and result combinations are also accessible in the *Edit Load Cases and Combinations* dialogue box.

Load cases exported from a time history analysis are special: They do not contain loads, only the results of the specific time step.

## 4.3.1 Support Forces

The nodal support forces are provided in Table 5.18 which is shown in Figure 4.17. Similarly, the line support forces are listed in Table 5.19. The maximum values are provided when the *Dynamic envelope* has been selected.



5.18 Nodes - Support Forces

DLC2 - Time History Analysis - Accel | Dynamic envelope

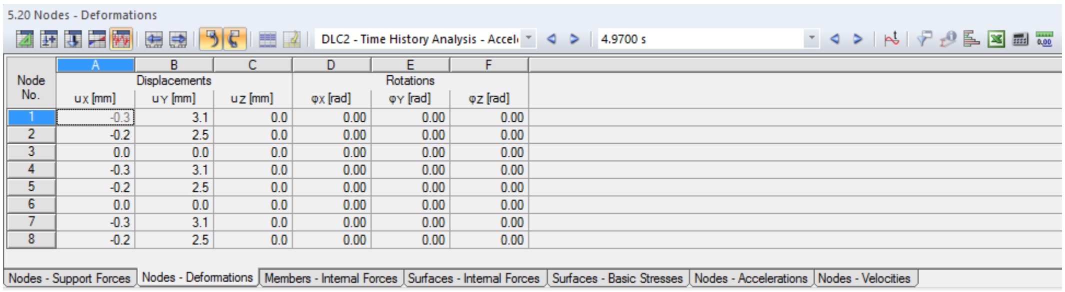
| Node No. | A | B | C | D | E | F |
|---|---|---|---|---|---|---|
| | Support Forces | | | Support Moments | | |
| | Px [N] | PY [N] | PZ [N] | MX [Nm] | MY [Nm] | MZ [Nm] |
| 3 | 274.665 | 1926.320 | 5659.380 | 0.000 | 0.000 | 0.342 |
| 6 | 251.237 | 1906.640 | 5475.440 | 0.000 | 0.000 | 0.348 |
| 9 | 41591.400 | 1923.390 | 52382.500 | 0.000 | 0.000 | 0.455 |
| 20 | 41699.900 | 1908.670 | 57930.500 | 0.000 | 0.000 | 0.409 |
| 23 | 7.979 | 934.834 | 52082.000 | 0.000 | 0.000 | 2.746 |
| 34 | 9.638 | 899.454 | 54862.600 | 0.000 | 0.000 | 2.422 |
| 49 | 7.096 | 909.148 | 2349.010 | 0.000 | 0.000 | 2.422 |
| 60 | 8.840 | 909.822 | 2462.860 | 0.000 | 0.000 | 2.139 |

Nodes - Support Forces | Nodes - Deformations | Members - Internal Forces | Surfaces - Internal Forces | Surfaces - Basic Stresses | Nodes - Accelerations | Nodes - Velocities

Figure 4.17: Table *5.18 Nodes - Support Forces* with results in directions X, Y, and Z

## 4.3.2 Nodal Deformations

The nodal deformations are provided in Table 5.20 as shown in Figure 4.18. The maximum values are provided when the *Dynamic envelope* has been selected.



5.20 Nodes - Deformations

DLC2 - Time History Analysis - Accel | 4.9700 s

| Node No. | A | B | C | D | E | F |
|---|---|---|---|---|---|---|
| | Displacements | | | Rotations | | |
| | uX [mm] | uY [mm] | uZ [mm] | φX [rad] | φY [rad] | φZ [rad] |
| 1 | -0.3 | 3.1 | 0.0 | 0.00 | 0.00 | 0.00 |
| 2 | -0.2 | 2.5 | 0.0 | 0.00 | 0.00 | 0.00 |
| 3 | 0.0 | 0.0 | 0.0 | 0.00 | 0.00 | 0.00 |
| 4 | -0.3 | 3.1 | 0.0 | 0.00 | 0.00 | 0.00 |
| 5 | -0.2 | 2.5 | 0.0 | 0.00 | 0.00 | 0.00 |
| 6 | 0.0 | 0.0 | 0.0 | 0.00 | 0.00 | 0.00 |
| 7 | -0.3 | 3.1 | 0.0 | 0.00 | 0.00 | 0.00 |
| 8 | -0.2 | 2.5 | 0.0 | 0.00 | 0.00 | 0.00 |

Nodes - Support Forces | Nodes - Deformations | Members - Internal Forces | Surfaces - Internal Forces | Surfaces - Basic Stresses | Nodes - Accelerations | Nodes - Velocities

Figure 4.18: Table *5.20 Nodes - Deformations* with displacements $u_X$, $u_Y$, $u_Z$ and rotations $\varphi_X$, $\varphi_Y$, $\varphi_Z$

## 4.3.3 Member Internal Forces

The internal forces of members are listed in Table 5.21 as shown in Figure 4.19. The maximum values are provided when the *Dynamic envelope* has been selected.

5.21 Members - Internal Forces

DLC2 - Time History Analysis - Accel | Dynamic envelope

| Member No. | A Node | B X [m] | C Normal Force N [N] | D Shear Force $V_y$ [N] | E $V_z$ [N] | F Moments $M_X$ [Nm] | G $M_Y$ [Nm] | H $M_Z$ [Nm] |
|---|---|---|---|---|---|---|---|---|
| 1 | 2 | 0.000 | 2520.559 | 813.351 | 1041.147 | 0.280 | 1969.617 | 1593.518 |
| | | 0.333 | 2520.559 | 813.351 | 1041.147 | 0.280 | 1586.546 | 1322.401 |
| | | 0.667 | 2520.559 | 813.351 | 1041.147 | 0.280 | 1227.807 | 1051.284 |
| | | 1.000 | 2520.559 | 813.351 | 1041.147 | 0.280 | 869.069 | 780.167 |
| | | 1.333 | 2520.559 | 813.351 | 1041.147 | 0.280 | 510.331 | 509.051 |
| | | 1.667 | 2520.559 | 813.351 | 1041.147 | 0.280 | 228.503 | 237.934 |
| | | 2.000 | 2520.559 | 813.351 | 1041.147 | 0.280 | 362.888 | 58.901 |
| | | 2.000 | 2520.559 | 813.351 | 1041.147 | 0.280 | 362.905 | 58.914 |
| | | 2.333 | 2520.559 | 813.351 | 1041.147 | 0.280 | 668.926 | 291.195 |
| | | 2.667 | 2520.559 | 813.351 | 1041.147 | 0.280 | 996.201 | 525.518 |
| | | 3.000 | 2520.559 | 813.351 | 1041.147 | 0.280 | 1326.312 | 759.841 |
| | | 3.333 | 2520.559 | 813.351 | 1041.147 | 0.280 | 1656.423 | 994.163 |
| | | 3.667 | 2520.559 | 813.351 | 1041.147 | 0.280 | 1994.627 | 1228.486 |
| | 1 | 4.000 | 2520.559 | 813.351 | 1041.147 | 0.280 | 2341.676 | 1462.809 |
| 2 | 3 | 0.000 | 6922.875 | 274.665 | 1587.437 | 0.431 | 0.001 | 0.000 |
| | | 0.333 | 6922.875 | 274.665 | 1587.437 | 0.431 | 529.145 | 89.991 |
| | | 0.667 | 6922.875 | 274.665 | 1587.437 | 0.431 | 1058.291 | 179.983 |
| | | 1.000 | 6922.875 | 274.665 | 1587.437 | 0.431 | 1587.436 | 269.974 |

Nodes - Support Forces | Nodes - Deformations | Members - Internal Forces | Surfaces - Internal Forces | Surfaces - Basic Stresses | Nodes - Accelerations | Nodes - Velocities

Figure 4.19: Table *5.21 Members - Internal Forces* with normal forces N, shear forces $V_y$ and $V_Z$, as well as moments $M_X$, $M_Y$, and $M_Z$

## 4.3.4 Surface Results

Table 5.22 provides the surface internal forces as shown in Figure 4.20.

5.22 Surfaces - Internal Forces

DLC2 - Time History Analysis - Accel | Dynamic envelope

| Surface No. | A FE Mesh Point | B Point Coordinate X [m] | C Y [m] | D Z [m] | E Moments $m_x$[Nm/m] | F $m_y$[Nm/m] | G $m_{xy}$[Nm/m] | H Shear Forces $v_x$[Nm] | I $v_y$[Nm] | J Normal Forces $n_x$[Nm] | K $n_y$[Nm] | L $n_{xy}$[Nm] |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 1 | 0.000 | 0.000 | -8.000 | 2212.890 | 1536.225 | 1488.380 | 14096.240 | 8306.479 | 31063.811 | 32369.551 | 15781.600 |
| | 4 | 0.000 | 5.000 | -8.000 | 2713.237 | 1604.514 | 1207.685 | 9832.038 | 9816.210 | 29938.760 | 37789.371 | 13201.730 |
| | 7 | 4.000 | 0.000 | -8.000 | 3196.911 | 1651.514 | 1806.931 | 19142.881 | 10332.910 | 11261.030 | 34375.730 | 15594.910 |
| | 10 | 4.000 | 1.000 | -8.000 | 236.673 | 458.444 | 505.942 | 691.878 | 6670.127 | 1338.121 | 11519.150 | 5528.640 |
| | 12 | 4.000 | 2.000 | -8.000 | 760.427 | 385.604 | 258.256 | 3404.830 | 3919.046 | 7117.260 | 23240.779 | 3398.660 |
| | 14 | 4.000 | 3.000 | -8.000 | 1194.780 | 387.147 | 312.262 | 4442.152 | 3945.794 | 9086.512 | 26835.369 | 3434.363 |
| | 16 | 4.000 | 4.000 | -8.000 | 185.000 | 411.120 | 506.823 | 351.166 | 5661.204 | 1390.249 | 22548.920 | 5778.351 |
| | 18 | 4.000 | 5.000 | -8.000 | 3744.523 | 1737.086 | 2015.475 | 23645.689 | 8956.198 | 9394.306 | 38941.148 | 16568.461 |
| | 126 | 1.000 | 0.500 | -8.000 | 289.893 | 193.475 | 143.643 | 1401.210 | 321.285 | 3603.118 | 1620.247 | 4598.778 |
| | 144 | 1.500 | 0.000 | -8.000 | 357.987 | 111.342 | 120.712 | 4856.914 | 272.580 | 8413.378 | 535.895 | 5055.832 |

Nodes - Support Forces | Nodes - Deformations | Members - Internal Forces | Surfaces - Internal Forces | Surfaces - Basic Stresses | Nodes - Accelerations | Nodes - Velocities

Figure 4.20: Table *5.22 Surfaces - Internal Forces* with results $m_x$, $m_y$, $m_{xy}$, $v_x$, $v_y$, $n_x$, $n_y$, and $n_{xy}$

The stresses of the surfaces are listed in Table 5.23 as shown in Figure 4.21.

5.23 Surfaces - Basic Stresses

DLC2 - Time History Analysis - Accel | Dynamic envelope

| Surface No. | A FE Mesh Point | B Point Coordinate X [m] | C Y [m] | D Z [m] | E Normal and Shear Stresses $\sigma_{X+}$ [kN/cm²] | F $\sigma_{Y+}$ [kN/cm²] | G $\sigma_{XY+}$ [kN/cm²] | H $\sigma_{X-}$ [kN/cm²] | I $\sigma_{Y-}$ [kN/cm²] | J $\sigma_{XY-}$ [kN/cm²] | K $\tau_X$ [kN/cm²] | L $\tau_Y$ [kN/cm²] |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 1 | 0.000 | 0.000 | -8.000 | 0.033 | 0.019 | 0.025 | 0.074 | 0.052 | 0.030 | 0.053 | 0.000 |
| | 4 | 0.000 | 5.000 | -8.000 | 0.044 | 0.018 | 0.017 | 0.059 | 0.059 | 0.028 | 0.050 | 0.000 |
| | 7 | 4.000 | 0.000 | -8.000 | 0.081 | 0.026 | 0.052 | 0.078 | 0.055 | 0.057 | 0.115 | -0.004 |
| | 10 | 4.000 | 1.000 | -8.000 | 0.006 | 0.007 | 0.013 | 0.006 | 0.015 | 0.016 | 0.016 | 0.001 |
| | 12 | 4.000 | 2.000 | -8.000 | 0.022 | 0.014 | 0.006 | 0.018 | 0.021 | 0.007 | 0.023 | 0.013 |
| | 14 | 4.000 | 3.000 | -8.000 | 0.034 | 0.011 | 0.006 | 0.019 | 0.028 | 0.007 | 0.036 | 0.010 |
| | 16 | 4.000 | 4.000 | -8.000 | 0.004 | 0.007 | 0.012 | 0.006 | 0.022 | 0.017 | 0.020 | -0.003 |
| | 18 | 4.000 | 5.000 | -8.000 | 0.094 | 0.022 | 0.055 | 0.062 | 0.063 | 0.058 | 0.120 | -0.002 |
| | 126 | 1.000 | 0.500 | -8.000 | 0.007 | 0.005 | 0.004 | 0.008 | 0.005 | 0.003 | 0.007 | 0.002 |
| | 144 | 1.500 | 0.000 | -8.000 | 0.009 | 0.002 | 0.006 | 0.007 | 0.001 | 0.001 | 0.010 | 0.000 |

Nodes - Support Forces | Nodes - Deformations | Members - Internal Forces | Surfaces - Internal Forces | Surfaces - Basic Stresses | Nodes - Accelerations | Nodes - Velocities

Figure 4.21: Table *5.23 Surfaces - Basic Stresses* with normal and shear stresses

## 4.3.5 Nodal Accelerations and Velocities

Additionally to the standard results, nodal accelerations and velocities are provided in Table 5.25 shown in Figure 4.22 and Table 5.26 shown in Figure 4.23. Those results are available for each time step. The dynamic envelope is provided, too.

5.25 Nodes - Accelerations

DLC2 - Time History Analysis - Accel | 5.0400 s

| Node No. | A | B | C | D | E | F |
|---|---|---|---|---|---|---|
| | Accelerations | | | Angular Accelerations | | |
| | $\ddot{u}_X$ [m/s²] | $\ddot{u}_Y$ [m/s²] | $\ddot{u}_Z$ [m/s²] | $\ddot{\varphi}_X$ [rad/s²] | $\ddot{\varphi}_Y$ [rad/s²] | $\ddot{\varphi}_Z$ [rad/s²] |
| 1 | 0.33 | 0.00 | 0.04 | 0.08 | -0.09 | 0.01 |
| 2 | 0.09 | 0.15 | 0.03 | 0.05 | -0.09 | 0.01 |
| 3 | 0.00 | 0.00 | 0.00 | 0.03 | 0.01 | 0.00 |
| 4 | 0.34 | 0.02 | 0.04 | -0.10 | -0.09 | -0.01 |
| 5 | 0.10 | 0.17 | 0.03 | -0.06 | -0.08 | -0.01 |
| 6 | 0.00 | 0.00 | 0.00 | 0.09 | 0.01 | 0.00 |
| 7 | 0.35 | -0.02 | 0.04 | 0.07 | 0.05 | -0.02 |
| 8 | 0.12 | 0.14 | 0.02 | 0.07 | 0.08 | 0.00 |
| 9 | 0.00 | 0.00 | 0.00 | 0.02 | -0.08 | 0.00 |
| 10 | 0.37 | -0.02 | 0.13 | 0.08 | 0.10 | -0.01 |

Nodes - Support Forces | Nodes - Deformations | Members - Internal Forces | Surfaces - Internal Forces | Surfaces - Basic Stresses | Nodes - Accelerations | Nodes - Velocities

Figure 4.22: Table *5.25 Nodes - Accelerations* with accelerations $\ddot{u}_X$, $\ddot{u}_Y$, $\ddot{u}_Z$ and rotations $\ddot{\varphi}_X$, $\ddot{\varphi}_Y$, $\ddot{\varphi}_Z$

5.26 Nodes - Velocities

DLC2 - Time History Analysis - Accel | 5.0400 s

| Node No. | A | B | C | D | E | F |
|---|---|---|---|---|---|---|
| | Velocities | | | Angular Velocities | | |
| | $\dot{u}_X$ [mm/s] | $\dot{u}_Y$ [mm/s] | $\dot{u}_Z$ [mm/s] | $\dot{\varphi}_X$ [mrad/s] | $\dot{\varphi}_Y$ [mrad/s] | $\dot{\varphi}_Z$ [mrad/s] |
| 1 | 30.71 | -25.27 | 0.87 | 1.21 | -1.71 | 0.20 |
| 2 | 18.63 | -18.03 | 0.58 | 0.19 | -2.15 | 0.59 |
| 3 | 0.00 | 0.00 | 0.00 | -6.72 | -5.90 | 0.00 |
| 4 | 31.05 | -24.78 | 0.67 | -2.14 | -1.65 | -0.07 |
| 5 | 18.68 | -17.59 | 0.44 | -2.23 | -1.85 | 0.05 |
| 6 | 0.00 | 0.00 | 0.00 | -5.42 | -6.07 | 0.00 |
| 7 | 30.83 | -25.60 | 0.43 | 0.84 | -0.01 | -0.77 |
| 8 | 18.80 | -18.13 | 0.07 | 0.22 | 0.07 | -0.73 |
| 9 | 0.00 | 0.00 | 0.00 | -6.77 | -7.07 | 0.00 |
| 10 | 31.37 | -25.59 | 1.76 | 1.28 | 1.77 | -0.26 |

Nodes - Support Forces | Nodes - Deformations | Members - Internal Forces | Surfaces - Internal Forces | Surfaces - Basic Stresses | Nodes - Accelerations | Nodes - Velocities

Figure 4.23: Table *5.26 Nodes - Velocities* with velocities $\dot{u}_X$, $\dot{u}_Y$, $\dot{u}_Z$ and rotations $\dot{\varphi}_X$, $\dot{\varphi}_Y$, $\dot{\varphi}_Z$

Nodal accelerations and velocities are not included in the load cases or result combinations that have been exported to the main program RFEM.

## 4.3.6 Time Course Monitor

All results that are listed in the tables discussed above – apart from the surface stresses – can be displayed versus the time with the *Time Course Monitor*. It represents the most important postprocessing tool of a time history analysis. The *Time Course Monitor* is illustrated in Figure 4.24.

This graphic is available with the [Time Course Monitor] button. It can be found in the table toolbar and in the *Factors* tab of the panel.

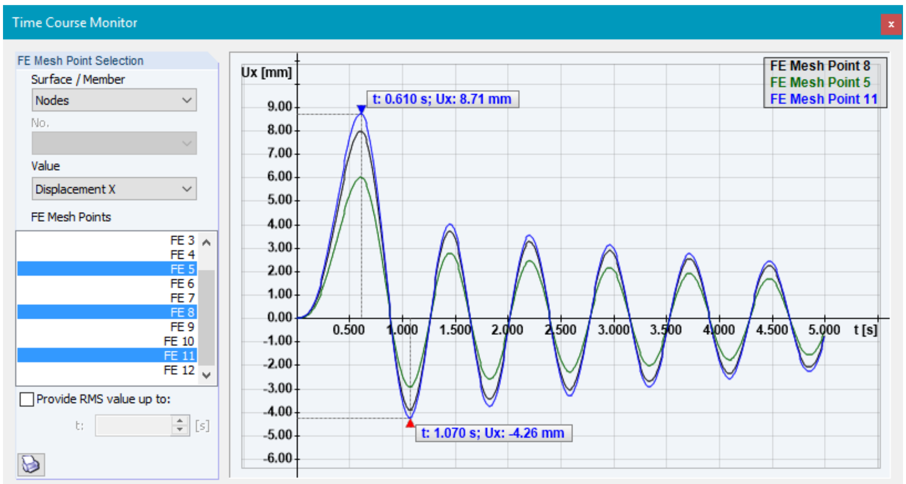
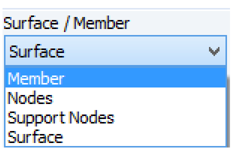
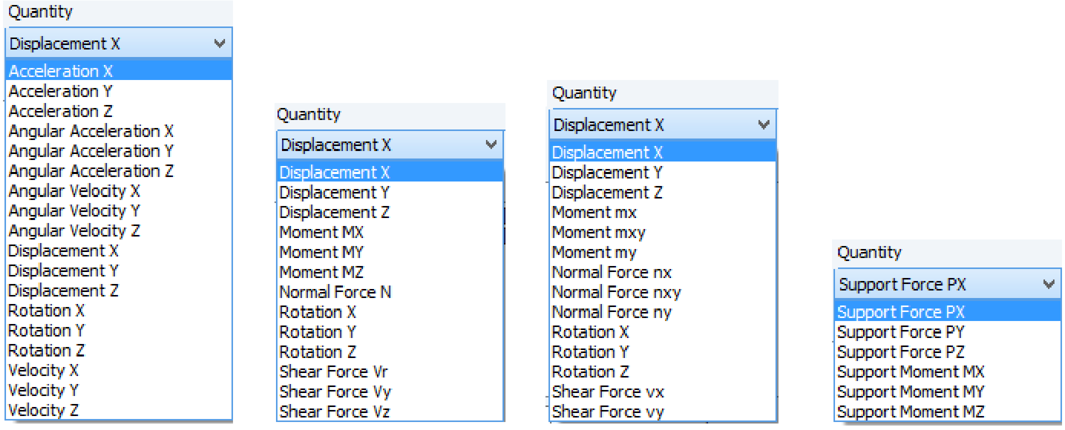



Figure 4.24: *Time Course Monitor* to display results versus time – here the displacements $u_X$ of three nodes are displayed, showing the maximum and minimum values.

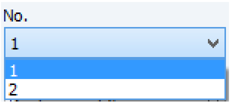



You can choose between *Nodes*, *Members*, *Surfaces*, and *Support Nodes* in the drop-down menu as seen on the left. Depending on this choice, different results are available as shown in Figure 4.25.




a) Results on nodes b) Results on members c) Results on surfaces d) Results on supports

Figure 4.25: Available results for (a) nodes, (b) members, (c) surfaces, and (d) support nodes

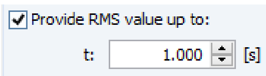



For members and surfaces, you have to select the member or surface number from the list.

Depending on the choice of objects, either a list of finite element mesh points (see Figure 4.24) or a list of distances on members (see Figure 4.26) is available. A multiple choice of listed points or positions is possible by using the [Ctrl] or [Shift] key.

The legend in the graphic is set automatically; the axis labels and axis scales are also adjusted depending on the selection. The minimum and maximum results with the corresponding time steps are provided. You can use the mouse wheel to zoom into the graph. Values will be displayed on the pointer when it is moved along the graph.

The work window of RFEM, the tables and the *Time Course Monitor* are interacting: When you select the row of a specific member in a table or a node in the graphic, the settings in the *Time Course Monitor* will be adjusted. You only have to choose the result that is to be displayed (Figure 4.25).




The root mean square (*RMS*) value of any results up to a specific time can be displayed if the check box shown on the left is selected and a time is defined. The RMS value is calculated as follows:

$$R_{RMS} = \sqrt{\frac{1}{n}(R_1^2 + R_2^2 + \ldots + R_n^2)} \tag{4.8}$$

where $R_n$ are the results of each time step with $n$ considered time steps. The RMS value is illustrated in the *Time Course Monitor* as shown in Figure 4.26.

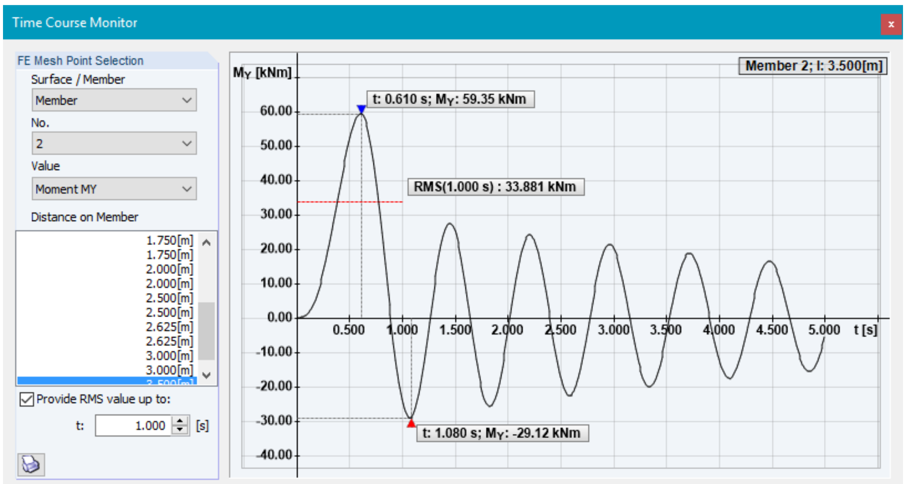
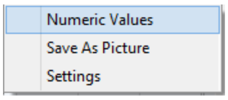


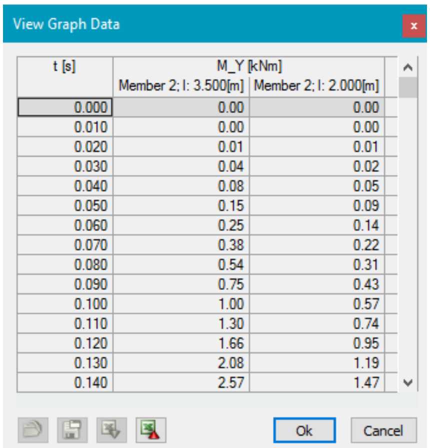

Figure 4.26: *Time Course Monitor* with *RMS value* – here the moment $M_Y$ is displayed versus time, showing the RMS value up to 1.0 s.

The RMS value is only available when a single graph is plotted. If you overlay the results of several nodes, the RMS check box is not available.

With the [Print] button you can print the *Time Course Monitor* to the printout report. In the following dialogue box, you can define the header of this graphic to be used in the report.




When you right-click on the *Time Course Monitor*, you have three options: (1) You can access the *Numeric Values* in tabular view. The time steps are listed with all results displayed in the graphics. An example is shown in Figure 4.27. The values as tabulated can be exported to Excel via the button. (2) You can save the *Time Course Monitor* as *Picture*, and (3) you can access the *Settings* of the graph. The *Settings* dialogue box of the *Time Course Monitor* is shown in Figure 4.28.




| t [s] | M_Y [kNm] | |
|---|---|---|
| | Member 2; l: 3.500[m] | Member 2; l: 2.000[m] |
| 0.000 | 0.00 | 0.00 |
| 0.010 | 0.00 | 0.00 |
| 0.020 | 0.01 | 0.01 |
| 0.030 | 0.04 | 0.02 |
| 0.040 | 0.08 | 0.05 |
| 0.050 | 0.15 | 0.09 |
| 0.060 | 0.25 | 0.14 |
| 0.070 | 0.38 | 0.22 |
| 0.080 | 0.54 | 0.31 |
| 0.090 | 0.75 | 0.43 |
| 0.100 | 1.00 | 0.57 |
| 0.110 | 1.30 | 0.74 |
| 0.120 | 1.66 | 0.95 |
| 0.130 | 2.08 | 1.19 |
| 0.140 | 2.57 | 1.47 |

Figure 4.27: Numeric values of plot displayed in *Time Course Monitor*

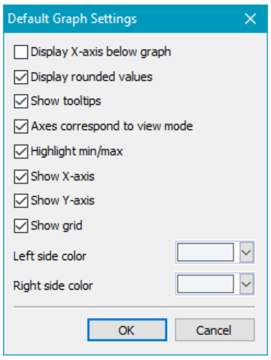



Figure 4.28: Settings of *Time Course Monitor*

## 4.3.7 Exported Load Cases

When you have performed a time history analysis in *RF-DYNAM Pro - Forced Vibrations*, you can export the results of single time steps to load cases (*LC*). The exported load cases for single time steps are listed in the *Edit Load Cases and Combinations* dialogue box shown in Figure 4.29.

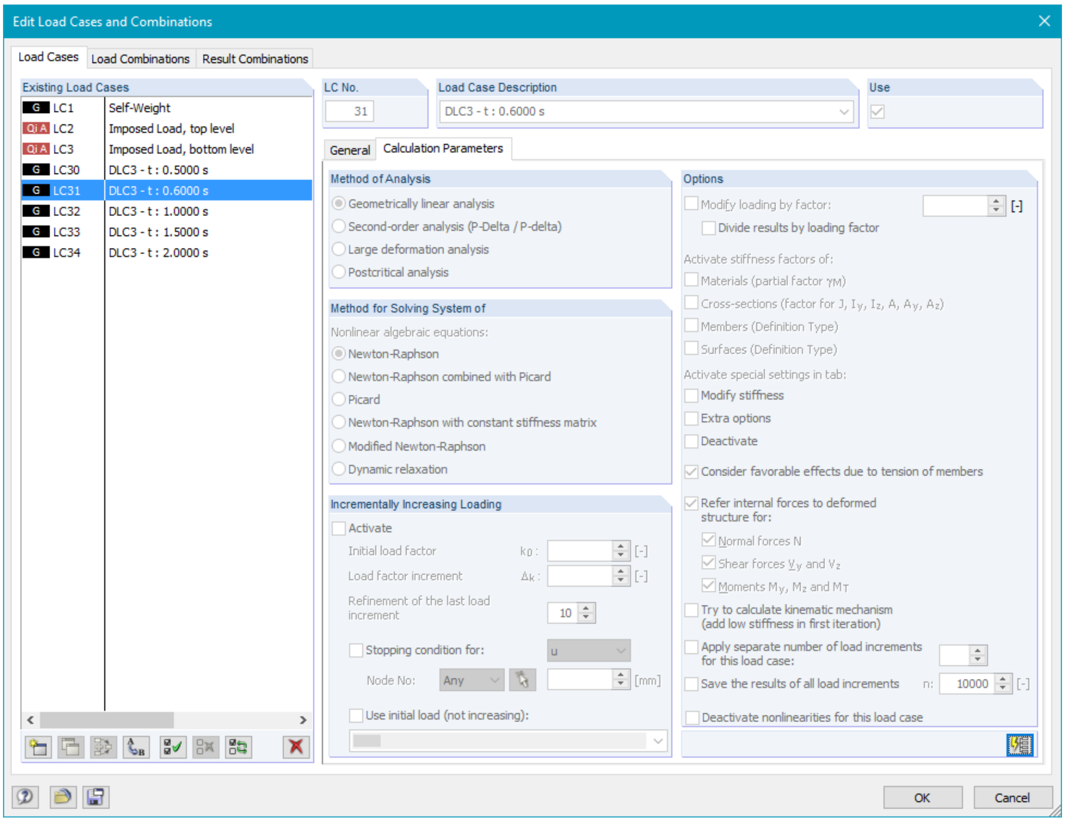


Figure 4.29: Dialogue box *Edit Load Cases and Combinations* with exported load cases *LC* from a time history analysis containing the results of single time steps

The exported *LC*s cannot be modified and the calculation parameters cannot be changed. Those load cases do not contain loads, they contain the results of the specific time step. They cannot be used in load combinations (*CO*), only in result combinations (*RC*).

## 4.3.8 Exported Result Combinations

When you have performed a time history analysis in *RF-DYNAM Pro - Forced Vibrations*, you can export an envelope of results over all time steps to a result combination (*RRC*). The *Edit Load Cases and Combinations* dialogue box is shown in Figure 4.30.

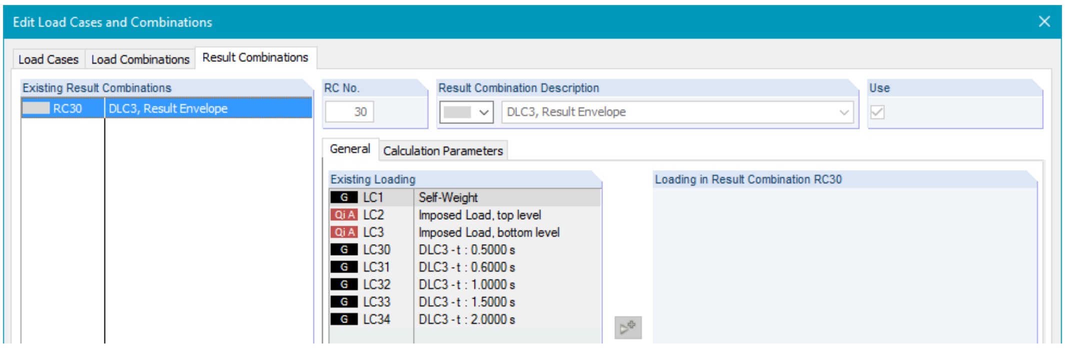


Figure 4.30: Dialogue box *Edit Load Cases and Combinations* with exported result combination *RC* containing the result envelope over all time steps from a time history analysis

The exported *RCs* cannot be modified and the calculation parameters cannot be changed.

## 4.3.9 Export to Excel or CSV

You can export all results tables that have been discussed in this section to MS Excel or save them in the *CSV* format using the **File** → **Export** option. The list of results tables is shown in Figure 4.31.

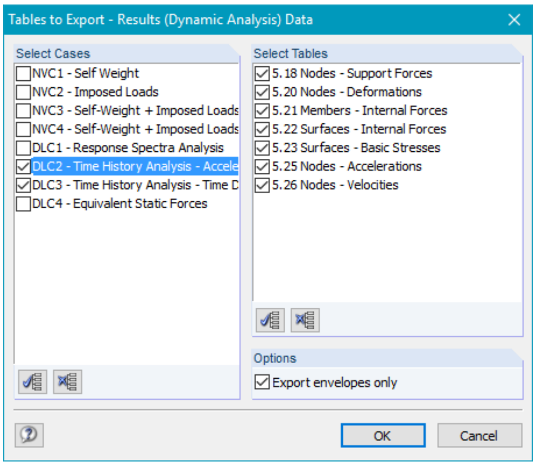
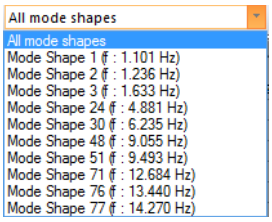



Figure 4.31: Results tables to export for dynamic load cases (*DLC*) with time history analysis

The *Export envelopes only* option is selected by default. Only the results tables that belong to the *Dynamic Envelopes* will be exported. When you clear this check box, the results tables of all time steps will be exported. Note that exporting can take considerably long when you export all time steps.

# 4.4 Dynamic Load Cases - Equivalent Load Analysis

The results tables that belong to equivalent load analysis cases are available when the corresponding dynamic load case (*DLC*) is chosen in the drop-down menu. Tables 5.8 to 5.10 belong to this type of dynamic load cases. The equivalent load analysis belongs to the module *RF-DYNAM Pro - Equivalent Loads*. The input data required for an equivalent load analysis is discussed in Section 2.5 and Section 2.11.




You can switch among the tables showing the equivalent loads for all eigenvalues of the system, or separately for each *Mode Shape*.

## 4.4.1 Equivalent Loads

For an equivalent load analysis, only the equivalent loads are calculated by RF-DYNAM Pro. The actual calculation has to be performed in the main program RFEM.

The equivalent loads $F_X$, $F_Y$, and $F_Z$ are calculated separately for each excitation direction of the response spectra, and separately for each mode shape of the system, as follows:

$$\begin{Bmatrix} F_X \\ F_Y \\ F_Z \end{Bmatrix} = \Gamma_X \cdot \begin{Bmatrix} u_X \\ u_Y \\ u_Z \end{Bmatrix} \cdot \mathbf{S}_{a,X}(T) \cdot \begin{Bmatrix} M_X \\ M_Y \\ M_Z \end{Bmatrix} \tag{4.9}$$

$$\begin{Bmatrix} F_X \\ F_Y \\ F_Z \end{Bmatrix} = \Gamma_Y \cdot \begin{Bmatrix} u_X \\ u_Y \\ u_Z \end{Bmatrix} \cdot \mathbf{S}_{a,Y}(T) \cdot \begin{Bmatrix} M_X \\ M_Y \\ M_Z \end{Bmatrix} \tag{4.10}$$

$$\begin{Bmatrix} F_X \\ F_Y \\ F_Z \end{Bmatrix} = \Gamma_Z \cdot \begin{Bmatrix} u_X \\ u_Y \\ u_Z \end{Bmatrix} \cdot \mathbf{S}_{a,Z}(T) \cdot \begin{Bmatrix} M_X \\ M_Y \\ M_Z \end{Bmatrix} \tag{4.11}$$

where

| | |
|---|---|
| $\Gamma$ | Participation factors in X, Y, and Z-direction as defined in Equation 4.4 using a modal mass of $M_i = 1$ kg |
| u | Displacement values in X, Y, and Z-direction of the mode shape scaled so that $M_i = \mathbf{u}_i^T \mathbf{M} \mathbf{u}_i = 1$ kg |
| $\mathbf{S}_a(T)$ | Acceleration read from the response spectra diagram using the natural period T of the considered eigenvalue |
| M | Mass in the direction X, Y, and Z at the considered FE-node |

With the formulas provided in Equation 4.9, the equivalent loads resulting from a response spectrum in X-direction are calculated. Accordingly, Equation 4.10 describes the loads resulting from a response spectrum in Y-direction, and Equation 4.11 the loads resulting from a response spectrum in Z-direction.

Those equivalent static forces exist at each FE-node within the structure, provided that the mass *M* or the mode shape *u* is not 0 at this point.

The results tables for the equivalent load analysis are separated into the three excitation directions. Table 5.8 gives the resulting equivalent loads for response spectra acting in X-direction, Table 5.9 those for response spectra acting in Y-direction, and Table 5.10 those for response spectra acting in Z-direction. Table 5.9 is illustrated in Figure 4.32.

When the equivalent loads are listed for only one eigenvalue of the system, the *sum* of equivalent loads is listed at the bottom of Tables 5.8, 5.9, and 5.10. It is represented by the formulas

$$\sum F_{X,i} = \left|\sum (F_{X,i})\right| \tag{4.12}$$

$$\sum F_{Y,i} = \left|\sum (F_{Y,i})\right| \tag{4.13}$$

$$\sum F_{Y,i} = \left|\sum (F_{Y,i})\right| \tag{4.14}$$

where *i* is the number of the considered eigenvalue. This sum of forces of each eigenvalue is the base shear force.

When the equivalent loads are listed for *All mode shapes* of the system in one table, the overall sum of the base shear forces from each eigenvalue is listed at the bottom of Tables 5.8, 5.9, and 5.10.

Table 5.9 is illustrated in Figure 4.32, listing the equivalent loads that belong to *Mode shape 1* and the base shear forces resulting from the excitation in Y-direction. Note that also the *LC* numbers are provided to which the equivalent loads are exported.

5.9 Equivalent Loads (Y-excitations)

DLC4 - Equivalent Static Forces | Mode Shape 1 (f : 1.304 Hz)

| | A | B | C | D | E | F | G | H | I | J |
|---|---|---|---|---|---|---|---|---|---|---|
| FE Mesh Point | Mode shape No. | LC No. | Object Type | Location X [m] | Y [m] | Z [m] | Equivalent Load Fx [N] | FY [N] | Fz [N] | Mz [Nm] |
| 294 | 1 | 100 | Surface | 2.500 | 0.500 | -4.000 | -0.2 | -268.9 | -0.6 | 188.25 |
| | 1 | 101 | Surface | 2.500 | 0.500 | -4.000 | -0.2 | -268.9 | -0.6 | -188.25 |
| 295 | 1 | 100 | Surface | 3.000 | 0.000 | -4.000 | -0.3 | -156.7 | 0.4 | 109.79 |
| | 1 | 101 | Surface | 3.000 | 0.000 | -4.000 | -0.3 | -156.7 | 0.4 | -109.79 |
| 296 | 1 | 100 | Surface | 3.000 | 0.500 | -4.000 | -0.1 | -268.8 | -0.9 | 188.20 |
| | 1 | 101 | Surface | 3.000 | 0.500 | -4.000 | -0.1 | -268.8 | -0.9 | -188.20 |
| 297 | 1 | 100 | Surface | 3.500 | 0.000 | -4.000 | -0.3 | -156.6 | 0.5 | 109.71 |
| | 1 | 101 | Surface | 3.500 | 0.000 | -4.000 | -0.3 | -156.6 | 0.5 | -109.71 |
| 298 | 1 | 100 | Surface | 3.500 | 0.500 | -4.000 | 0.2 | -268.8 | -1.7 | 188.23 |
| | 1 | 101 | Surface | 3.500 | 0.500 | -4.000 | 0.2 | -268.8 | -1.7 | -188.23 |
| 299 | 1 | 100 | Surface | 4.000 | 0.500 | -4.000 | 0.3 | -194.8 | -2.2 | 136.41 |
| | 1 | 101 | Surface | 4.000 | 0.500 | -4.000 | 0.3 | -194.8 | -2.2 | -136.41 |
| 300 | 1 | 100 | Surface | 0.500 | 1.000 | -4.000 | -0.7 | -268.9 | -2.7 | 188.43 |
| | 1 | 101 | Surface | 0.500 | 1.000 | -4.000 | -0.7 | -268.9 | -2.7 | -188.43 |
| 301 | 1 | 100 | Surface | 0.000 | 1.000 | -4.000 | -0.6 | -194.7 | -2.9 | 136.44 |
| | 1 | 101 | Surface | 0.000 | 1.000 | -4.000 | -0.6 | -194.7 | -2.9 | -136.44 |
| 302 | 1 | 100 | Surface | 1.000 | 1.000 | -4.000 | -0.5 | -268.9 | -1.8 | 188.37 |
| | 1 | 101 | Surface | 1.000 | 1.000 | -4.000 | -0.5 | -268.9 | -1.8 | -188.37 |
| sum | | | | | | | 0.1 | 119769.8 | 0.0 | |

Equivalent Loads (X-excitations) | Equivalent Loads (Y-excitations) | Equivalent Loads (Z-excitations)

Figure 4.32: Table *5.9 Equivalent Loads (Y-excitations)* of mode shape 1 resulting from response spectrum in Y-direction

## 4.4.2 Torsional Moments

Accidental torsional actions can be considered as described in Section 2.11.2 to account for uncertainties in the location of masses. The centre of mass is considered to be displaced by the accidental eccentricities $e_X$ and $e_Y$. The torsional moments are calculated as follows:

$$M_Z = |F_X \cdot e_Y| + |F_Y \cdot e_X| \tag{4.15}$$

where $M_Z$ is the torsional moment and $F_X$ and $F_Y$ are the equivalent loads on each FE node as defined in Equations 4.9, 4.10, and 4.11. The torsional moments $M_Z$ are considered in both positive and negative directions.

## 4.4.3 Exported Load Cases

The equivalent loads as listed in the dynamic results tables are exported in load cases of the main program RFEM. This is done separately for each eigenvalue and for each excitation direction. The list of generated load cases can be viewed in the *Edit Load Cases and Combinations* dialogue box as shown in Figure 4.33. Here, the accidental torsional actions (described in Section 2.11.2) are activated and two load cases for each mode and direction are generated. The torsional moments are considered in the positive and negative directions.

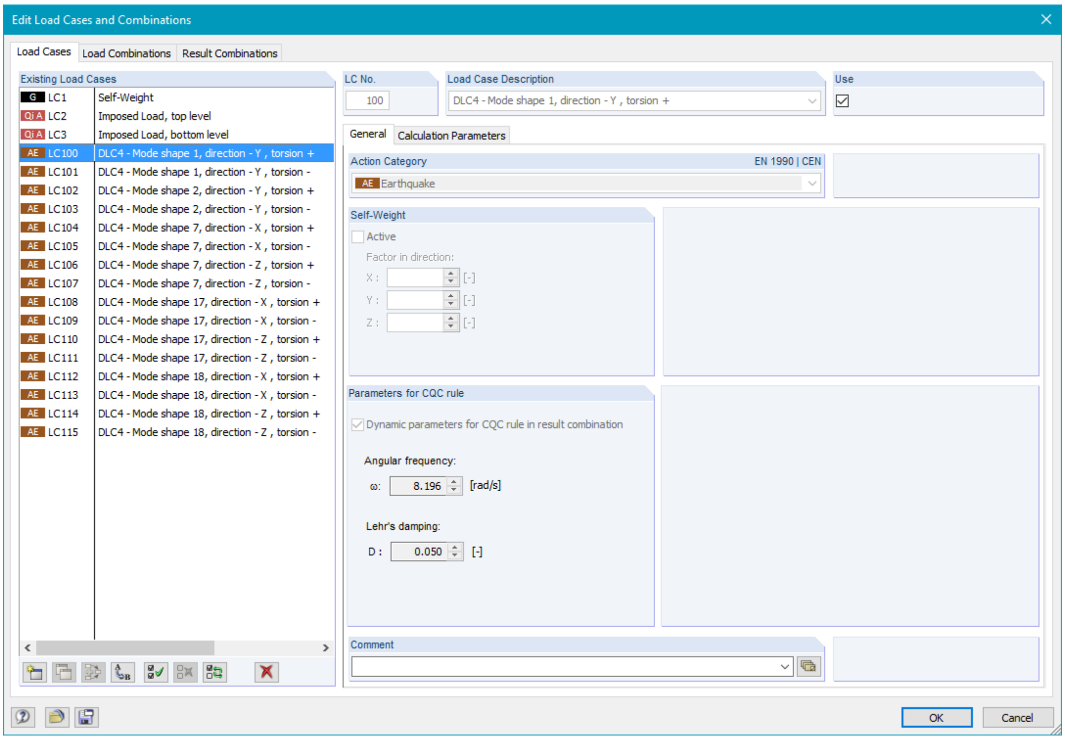


Figure 4.33: Dialogue box *Edit Load Cases and Combinations* with exported load cases *LC* from equivalent load analysis showing two load cases for each mode and direction due to accidental torsion

The *Angular frequency* $\omega$ and *Lehr's damping* value *D* are also exported as *LC* properties from RF-DYNAM Pro to enable the *CQC* modal combination. If the *SRSS* rule was set in RF-DYNAM Pro, the exported damping value $D = 0$.

You can adjust the *Calculation Parameters* to your needs in the second sub-tab. Those parameters are shown in Figure 4.34.

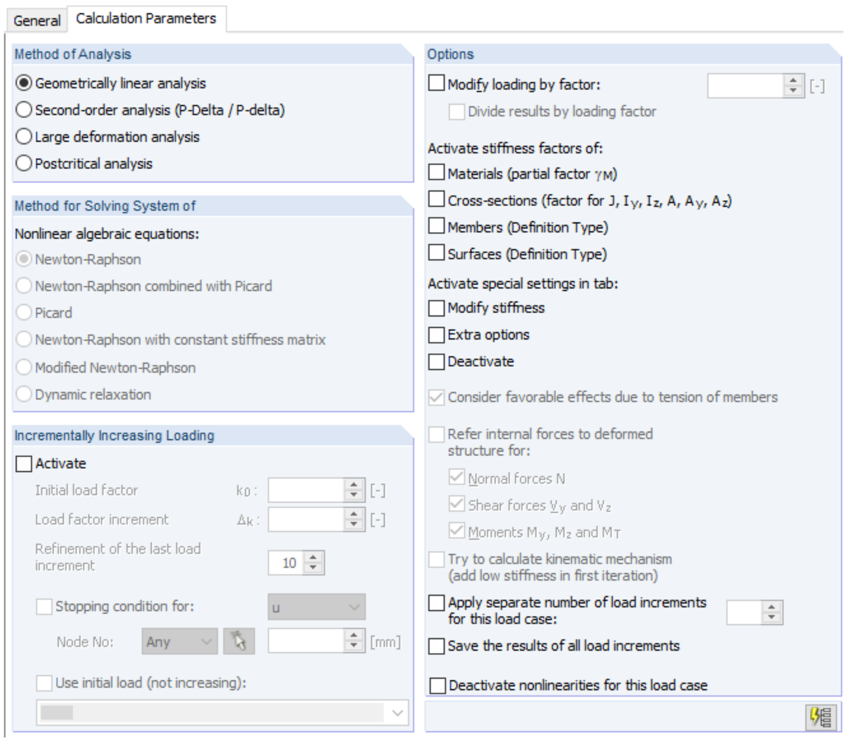


Figure 4.34: Calculation parameters of exported load cases from equivalent load analysis

Changing the calculation parameters may be necessary when

- the structure has a dominant mode shape and the analysis shall consider the P-$\Delta$ effect,
- stiffness modifications were applied in the natural vibration cases (see Section 2.4.7) and the *LC* shall use same modifications for consistency reasons (FAQ 002237 provides an example),
- non-linearities shall be deactivated in the load cases (equivalent loads are based on linearly calculated eigenvalues and mode shapes, thus *LC*s should not consider non-linearities).

There might be a conflict of non-linearly calculated load cases and result combinations that are applied in a dynamic analysis. Please consult the Section 5.5 of the RFEM manual to understand the important difference between load combinations (*CO*) and result combinations (*RC*). Due to the quadratic combination rules, only result combinations are possible in a dynamic analysis.

The equivalent loads can be displayed in the work window by pressing the [Show Loads] button. If more than 5,000 loads are generated, the display is deactivated by default. But in the *Details* dialogue box, the number of displayed equivalent loads can be changed (see Section 2.12).

### 4.4.4 Exported Result Combinations

When performing an equivalent load analysis in the add-on module *RF-DYNAM Pro - Equivalent Loads*, result combinations are produced in two steps: (1) The modal responses are combined with the *SRSS* or *CQC* rule and are exported in result combinations (*RC*) separately for each excitation direction. (2) The directional results are combined either with the *SRSS* or with the 100% / 30% rule and the final result combinations (*RC*) are exported as well.

A list of generated result combinations can be found in the *Edit Load Cases and Combinations* dialogue box displayed in Figure 4.35. In this figure, the modal combination for the Z-direction is shown.

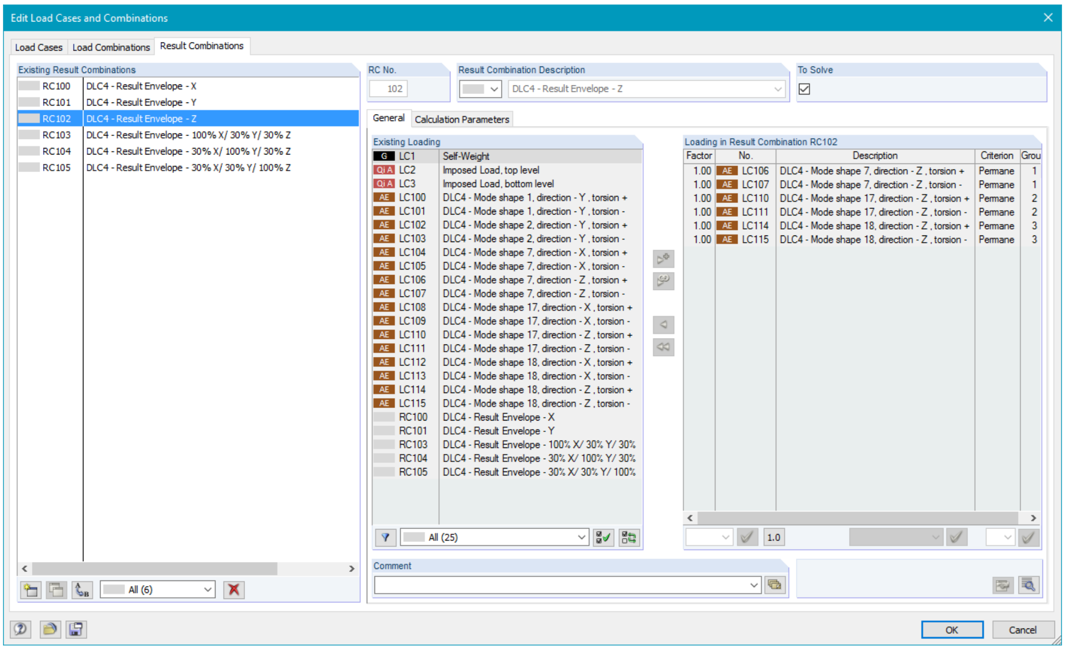



Figure 4.35: Dialogue box *Edit Load Cases and Combinations* with exported result combinations from the equivalent load analysis showing *RC* for modal combination in Z-direction

Dialogue box section *Loading in Result Combination RC102* in Figure 4.35 lists the load cases (*LC*) that contain the equivalent loads relevant in the considered excitation direction. Here, the accidental torsional actions (described in Section 2.11.2) are activated, so two load cases for each mode and direction exist. They are combined with the *OR* criterion before the *SRSS* or *CQC* rule is applied.

The load cases are combined with the *CQC* rule according to the *equivalent linear combination* (see Figure 4.36). For the *Extreme value signs*, the *LC* of the dominant mode shape is set in the drop-down menu. Those calculation parameters are automatically generated by RF-DYNAM Pro depending on the settings you have made in the add-on module. The calculation parameters of the result combination (*RC*) are shown in Figure 4.36.

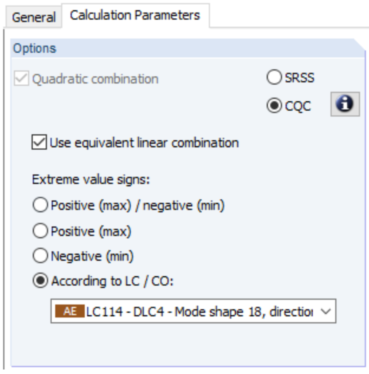



Figure 4.36: Calculation parameters of exported *RC* for modal combination with *CQC* rule as equivalent linear combination applied and signs of *LC* corresponding to dominant mode shape set automatically

The calculation parameters of the result combinations can be changed in RFEM independently of the settings in RF-DYNAM Pro.

In the second step of combination, the responses resulting from different excitation directions are combined. One of the final result combinations (*RC*) is selected in Figure 4.37. Note the factors *1.00* and *0.30* that have been applied to account for the 100% / 30% rule.

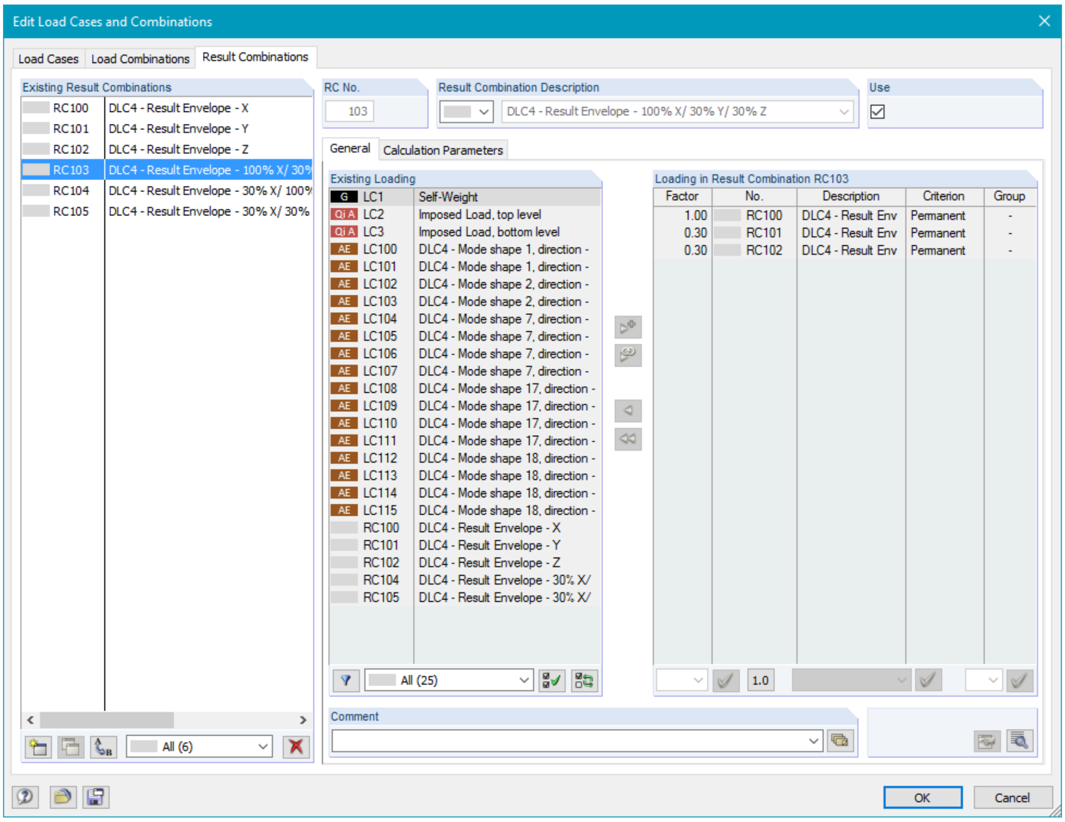



Figure 4.37: Dialogue box *Edit Load Cases and Combinations* with exported result combinations from the equivalent load analysis showing *RC* of directional combination with 100% / 30% rule

### 4.4.5 Export to Excel or CSV

You can export all results tables that have been discussed in this section to MS Excel or save them in the *CSV* format using the **File** → **Export** option. The list of results tables is shown in Figure 4.38.

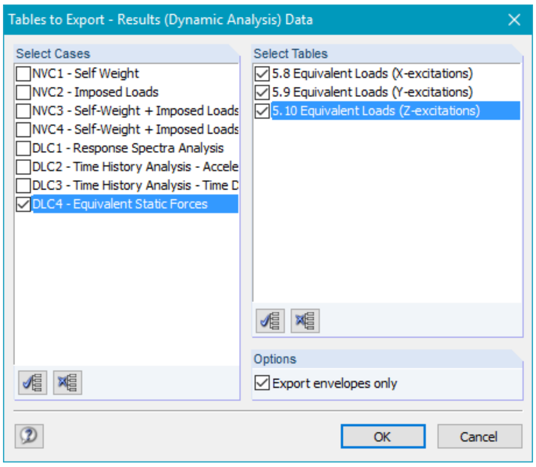



Figure 4.38: Results tables to export for dynamic load cases (*DLC*) with equivalent load analysis

The *Export envelopes only* option is selected by default. Only the results tables containing the equivalent loads of all mode shapes will be exported. When you clear this check box, the tables that belong to each single mode shape will be exported as well.

## 4.5 Automatic Combinations in RFEM

In RFEM, you have the option to create load combinations (*CO*) and result combinations (*RC*) automatically. This is described in Chapter 5 of the RFEM manual.

Note that those automatic combinations are not yet supported for the load cases and result combinations imported from RF-DYNAM Pro. You can still use the automatic combination feature for all the other load cases that you want to combine (*i.e* of action self-weight or imposed loads).

In RFEM, you activate the automatic combination feature in the *General Data* as seen in Figure 4.39.

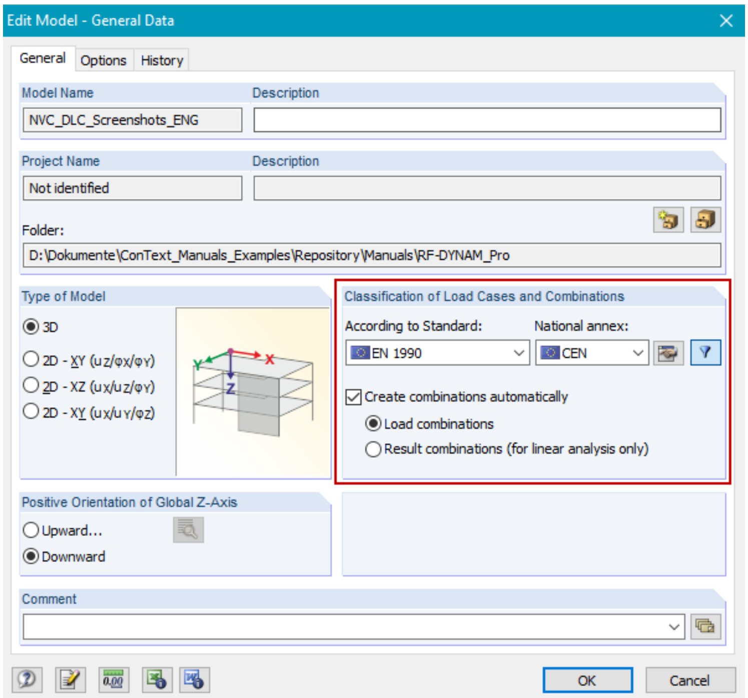



Figure 4.39: Activation of automatic combination feature in *General Data* dialogue box of RFEM

When you have activated the automatic combinations before you calculate any of the RF-DYNAM Pro cases, the *LC*s and *RC*s imported from RF-DYNAM Pro will not be included in any of the *Actions*, *i.e.* no action of the *Earthquake* category will exist. Consequently, the dynamic *LC*s and *RC*s will not influence the load and result combinations that are created automatically.

Figure 4.40 shows the *Action* tab of the *Edit Load Cases and Combination* dialogue box. The *LC*s imported from *RF-DYNAM Pro - Forced Vibrations* are not included in the *Permanent* action *A1*, although the *Permanent* action category is allocated.

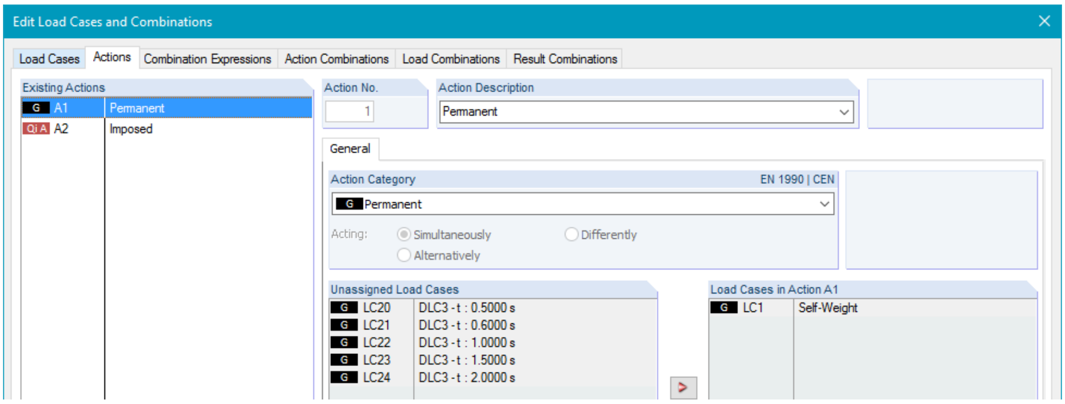



Figure 4.40: *Actions* tab of *Edit Load Cases and Combinations* dialogue box when automatic combinations were activated before the RF-DYNAM Pro calculation – RF-DYNAM Pro results are available, but imported *LC*s are not included in *Existing Actions* and no action of type *Earthquake* is available

When you switch on the *Create combinations automatically* option after you have calculated the results in RF-DYNAM Pro and exported the *LC*s and *RC*s, make sure you keep the user-defined result combinations as indicated in Figure 4.41.

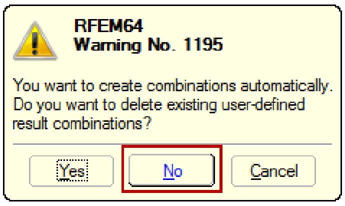



Figure 4.41: RFEM warning when activating automatic combination feature – make sure to keep existing *RC*s if RF-DYNAM Pro results are available

In this order, the *Actions* will take the *LC*s from RF-DYNAM Pro into account, and an action of the *Earthquake* category will be generated. This is illustrated in Figure 4.42.

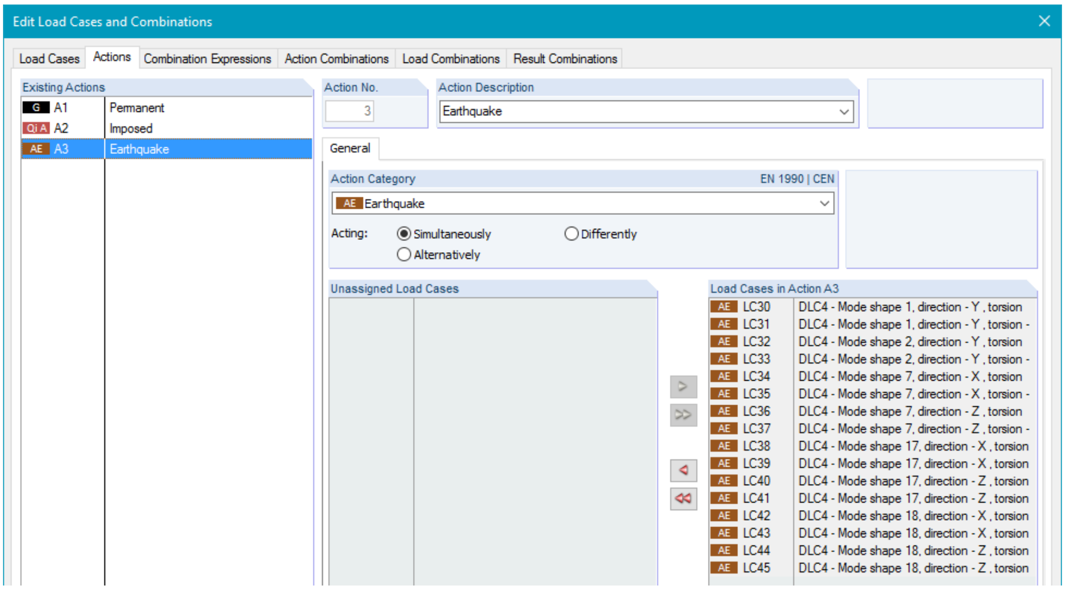



Figure 4.42: *Actions* tab of *Edit Load Cases and Combinations* dialogue box when automatic combinations were activated after the RF-DYNAM Pro calculation – *Earthquake* action exists with *LC*s imported from RF-DYNAM Pro included

Nevertheless, this does not influence the automatic combinations, which means that no *Action Combinations*, *Load Combinations* or *Result Combinations* will be created for the design situation *Earthquake*. Load cases that have been imported from a time history analysis are included in load combinations, though, but as they do not contain any loads (they contain only results of a specific time step as described in Section 4.3), they do not have an influence on the results.

The result combination for the design situation *Earthquake* must be defined manually. Creating *Load Combinations* is not sufficient, due to the quadratic combination rules that are requested for a response spectrum analysis. For the time history analysis, *Load Combinations* are not suited either because the exported *LC*s are only for single time steps and do not contain loads.

In EN 1990 [4] Section 6.4.3.4, Equation (6.12) provides the combination expression for the design situation *Earthquake* as follows:

$$E_d = \sum_{j \geq 1} G_{k,j} \text{ “+” } P \text{ “+” } A_{Ed} \text{ “+” } \sum \Psi_{2,i} Q_{k,i} \tag{4.16}$$

where $A_{Ed}$ shall be the resulting *RC*s from RF-DYNAM Pro. The combination factors $\Psi_2$ are listed in EN 1990 [4] Table A.1.1.

An example how this is done for a response spectrum analysis can be found in Section 5.1.

## 4.6 Printout Report

You can create a printout report containing the dynamic results. Detailed information on the printout report can be found in Chapter 10 of the RFEM manual.

When dynamic results are available, you can include the *Input Data* and the *Natural Vibration Case* data in the *Printout Report Selection* dialogue box as shown in Figure 4.43 and Figure 4.44.

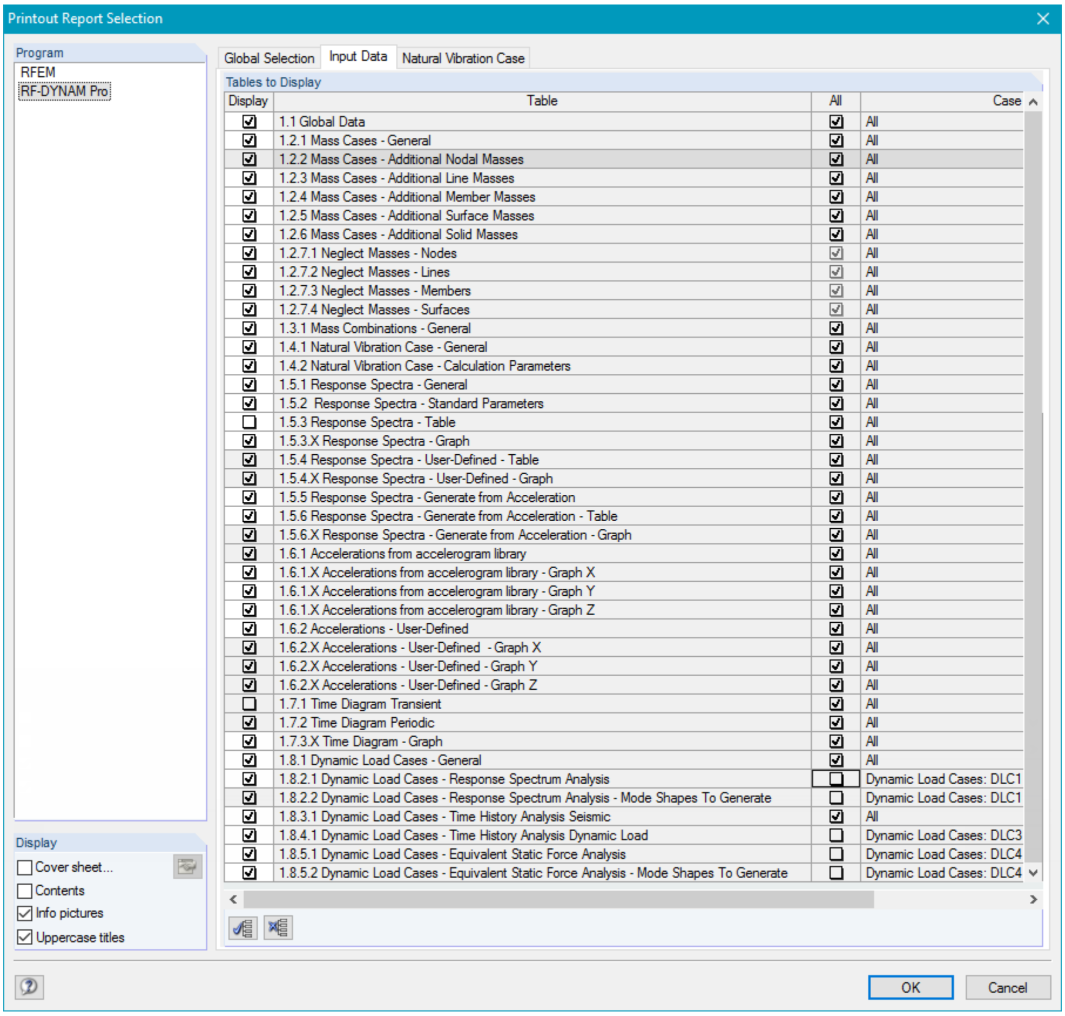



Figure 4.43: Dialogue box *Printout Report Selection* with *Input Data* available for RF-DYNAM Pro case

The *Mode Shapes* table that represent the selection of mode shapes and corresponding values read from the response spectrum can be included by selecting Table *1.8.2.2* and/or *1.8.5.2*. This table was discussed in Section 2.9.7 and is relevant for a response spectrum analysis and an equivalent load analysis.

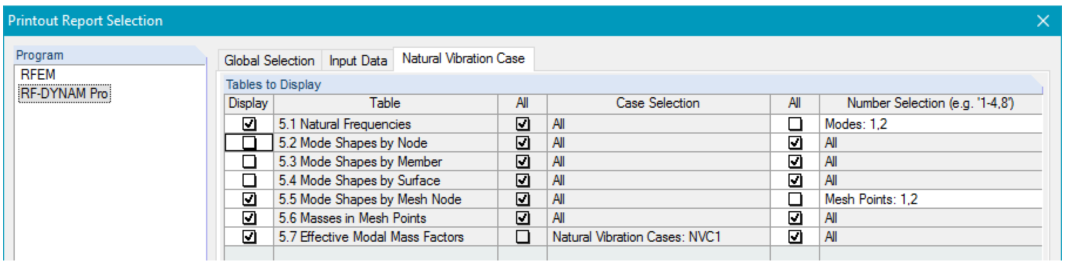



Figure 4.44: Dialogue box *Printout Report Selection* with *Natural Vibration Case* data available

Each of the entries can be selected for all cases (*NVCs* or *DLCs*). When you clear the *All* check box, you can select specific cases.

All other RF-DYNAM Pro results are available in exported load cases or result combinations. The printout report for those results can be adjusted as described in Chapter 10 of the RFEM manual.

## 4.7 Units and Decimal Places

You can access the *Units and Decimal Places* dialogue box with the button shown on the left. The dialogue box with the *Results* tab of the RF-DYNAM Pro add-on module open is shown in Figure 4.45.

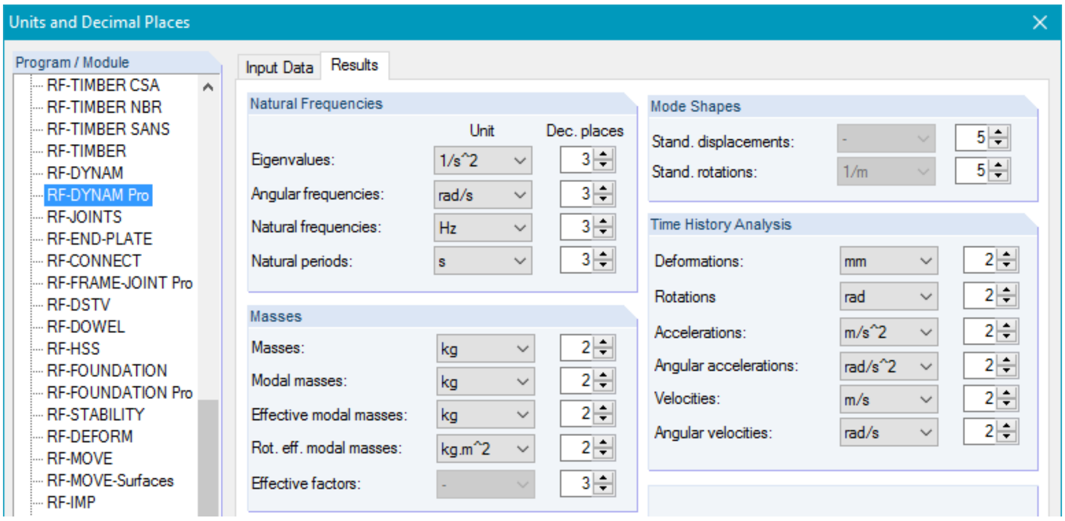



Figure 4.45: Dialogue box *Units and Decimal Places* showing settings of RF-DYNAM Pro *Results* data

Select the add-on module *RF-DYNAM Pro* in the list of modules. In the *Results* tab, you can choose the units from the drop-down menus and adjust the decimal places.

As the RF-DYNAM Pro results are embedded in the main program RFEM, most of the result values can be adjusted in the RFEM list of units and decimal places as shown in Figure 4.46.

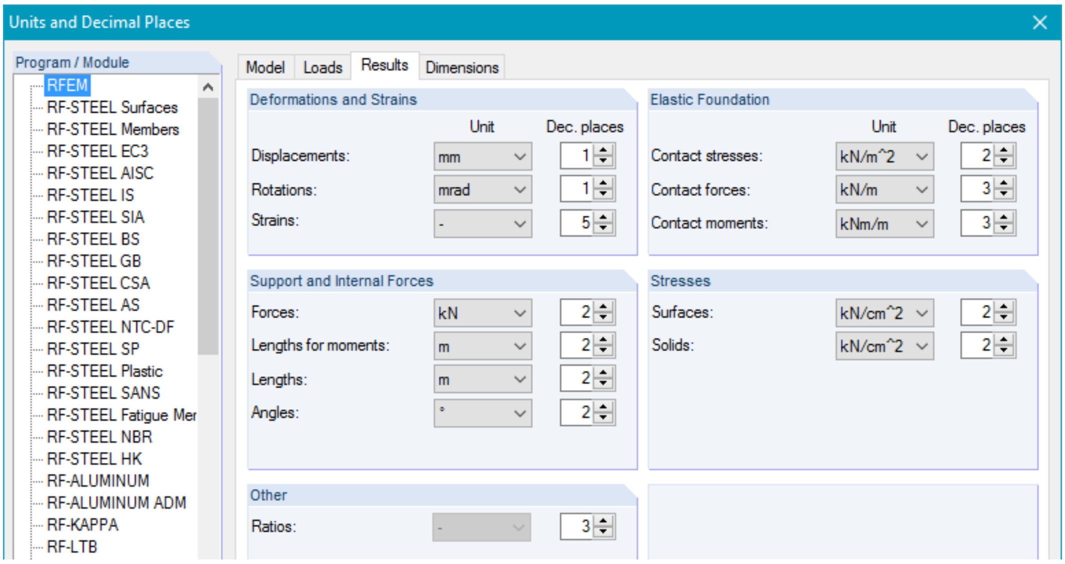



Figure 4.46: Dialogue box *Units and Decimal Places* showing settings for RFEM *Results*

# 5 Examples

An example how to perform a natural vibration analysis and a multi-modal response spectra analysis using the module *RF-DYNAM Pro - Equivalent Loads* is shown in the **webinar** Natural Frequencies and Equivalent Static Force Analysis with RFEM.

The simulation of walking and running across a pedestrian bridge demonstrating a complex time history analysis using the module *RF-DYNAM Pro - Forced Vibrations* is presented in the **webinar** Time History Analysis in RF-/DYNAM Pro - Walking and Running Across Pedestrian Bridge. It is shown how time diagrams can be defined via functions employing parameters that are defined in the main program RFEM. Also used are multiple load-time diagram sets.

In the **webinar** Nonlinear Time History Analysis – Machine-Induced Vibrations, induced vibrations on a structure with nonlinear tension members are analysed with the nonlinear time history solvers.

More examples are available on our website that demonstrate the accuracy of the dynamic modules. The **verification example** of a Cantilever Beam (SDOF) with Periodic Excitation studies the response of an SDOF system. There the results of *RF-DYNAM Pro - Forced Vibrations* are compared with the analytical solution. The **verification example** of Equivalent Loads compares the equivalent loads calculated by *RF-DYNAM Pro - Equivalent Loads* with the analytical solution. Considered is a 5-DOF system with masses acting in X-direction.

In the following, Section 5.1 presents an example how an earthquake analysis is performed with *RF-DYNAM Pro - Equivalent Loads*. The base shear forces in each floor are evaluated and the storey drift is calculated. In Section 5.2, machine-induced vibrations are analysed with the module *RF-DYNAM Pro - Forced Vibrations*. The case of resonance is studied and the structure is modified to avoid resonance.

## 5.1 Earthquake Analysis according to EN 1998-1 CEN

This example presents an earthquake analysis according to EN 1998-1 CEN using the multi-modal response spectrum analysis by means of the **RF-DYNAM Pro - Equivalent Loads** module.

### 5.1.1 System and Load Cases in RFEM

The structural system is shown in Figure 5.1. The materials and cross-sections are used as illustrated.

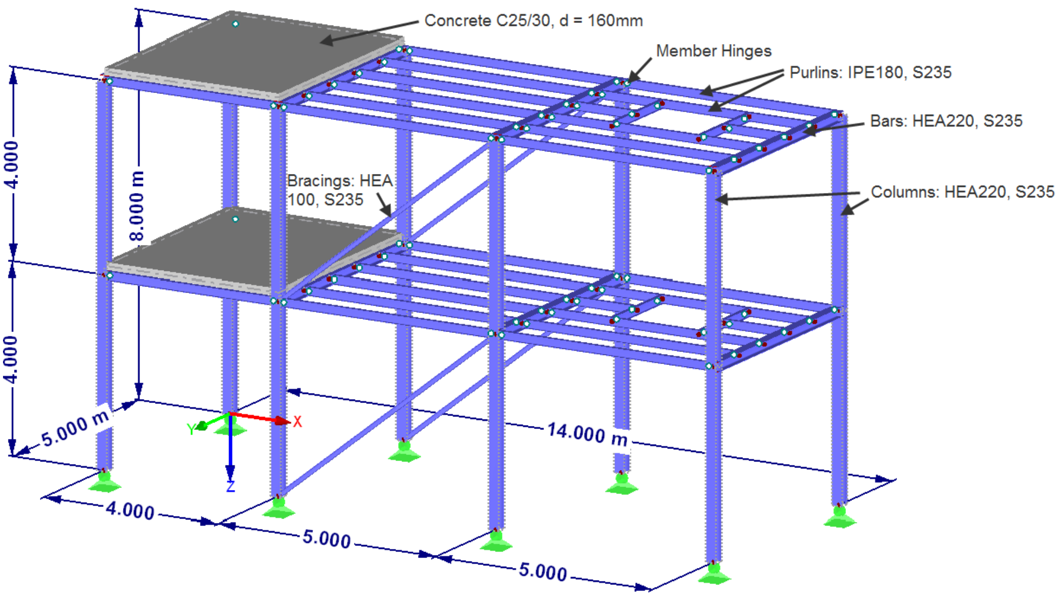



Figure 5.1: Structural system with dimensions, materials and cross-sections

Three load cases are defined with self-weight and imposed loads. Those load cases are imported as masses in RF-DYNAM Pro. *LC1* defines the self-weight in $+$Z direction and 0.5 kN/m$^2$. *LC2* and *LC3* contain 2.5 kN/m$^2$ imposed load acting in the top and bottom level of the building, respectively.

In the *General* tab of RF-DYNAM Pro, the *Mass combinations* and the *Response spectrum analysis with generation of equivalent loads* options are activated as shown in Figure 5.2.

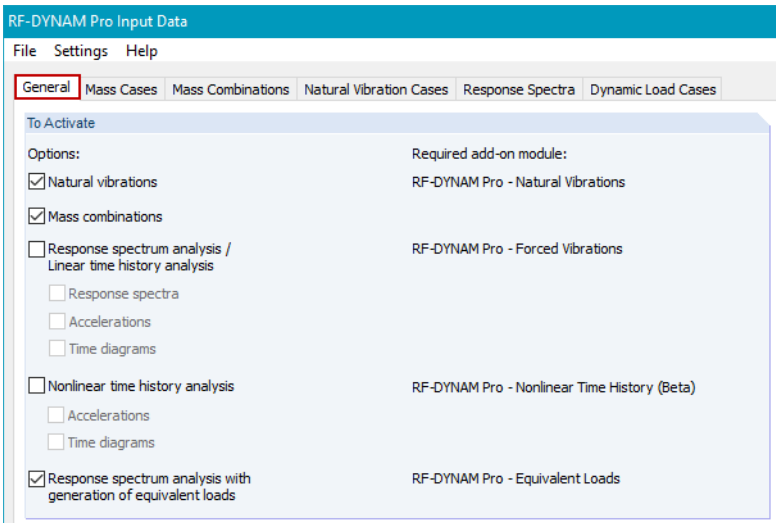



Figure 5.2: *General* settings for equivalent load analysis

## Definition of Masses

Apart from the self-weight, additional masses have to be considered to calculate inertia effects. This is regulated in EN 1998-1 [1] Section 3.2.4.

$$\sum G_{k,j} \text{``}+\text{''} \sum \psi_{E,i} \cdot Q_{k,i} \tag{5.1}$$

where $G_{k,j}$ are the permanent loads and $Q_{k,i}$ any imposed load. $\psi_{E,i}$ are the combination factors for the imposed loads defined as

$$\psi_{E,i} = \varphi \cdot \psi_{2,i} \tag{5.2}$$

where $\psi_{2,i}$ are combination factors regulated in EN 1990 [4] Table A.1.1. Assuming that the building of this example is of category B (building with offices), $\psi_2 = 0.3$. The values of $\varphi$ recommended in EN 1998-1 CEN [1] are $\varphi = 1.0$ for the top level of a building and $\varphi = 0.5$ for all other levels. To account for the different values of $\varphi$, the load cases with imposed loads have been split in *LC2* and *LC3*.

In RF-DYNAM Pro, *Mass Cases* are defined to import masses from the load cases of RFEM. The mass cases *MC1* to *MC3* are shown in Figure 5.3 and Figure 5.4.

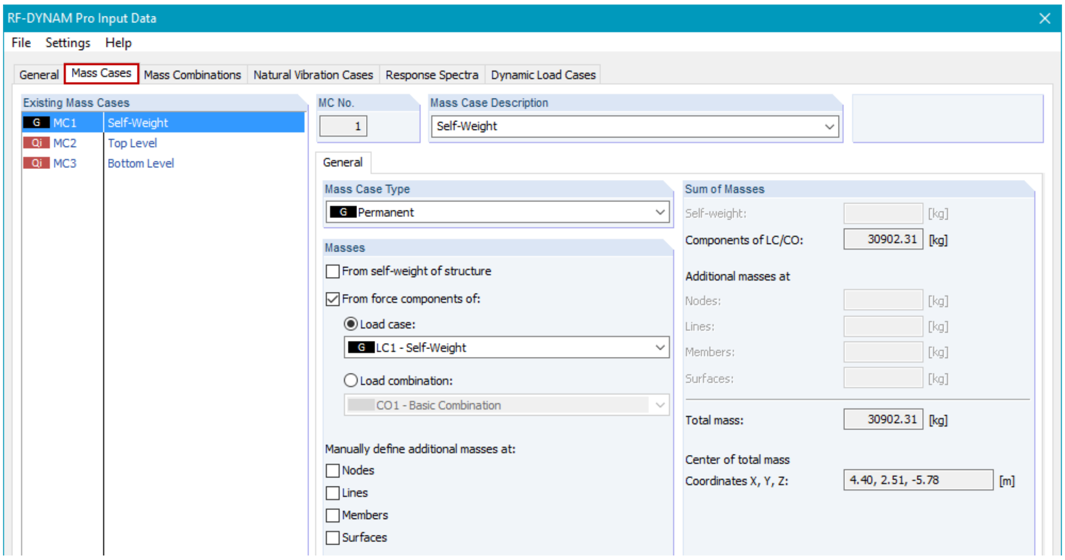



Figure 5.3: Mass case *MC1* to import self-weight from *LC1* (*From self-weight of structure* option not selected as self-weight is activated in *LC1*)

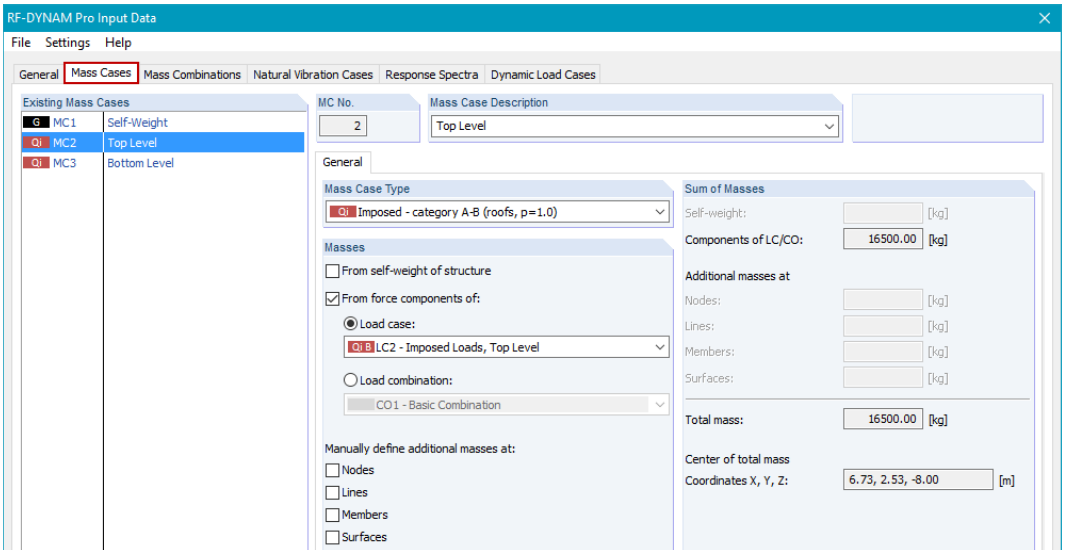



Figure 5.4: Mass case *MC2* to import imposed loads from *LC2* acting in top floor

Mass case *MC3* is done in analogy to *MC2*.

The mass cases are combined in a *Mass Combination* shown in Figure 5.5. The combination factors $\psi_{E,i} = \varphi \cdot \psi_{2,i}$ are applied in accordance with EN 1998-1 CEN.

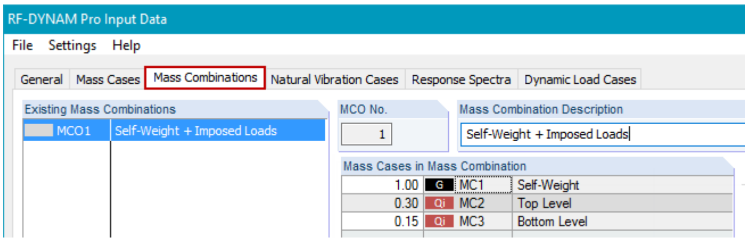



Figure 5.5: Mass combination *MCO1* combining masses in compliance with EN 1998-1

## 5.1.2 Seismic Analysis in RF-DYNAM Pro

### Natural Vibration Analysis

In this example, a three-dimensional natural vibration analysis is performed. The masses are acting in the X- and Y-directions. A spatial model is analysed. Simplying the structure to a planar model (that means that masses are considered to act either in X- or Y-direction) is only allowed for regular structures in accordance with EN 1998-1 [1] Section 4.2.3.

The Z-direction is neglected in this example as the vertical response spectrum is lower than 2.5 $\mathrm{m/s^2}$ (see EN 1998-1 [1] Section 4.3.3.5.2).

The diagonal matrix (translational DOF) concentrates masses in each FE-node; rotational masses are neglected. This matrix is of sufficient accuracy in this example.

The diagonal matrix (translational and rotational DOFs) would consider the rotational masses. The consistent matrix would distribute the masses in accordance with finite element shape functions and therefore increase the accuracy. But depending on the application, this is not always useful as much more local mode shapes are identified that are not of relevance for the global earthquake design of the structure.

The *Lanczos* solver is used for the analysis. This solver is recommended for most structures. The mode shapes are scaled to the maximum value of *1*. Stiffness modifications are not considered.

The settings of natural vibration case *NVC1* are illustrated in Figure 5.6.

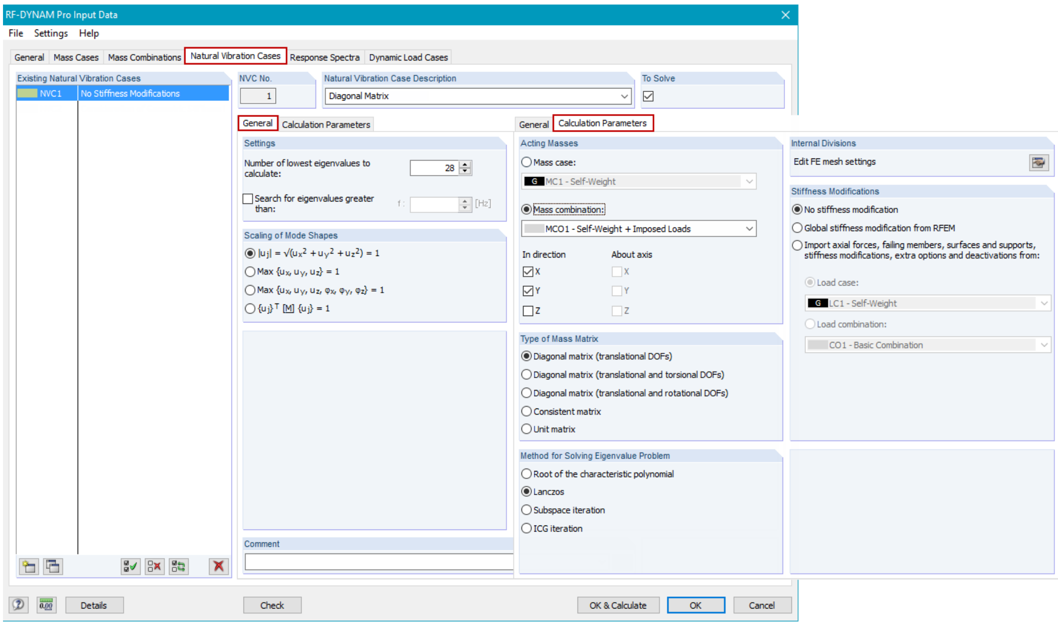



Figure 5.6: *NVC1* defining mass matrix, number of eigenvalues, solver, and scaling of mode shapes

The natural frequencies *f*, the natural periods *T*, and the corresponding effective modal mass factors $f_{me}$ are shown in Figure 5.7. In accordance with EN 1998-1 [1] Section 4.3.3.3, the sum of the effective modal masses must be greater than 90% of the total seismic load. The first three eigenvalues are relevant in the Y-direction, and only eigenvalue 28 is relevant in the X-direction.

| Mode No. | Frequency f [Hz] | Period T [s] | Effective Modal Mass Factor [-] f meX [kg] | f meY [kg] | f meZ [kg] |
|---|---|---|---|---|---|
| 1 | 1.201 | 0.833 | 0.000 | 0.892 | 0.000 |
| 2 | 1.394 | 0.718 | 0.000 | 0.035 | 0.000 |
| 3 | 1.944 | 0.514 | 0.000 | 0.031 | 0.000 |
| 28 | 5.280 | 0.189 | 0.917 | 0.000 | 0.000 |

Figure 5.7: Relevant natural frequencies *f [Hz]*, natural periods *T [s]* and corresponding effective modal mass factors $f_{me}$ $[-]$

The listed eigenvalues are considered in the multi-modal response spectrum analysis.

The dominant mode shape in Y-direction is illustrated in Figure 5.8, the dominant mode shape in X-direction in Figure 5.9.

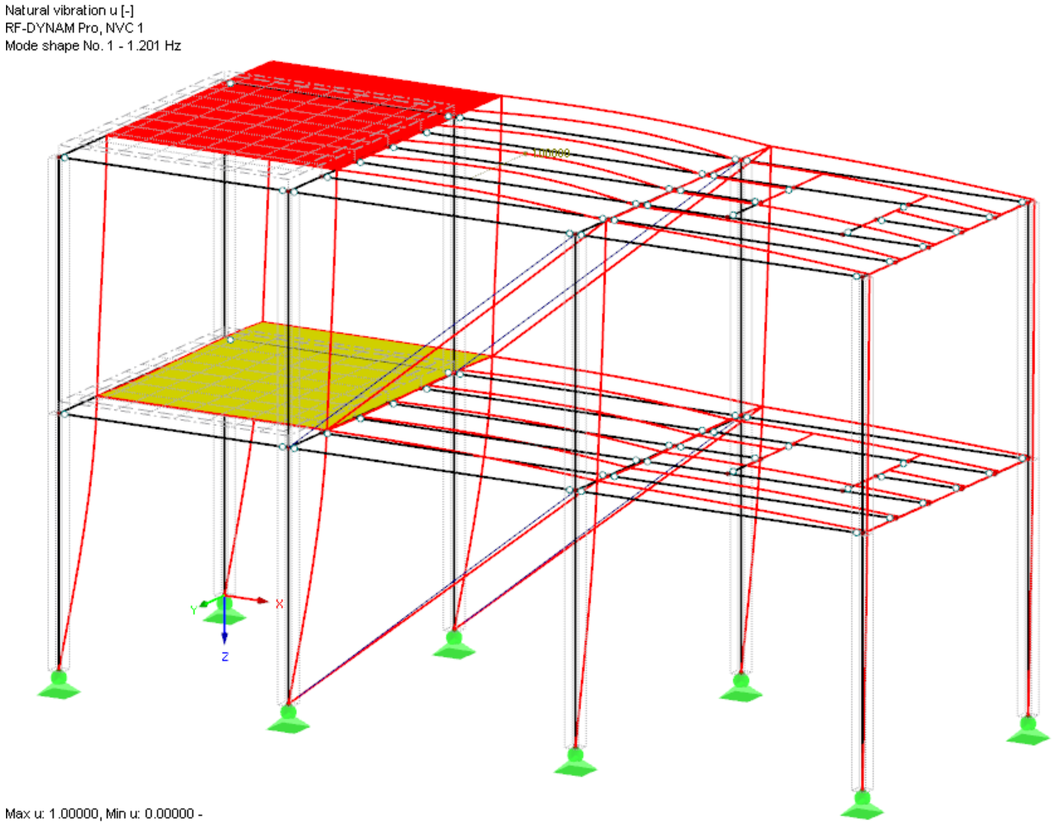



Figure 5.8: Dominant mode shape in Y-direction with natural frequency of 1.201 Hz

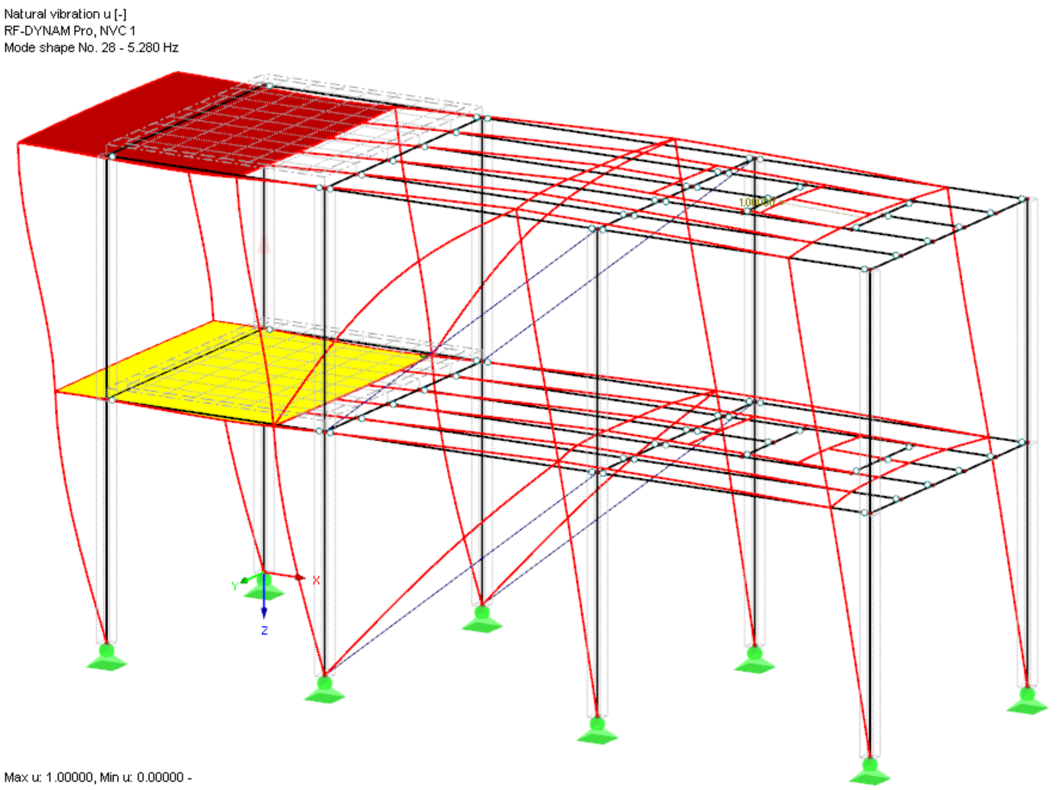



Figure 5.9: Dominant mode shape in X-direction with natural frequency of 5.28 Hz

## Response Spectrum in Accordance with EN 1998-1 CEN

The horizontal design response spectrum in accordance with EN 1998-1 [1] Section 3.2.2.5 is defined as:

$$0 \leq T \leq T_B \quad : S_d(T) = a_g \cdot S \cdot \left[\frac{2}{3} + \frac{T}{T_B} \cdot \left(\frac{2.5}{q} - \frac{2}{3}\right)\right] \tag{5.3}$$

$$T_B \leq T \leq T_C : S_d(T) = a_g \cdot S \cdot \frac{2.5}{q} \tag{5.4}$$

$$T_C \leq T \leq T_D : S_d(T) \begin{cases} = a_g \cdot S \cdot \frac{2.5}{q} \cdot \frac{T_C}{T} \\ \geq \beta \cdot a_g \end{cases} \tag{5.5}$$

$$T_D \leq T \quad\quad : S_d(T) \begin{cases} = a_g \cdot S \cdot \frac{2.5}{q} \cdot \frac{T_C T_D}{T^2} \\ \geq \beta \cdot a_g \end{cases} \tag{5.6}$$

where

| | | |
|---|---|---|
| T | : | Vibration period in [s] |
| $a_g$ | : | Design ground acceleration, $a_g = \gamma_1 \cdot a_{gR}$, in $[m/s^2]$ |
| $a_{gR}$ | : | Reference ground acceleration in $[m/s^2]$ |
| $\gamma_1$ | : | Importance factor |
| $T_B$ | : | Lower limit of the period of the constant spectral acceleration branch, in [s] |
| $T_C$ | : | Upper limit of the period of the constant spectral acceleration branch, in [s] |
| $T_D$ | : | Value defining the beginning of the constant displacement response range of the spectrum, in [s] |
| S | : | Soil factor |
| $S_d(T)$ | : | Acceleration values of the design spectrum in $[m/s^2]$ |
| q | : | Behaviour factor that describes the ductility |
| $\beta$ | : | Lower bound factor for the horizontal design spectrum |

In the example, the reference ground acceleration of $a_{gR} = 1.0$ and importance factor of $\gamma_1 = 1.0$ (importance category II) are assumed. The parameters $S$, $T_B$, $T_C$, and $T_D$ are chosen in accordance with ground type C and spectra type 1. The behaviour factor $q$ is kept with 1 on the safe side, and the lower bound factor $\beta = 0.2$ is kept as recommended in the standard.

The settings in RF-DYNAM Pro and the resulting horizontal and vertical response spectra are illustrated in Figure 5.10 and Figure 5.11.

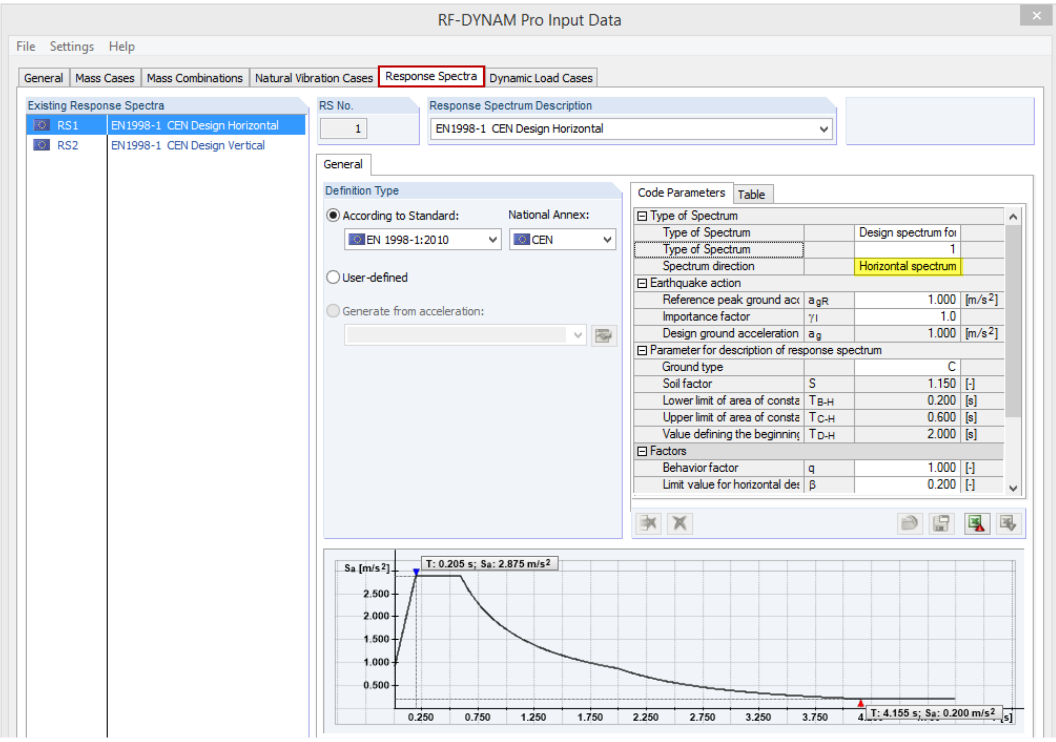



Figure 5.10: Horizontal design response spectrum in accordance with EN 1998-1 Section 3.2.2.5

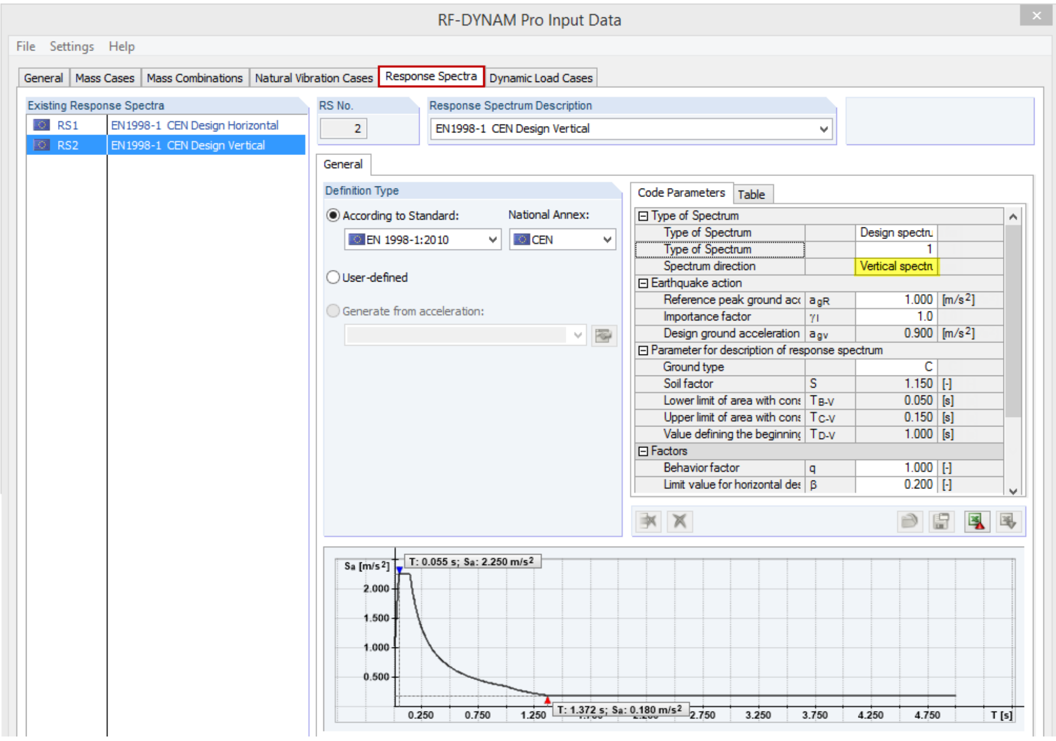



Figure 5.11: Vertical design response spectrum in accordance with EN 1998-1 Section 3.2.2.5 – peak value $S_d(T) \leq 2.5\ \mathrm{m/s^2}$ so that vertical direction can be neglected (see EN 1998-1 Section 4.3.3.5.2)

## Response Spectra Analysis with Equivalent Loads

In the **RF-DYNAM Pro - Equivalent Loads** module, a multi-modal response spectrum analysis is performed. Then equivalent loads are exported to RFEM separately for each eigenvalue and excitation direction. The settings are done in the *Dynamic Load Cases* tab as illustrated in Figure 5.12.

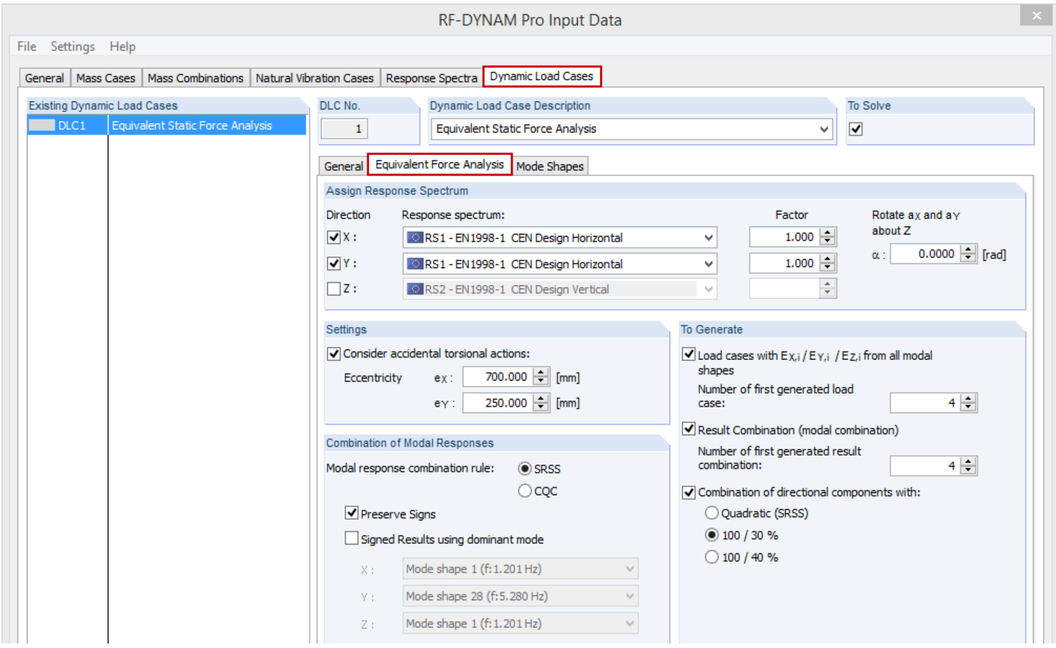



Figure 5.12: *Dynamic Load Cases* tab with settings for *Equivalent Force Analysis*

The horizontal design response spectrum is applied in the *X*- and *Y*-directions. The *Z*-direction does not need to be taken into account because the vertical spectrum is less than 2.5 $m/s^2$ (see EN 1998-1 [1] Section 4.3.3.5.2).

The option to *Consider accidental torsional actions* is selected to account for uncertainties in the location of mass. This is described in EN 1998-1 [1] Section 4.3.2. The eccentricities to be applied are calculated with $e_i = 0.05 \cdot L_i$ where $L_i$ is the length of the building.

The equivalent loads are exported to *Load cases*; the number of the first generated load case is set to *4*.

The combination of modal responses is done with the *SRSS* rule in this example. It can be used when the eigenvalues are independent. This is the case when the criteria $T_j \leq 0.9 \cdot T_i$ is fulfilled with $T_j \leq T_i$ (EN 1998-1 [1] Section 4.3.3.3.2). All adjacent natural periods $T_i$ are independent in this example:

$$0.718 \leq 0.9 \cdot 0.833 = 0.750 \qquad (5.7)$$

$$0.514 \leq 0.9 \cdot 0.718 = 0.646 \qquad (5.8)$$

The *SRSS* rule is applied in form of the equivalent linear combination. The formula of this combination rule is provided in Section 2.11.3 in Equation 2.24.

The results of different excitation directions are to be combined according to the *100 / 30 %* rule. The directional combinations are regulated in EN 1998-1 [1] Section 4.3.3.5.1.

In the *Mode Shapes* sub-tab, the results of the assigned natural vibration case *NVC1* are listed. Here specific eigenvalues can be selected that are to be used for the multi-modal response spectrum analysis. The response spectrum is shown in the graphic below the table. The corresponding acceleration values of each eigenvalue are listed in the table and are marked in red in the graphic. The selection of this example is shown in Figure 5.13.

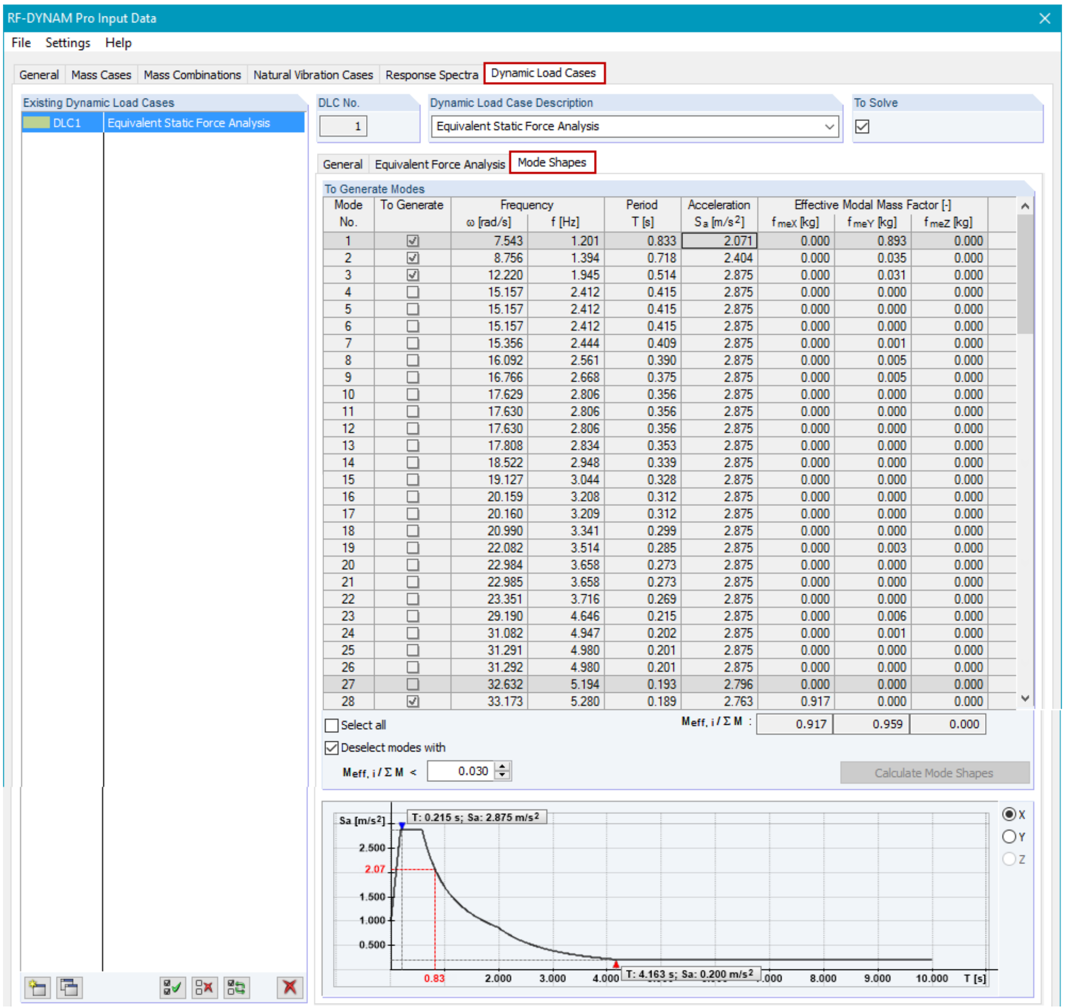



Figure 5.13: Selection of eigenvalues for multi-modal response spectrum analysis

The *Load Cases* and *Result Combination* are exported automatically. Due to the accidental torsion, two load cases for each eigenvalue and excitation direction are exported – one with positive torsional moments and one with negative torsional moments. The list of exported *Load Cases* is shown in Figure 5.14.

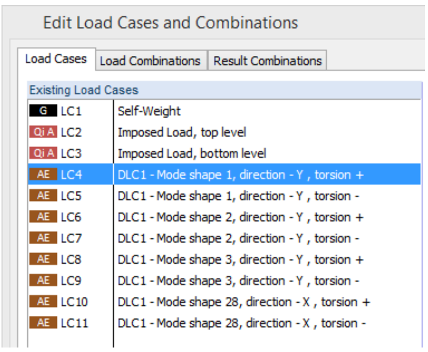



Figure 5.14: List of exported *Load Cases* containing equivalent loads separately for each eigenvalue and excitation direction – due to accidental torsion two load cases are created for each eigenvalue

The list of exported *Result Combinations* is shown in Figure 5.15.

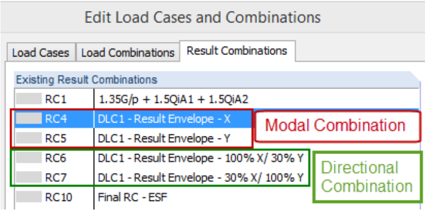



Figure 5.15: List of exported *Result Combination* for modal combination and directional combination

For the modal combination, load cases (*LC*) resulting from different eigenvalues but from the same excitation direction are combined with the SRSS rule. The two *LC*s with positive and negative torsional moments are grouped together (alternative combination). The combination scheme is shown in Figure 5.16.

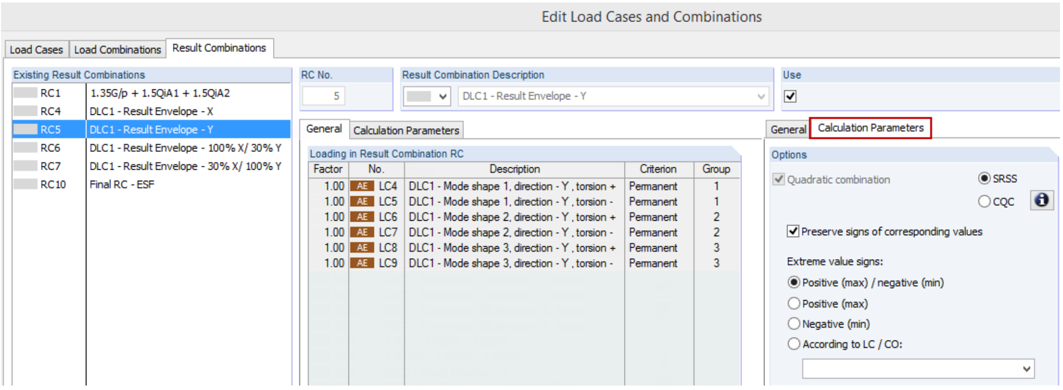



Figure 5.16: Modal combination scheme to combine modal response resulting from excitation in Y-direction

More information on the modal combination when accidental torsion is applied can be found in the Knowledge Base - 001118.

For the directional combination, the 100% / 30% rule is applied in this example. This is simply achieved with combination factors as shown in Figure 5.17.

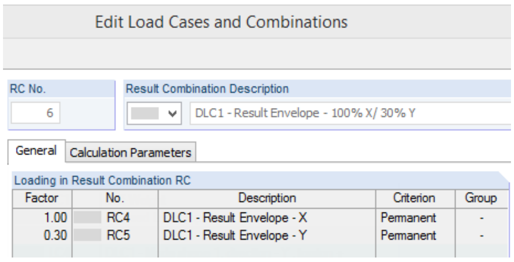



Figure 5.17: Directional combination scheme to combine results from different excitation directions with 100% / 30% rule

RF-DYNAM Pro exports two *RC*s in this example because two excitation directions are taken into account. The *RC* leading to the most unfavourable results is used for the seismic design situation.

## Base Shear Force

There are several options in RF-DYNAM Pro and RFEM to evaluate the base shear force.

- **List of equivalent loads in Tables 5.8 and 5.9**

  The equivalent loads are listed in the results tables 5.8, 5.9, and 5.10 together with the *sum* of the equivalent loads. When *Mode Shape 1* is selected, the equivalent loads $F_Y$ are listed in Table 5.9. The sum is provided as 141.29 kN as shown in Figure 5.18. Note that the equivalent loads are listed twice because accidental torsions have been activated. Thus, the loads are given for the cases *Torsion +* and *Torsion -*. The real base shear force for *Mode Shape 1* is $F_Y = 70.65$ kN.

5.9 Equivalent Loads (Y-excitations)

DLC1 - Equivalent Static Force Analy | Mode Shape 1 (f : 1.201 Hz)

| FE Mesh Point | Mode shape No. | LC No. | Object Type | Location X [m] | Y [m] | Z [m] | Equivalent Load $F_X$ [kN] | $F_Y$ [kN] | $F_Z$ [kN] | $M_Z$ [kNm] |
|---|---|---|---|---|---|---|---|---|---|---|
| 799 | 1 | 4 | Surface | 0.000 | 1.000 | -4.000 | 0.00 | -0.17 | 0.00 | 0.12 |
| | 1 | 5 | Surface | 0.000 | 1.000 | -4.000 | 0.00 | -0.17 | 0.00 | -0.12 |
| 800 | 1 | 4 | Surface | 1.000 | 1.000 | -4.000 | 0.00 | -0.23 | 0.00 | 0.16 |
| | 1 | 5 | Surface | 1.000 | 1.000 | -4.000 | 0.00 | -0.23 | 0.00 | -0.16 |
| 801 | 1 | 4 | Surface | 1.500 | 1.000 | -4.000 | 0.00 | -0.23 | 0.00 | 0.16 |
| | 1 | 5 | Surface | 1.500 | 1.000 | -4.000 | 0.00 | -0.23 | 0.00 | -0.16 |
| 802 | 1 | 4 | Surface | 2.000 | 1.000 | -4.000 | 0.00 | -0.23 | 0.00 | 0.16 |
| | 1 | 5 | Surface | 2.000 | 1.000 | -4.000 | 0.00 | -0.23 | 0.00 | -0.16 |
| 803 | 1 | 4 | Surface | 2.500 | 1.000 | -4.000 | 0.00 | -0.23 | 0.00 | 0.16 |
| | 1 | 5 | Surface | 2.500 | 1.000 | -4.000 | 0.00 | -0.23 | 0.00 | -0.16 |
| sum | | | | | | | 0.00 | 141.29 | 0.00 | |

Equivalent Loads (X-excitations) | Equivalent Loads (Y-excitations)

Figure 5.18: Sum of equivalent loads $\sum F_Y$ resulting from *Mode Shape 1*, dominant in Y-direction – note that the equivalent loads are listed twice for *Torsion +* and *Torsion -* cases

  The sum of equivalent loads resulting from *Mode Shape 28* is listed in Table 5.8 (see Figure 5.19). Again the equivalent loads are listed twice. The real base shear force is $F_X = 96.83$ kN.

5.8 Equivalent Loads (X-excitations)

DLC1 - Equivalent Static Force Analy | Mode Shape 28 (f : 5.280 Hz)

| FE Mesh Point | Mode shape No. | LC No. | Object Type | Location X [m] | Y [m] | Z [m] | Equivalent Load $F_X$ [kN] | $F_Y$ [kN] | $F_Z$ [kN] | $M_Z$ [kNm] |
|---|---|---|---|---|---|---|---|---|---|---|
| 799 | 28 | 10 | Surface | 0.000 | 1.000 | -4.000 | -0.16 | 0.00 | 0.00 | 0.04 |
| | 28 | 11 | Surface | 0.000 | 1.000 | -4.000 | -0.16 | 0.00 | 0.00 | -0.04 |
| 800 | 28 | 10 | Surface | 1.000 | 1.000 | -4.000 | -0.23 | 0.00 | 0.00 | 0.06 |
| | 28 | 11 | Surface | 1.000 | 1.000 | -4.000 | -0.23 | 0.00 | 0.00 | -0.06 |
| 801 | 28 | 10 | Surface | 1.500 | 1.000 | -4.000 | -0.23 | 0.00 | 0.00 | 0.06 |
| | 28 | 11 | Surface | 1.500 | 1.000 | -4.000 | -0.23 | 0.00 | 0.00 | -0.06 |
| 802 | 28 | 10 | Surface | 2.000 | 1.000 | -4.000 | -0.23 | 0.00 | 0.00 | 0.06 |
| | 28 | 11 | Surface | 2.000 | 1.000 | -4.000 | -0.23 | 0.00 | 0.00 | -0.06 |
| 803 | 28 | 10 | Surface | 2.500 | 1.000 | -4.000 | -0.23 | 0.00 | 0.00 | 0.06 |
| | 28 | 11 | Surface | 2.500 | 1.000 | -4.000 | -0.23 | 0.00 | 0.00 | -0.06 |
| sum | | | | | | | 193.66 | 0.01 | 0.00 | |

Equivalent Loads (X-excitations) | Equivalent Loads (Y-excitations)

Figure 5.19: Sum of equivalent loads $\sum F_X$ resulting from *Mode Shape 28*, dominant in X-direction

- **Information on load cases in Table 4.0**

  In Table *4.0 Results - Summary*, the sums of all loads and the sums of support forces are listed. For the dominant eigenvalue in Y-direction, which is exported to *LC4* and *LC5* respectively, the base shear force is $\sum F_Y = 70.65$ kN (see Figure 5.20).

4.0 Results - Summary

LC4 - DLC1 - Mode sha

| Description | Value | Unit |
|---|---|---|
| **LC4 - DLC1 - Mode shape 1, direction - Y , torsion +** | | |
| Sum of loads in X | 0.00 | kN |
| Sum of support forces in X | 0.00 | kN |
| Sum of loads in Y | -70.65 | kN |
| Sum of support forces in Y | -70.65 | kN |

Figure 5.20: Sum of loads $\sum F_Y$ for *LC 4* resulting from *Mode Shape 1*, dominant in Y-direction

  For the dominant eigenvalue in X-direction, which is exported to *LC10* and *LC11* respectively, the base shear force is $\sum F_X = 96.83$ kN (see Figure 5.21).

4.0 Results - Summary

LC10 - DLC1 - Mode sh

| A Description | B Value | C Unit |
|---|---|---|
| LC10 - DLC1 - Mode shape 28, direction - X , torsion + | | |
| Sum of loads in X | -96.83 | kN |
| Sum of support forces in X | -96.83 | kN |
| Sum of loads in Y | 0.01 | kN |
| Sum of support forces in Y | 0.01 | kN |

Figure 5.21: Sum of loads $\sum F_X$ for *LC 10* resulting from *Mode Shape 28*, dominant in X-direction

- **Evaluation with *Result Beam***

  A result beam can be placed anywhere in the model as a virtual member. It integrates the results over the objects that you define. For further details, see Chapter 4.17 of the RFEM manual and the Knowledge Base - 000704. The result beam is very useful to evaluate the base shear forces separately for each storey of the building. In this example, the result beam is placed outside the structural system. It includes all objects as shown in Figure 5.22.

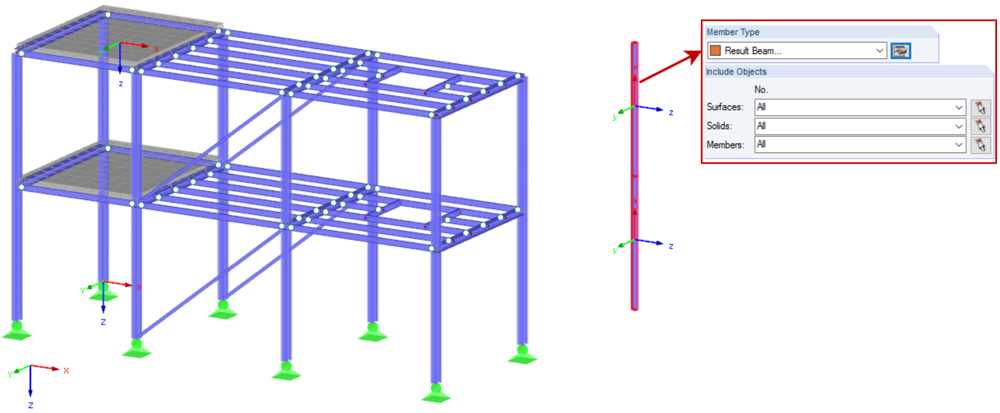



Figure 5.22: Structural system with result beam placed outside of model, including all objects

In *LC4*, the shear forces $V_y$ in the result beam provide the base shear forces resulting from *Mode shape 1* in Y-direction. In *LC10*, the shear forces $V_z$ in the result beam provide the base shear forces resulting from *Mode shape 28* in X-direction. Both are displayed in Figure 5.23 and the results are summarised in Table 5.1.

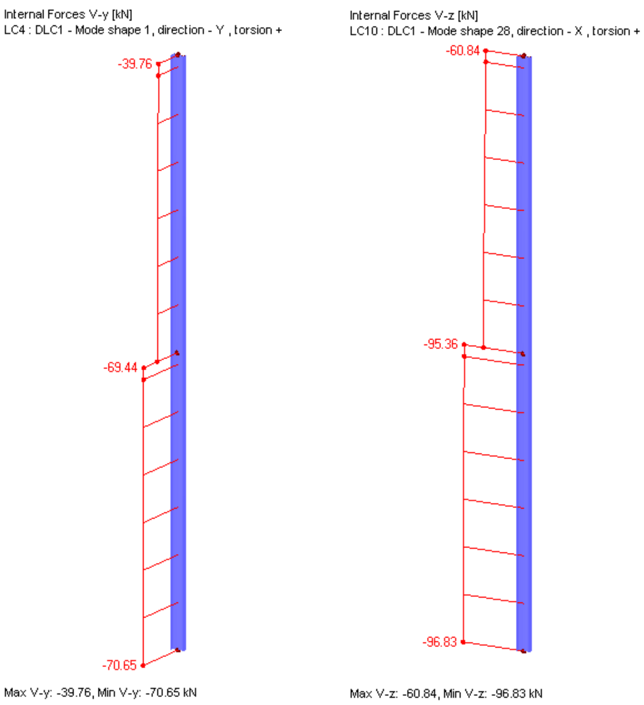



Figure 5.23: Base shear forces of result beam for LC4 ($V_y$ resulting from mode shape 1 in Y-direction) and LC10 ($V_z$ resulting from mode shape 28 in X-direction)

The base shear forces of the result combinations (*RC*) can also be evaluated, which gives the base shear forces of the combined results. *RC5* contains the results of the SRSS rule in the Y-direction, *RC4* the combined results in the X-direction.

The shear forces in the result beam are shown in Figure 5.24.

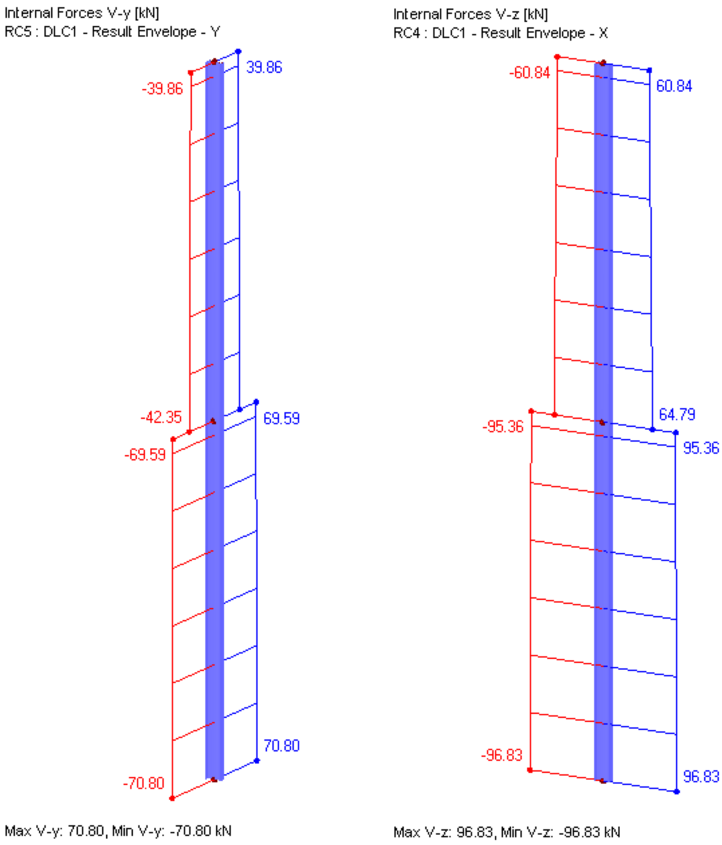



Figure 5.24: Base shear forces showing combined dynamic results of result beam for RC5 ($V_y$ in Y-direction) and RC4 ($V_z$ in X-direction)

The base shear forces of the single storeys are listed in Table 5.1.

| | Level Z = –8 m | Level Z = –4 m | ∑F |
|---|---|---|---|
| *LC4*: Mode Shape 1 - Y-Direction | 39.76 kN | 30.89 kN | 70.65 kN |
| *LC10*: Mode Shape 28 - X-Direction | 60.84 kN | 35.99 kN | 96.83 kN |
| *RC5*: SRSS - Y-Direction | 39.86 kN | 30.94 kN | 70.80 kN |
| *RC4*: SRSS - X-Direction | 60.84 kN | 35.99 kN | 96.83 kN |

Table 5.1: Base shear forces of single storeys resulting from dominant eigenvalues and modal combination

## Storey Drift

Maximal displacements can be used to evaluate the inter-storey drift. In accordance with EN 1998-1 [1] Section 4.4.2.2, the influence of the second-order effects (P-$\Delta$ effect) can be neglected when the inter-storey drift sensitivity coefficient $\theta$ is small enough. It is defined as follows:

$$\theta = \frac{P_{tot} \cdot d_r}{V_{tot} \cdot h} \leq 0.1 \tag{5.9}$$

where

- $\theta$ : Inter-storey drift sensitivity coefficient
- $P_{tot}$ : Total gravity load at and above the storey considered
- $d_r$ : Design inter-storey drift, i.e. difference of the lateral displacements $d_s$ at the top and bottom of the storey under consideration
- $V_{tot}$ : Total seismic storey shear
- h : Inter-storey height

To evaluate the total gravity load at and above the storey considered, the masses on each FE-node need to be analysed.

The masses on each FE-node together with the coordinates of those FE-nodes are listed in Table 5.6. That table can be exported to Excel and filter options can be used to summarise the masses for each storey. For the roof level Z = −8 m, the mass $m_{Z=-8\ m}$ = 19,588.21 kg has to be considered. The mass at level Z = −4 m consists of the bottom storey, which is $m_{bottom}$ = 17,927.17 kg, and the mass at the roof level. Altogether the mass to be considered at the level Z = −4 m is $m_{Z=-4\ m}$ = 37,515.38 kg. For the level Z = 0 m, the sum of masses is $m_{Z=0\ m}$ = 813.32 kg; those are not considered as this is the level of support.

The storey shear forces were discussed in the last paragraph. The loads are summarised in Table 5.1. The height of the storeys is h = 4 m.

The global deformations $u_Y$ in *RC5* (earthquake in Y-direction) are illustrated in Figure 5.25.

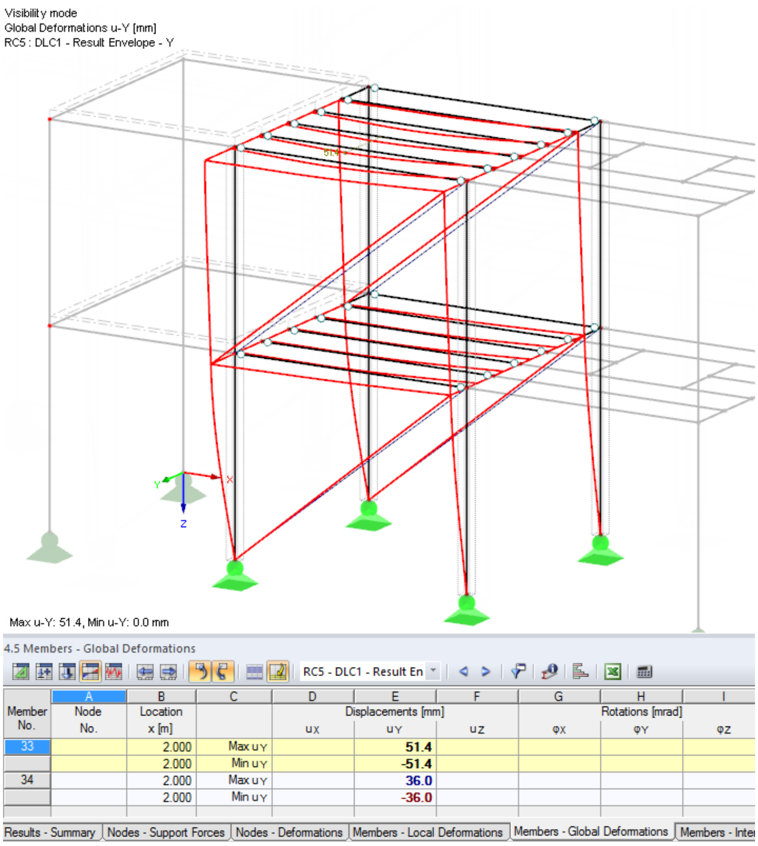



Figure 5.25: Global deformations $u_Y$ resulting from earthquake excitation in Y-direction (*RC5*)

The design inter-storey drift for the top level is $d_{r,top} = 51.4 - 36.0 = 15.4$ mm. For the bottom level, $d_{r,bottom} = 36.0$ mm. The inter-storey drift sensitivity coefficient for the top level can be calculated with

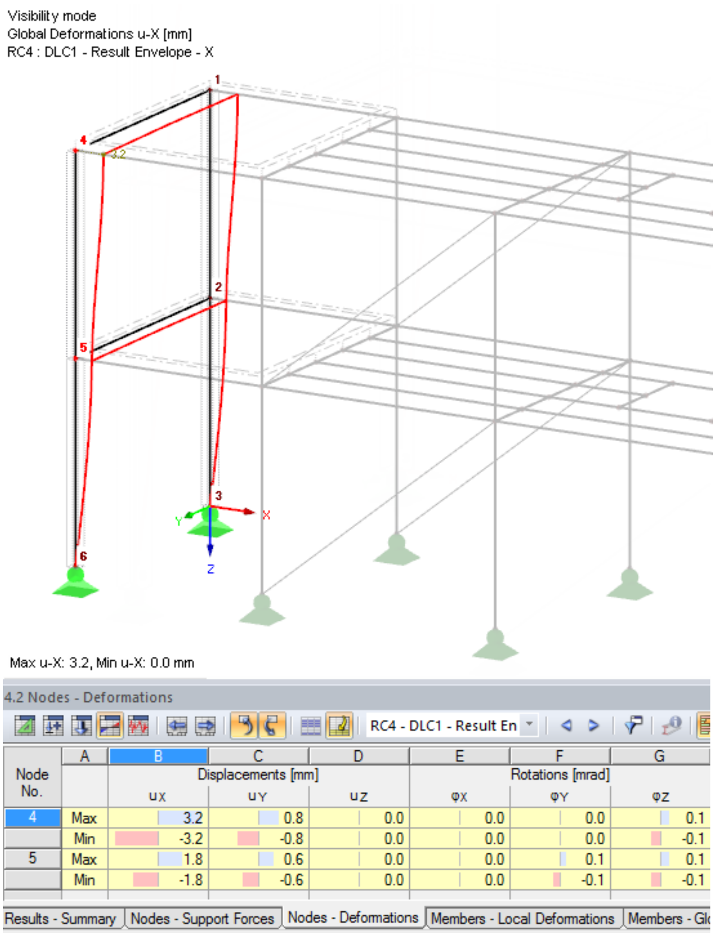
$$\theta_{top} = \frac{19{,}588.21/100 \cdot 15.4/1{,}000}{39.86 \cdot 4} = 0.02 \tag{5.10}$$

and for the bottom level with

$$\theta_{bottom} = \frac{37{,}515.38/100 \cdot 36.0/1{,}000}{30.94 \cdot 4} = 0.11. \tag{5.11}$$

The limit of 0.1 is exceeded in the bottom level.

The global deformations $u_X$ in *RC4* (earthquake in X-direction) are illustrated in Figure 5.26.




| Node No. | | Displacements [mm] uX | uY | uZ | Rotations [mrad] φX | φY | φZ |
|---|---|---|---|---|---|---|---|
| 4 | Max | 3.2 | 0.8 | 0.0 | 0.0 | 0.0 | 0.1 |
| | Min | -3.2 | -0.8 | 0.0 | 0.0 | 0.0 | -0.1 |
| 5 | Max | 1.8 | 0.6 | 0.0 | 0.0 | 0.1 | 0.1 |
| | Min | -1.8 | -0.6 | 0.0 | 0.0 | -0.1 | -0.1 |

Figure 5.26: Global deformations $u_X$ resulting from earthquake excitation in X-direction (*RC4*)

The design inter-storey drift for the top level is $d_{r,top} = 3.2 - 1.8 = 1.4$ mm. For the bottom level, $d_{r,bottom} = 1.8$ mm. The inter-storey drift sensitivity coefficient for the top level can be calculated with

$$\theta_{top} = \frac{19{,}588.21/100 \cdot 1.4/1{,}000}{60.84 \cdot 4} = 1.13 \times 10^{-3} \tag{5.12}$$

and for the bottom level with

$$\theta_{bottom} = \frac{37{,}515.38/100 \cdot 1.8/1{,}000}{35.99 \cdot 4} = 4.69 \times 10^{-3}. \tag{5.13}$$

In the X-direction of the building, the limit of 0.1 is not exceeded.

In this example the inter-storey drift sensitivity coefficient $\theta$ according to Equation 5.9 in the bottom level in Y-direction is slightly exceeded. In accordance with EN 1998-1 [1] Section 4.4.2.2 (3), second-order effects can be approximately taken into account by multiplying the seismic action effects with a factor of $1/(1-\theta)$, provided that the sensitivity coefficient $\theta$ is smaller than 0.2.

## Second-Order Effects

- **Approximate consideration of P-$\Delta$ effects with factor 1/(1 - $\theta$)**

  To increase results in order to approximately consider P-$\Delta$ effects, *Factor* $1/(1-0.11) = 1.124$ is used in the dynamic load case *DLC2* as shown in Figure 5.27. Only the Y-direction is increased by this factor.

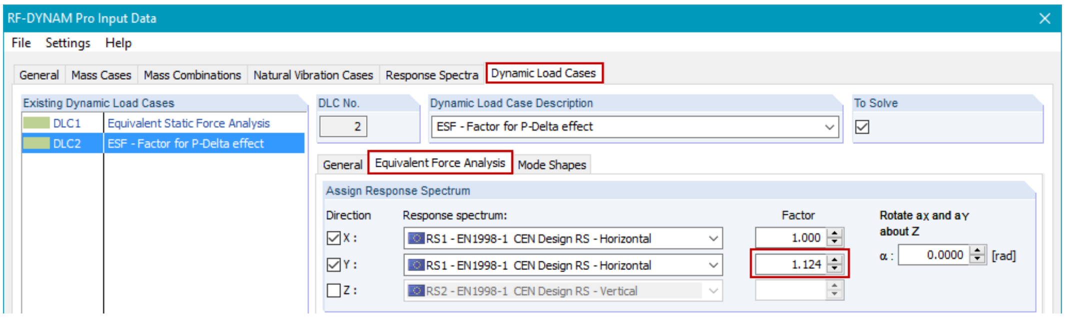



Figure 5.27: Factor $1/(1-\theta)$ to approximately consider P-$\Delta$ effects

- **Geometric stiffness matrix to consider P-$\Delta$ effects**

  When you have larger storey drifts and need to consider the P-$\Delta$ effects in a more exact manner, you can activate the influence of axial forces in RF-DYNAM Pro. This takes the geometric stiffness matrix into account. Frequencies and equivalent loads will then be calculated with a modified stiffness matrix. The influence of axial forces can be activated in the *Natural Vibration Cases* tab as shown in Figure 5.28.

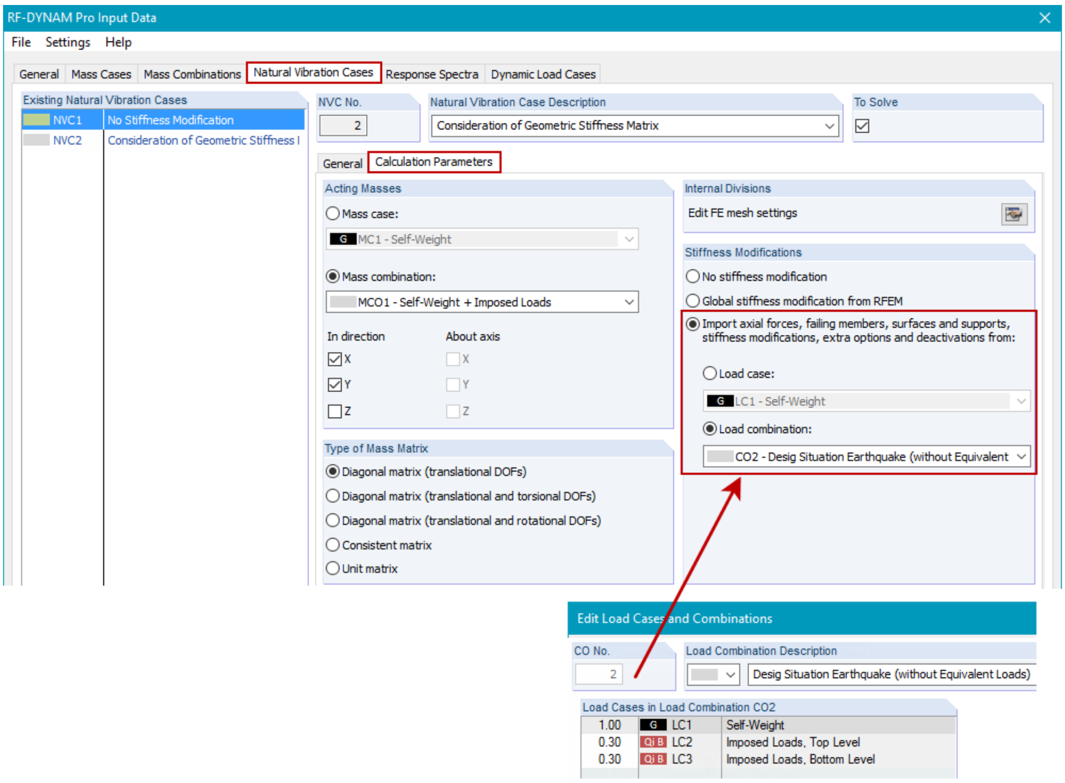
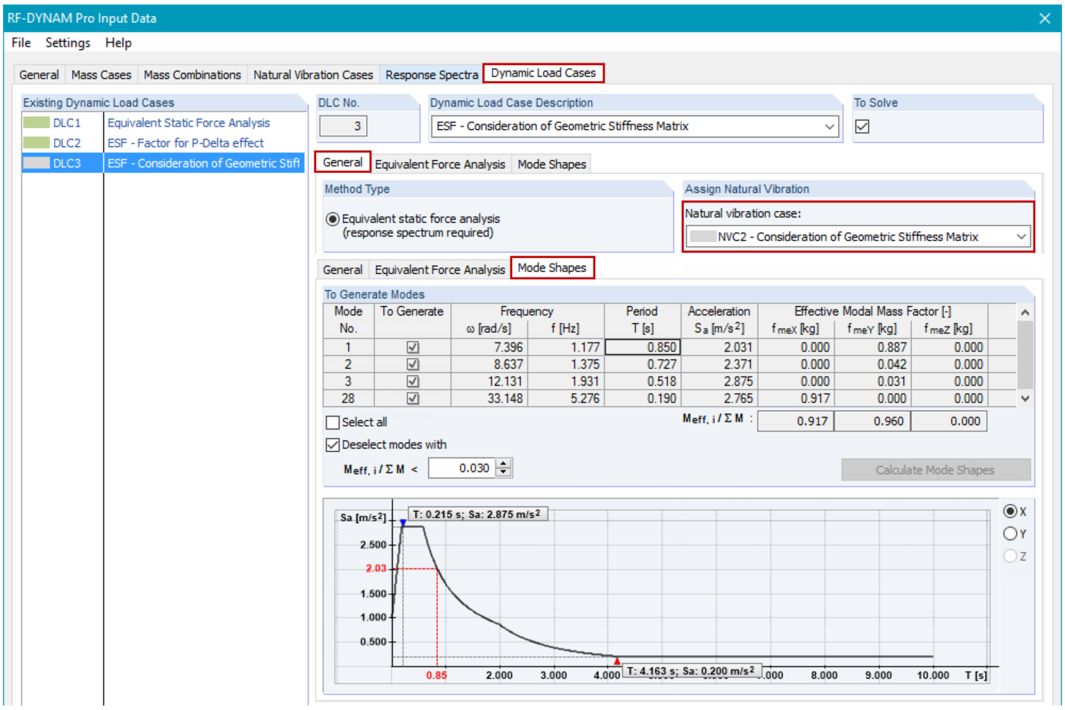



Figure 5.28: Consideration of axial forces in natural vibration case to consider geometric stiffness matrix with axial forces taken from *CO2*

  Load combination *CO2* combines the load cases with self-weight and imposed loads in accordance with the *Design Situation Earthquake*. The combination factors $\Psi_2$ are in accordance with EN 1990 [4] Table A.1.1. This *CO2* is used for the modification of the stiffness matrix, as this is the design combination which is relevant in an event of an earthquake.

The settings of the natural vibration case (*NVC*) influence the natural frequencies of the system. The *NVC* has to be assigned to the dynamic load case (*DLC*) so that the modified frequencies are used for the calculation of equivalent loads. The settings of *DLC3* are shown in Figure 5.29.




| Mode No. | To Generate | Frequency ω [rad/s] | Frequency f [Hz] | Period T [s] | Acceleration $S_a$ [m/s²] | Effective Modal Mass Factor [-] $f_{meX}$ [kg] | $f_{meY}$ [kg] | $f_{meZ}$ [kg] |
|---|---|---|---|---|---|---|---|---|
| 1 | ☑ | 7.396 | 1.177 | 0.850 | 2.031 | 0.000 | 0.887 | 0.000 |
| 2 | ☑ | 8.637 | 1.375 | 0.727 | 2.371 | 0.000 | 0.042 | 0.000 |
| 3 | ☑ | 12.131 | 1.931 | 0.518 | 2.875 | 0.000 | 0.031 | 0.000 |
| 28 | ☑ | 33.148 | 5.276 | 0.190 | 2.765 | 0.917 | 0.000 | 0.000 |
| Meff, i / Σ M : | | | | | | 0.917 | 0.960 | 0.000 |

Figure 5.29: Modified stiffness matrix assigned to *DLC* – the frequencies are slightly changed, as well as the corresponding acceleration values $S_a$ read from the response spectrum (cf Figure 5.13)

The equivalent loads are calculated with those modified frequencies and are then exported to the main program RFEM. The stiffness modifications are not automatically applied in the load cases. You need to consider the modified stiffness matrix also to calculate your final results like deformations and internal forces. The settings as shown in Figure 5.30 have to set manually.

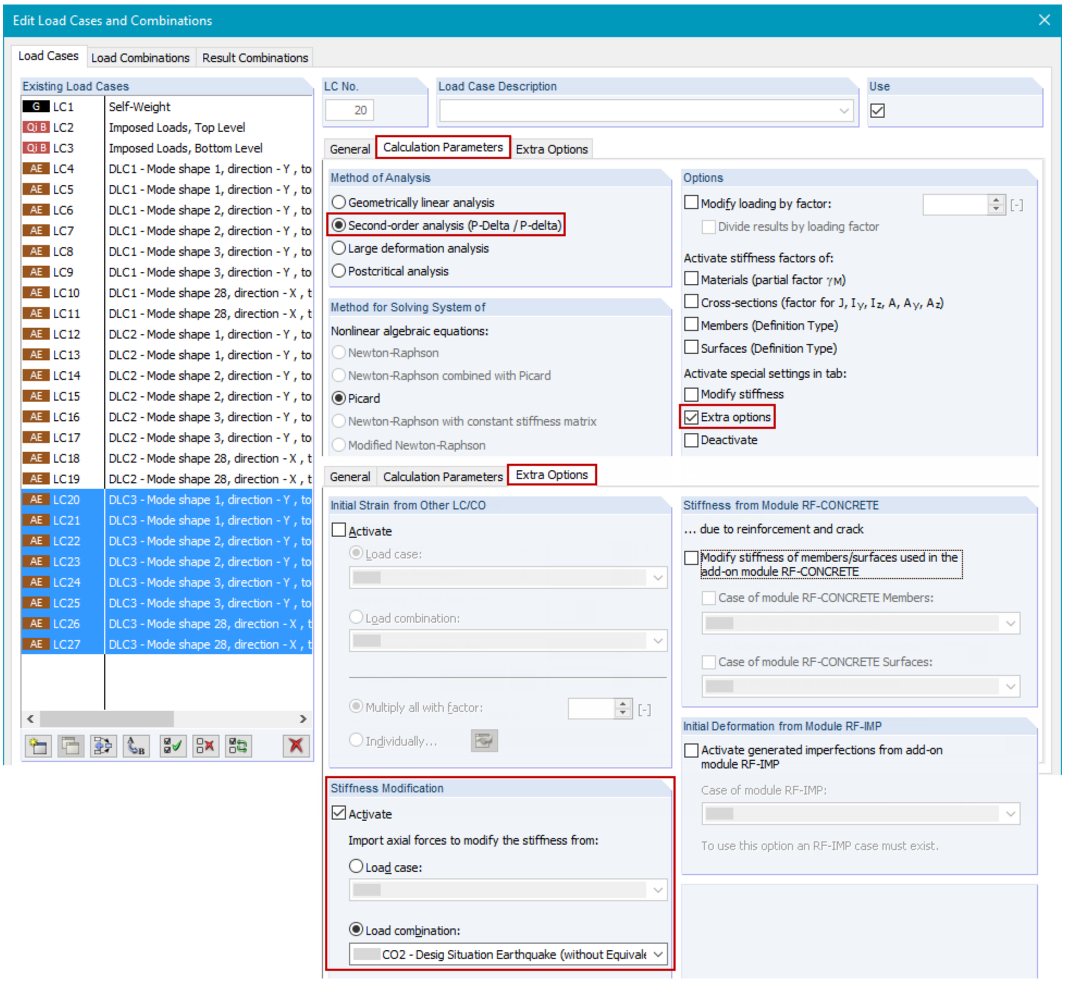



Figure 5.30: Stiffness modification of load cases exported from *DLC3* – the geometric stiffness matrix has to be taken into account for the calculation of those load cases

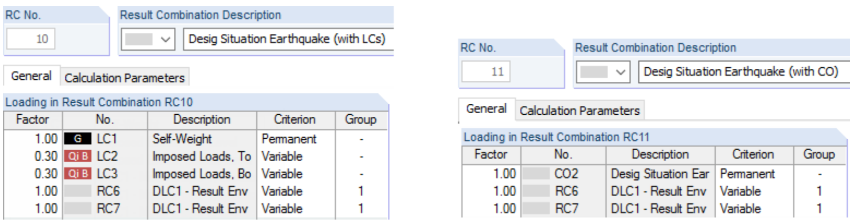
## Result Combination for Design Situation Earthquake

The two result combinations exported from RF-DYNAM Pro (*RC6*: 100% X, 30% Y and *RC7*: 30% X, 100% Y) are combined with *OR*. The disadvantageous *RC* is used as $A_{Ed}$ in the design combination for earthquake. This is defined in EN 1990 [4] Section 6.4.3.4, Equation (6.12).

$$E_d = \sum_{j\geq 1} G_{k,j} \text{ “+” } P \text{ “+” } A_{Ed} \text{ “+” } \sum \Psi_{2,i} Q_{k,i} \tag{5.14}$$

The combination factors $\Psi_2$ are listed in EN1990 [21] Table A.1.1. Note that those combination factors have already been used in the mass combinations (*MCO*).

The combination for the seismic design situation has to be defined manually in RFEM. This can be done using load cases (*LC*) or the priorly defined load combination *CO2*. Both options are shown in Figure 5.31.




| Factor | No. | Description | Criterion | Group |
|---|---|---|---|---|
| 1.00 | LC1 | Self-Weight | Permanent | - |
| 0.30 | LC2 | Imposed Loads, To | Variable | - |
| 0.30 | LC3 | Imposed Loads, Bo | Variable | - |
| 1.00 | RC6 | DLC1 - Result Env | Variable | 1 |
| 1.00 | RC7 | DLC1 - Result Env | Variable | 1 |

a) Combination with *LCs*

| Factor | No. | Description | Criterion | Group |
|---|---|---|---|---|
| 1.00 | CO2 | Desig Situation Ear | Permanent | - |
| 1.00 | RC6 | DLC1 - Result Env | Variable | 1 |
| 1.00 | RC7 | DLC1 - Result Env | Variable | 1 |

b) Combination with *CO2*

Figure 5.31: Seismic design situation in accordance with EN 1990: a) combination with load cases and b) combination with load combination *CO2* defined earlier using the same combination factors $\Psi_2$

The use of a load combination (*CO*) is recommended when nonlinearities exist in the model, or when second-order effects have to be considered.

The resulting moment $M_Y$ and the support forces on selected members of the structure are illustrated in Figure 5.32.

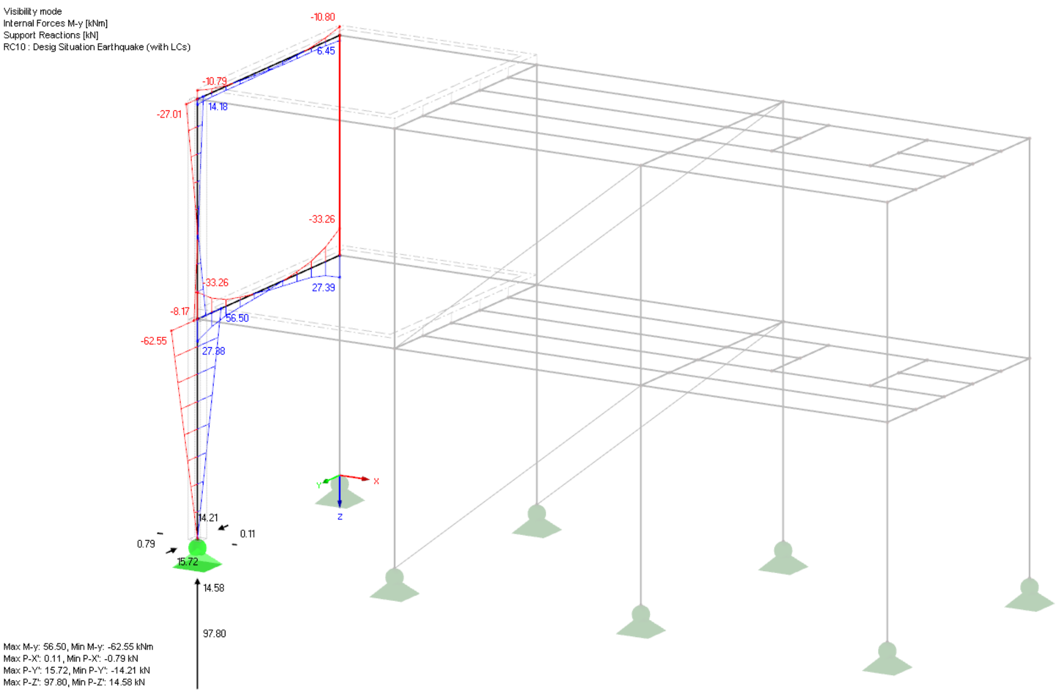
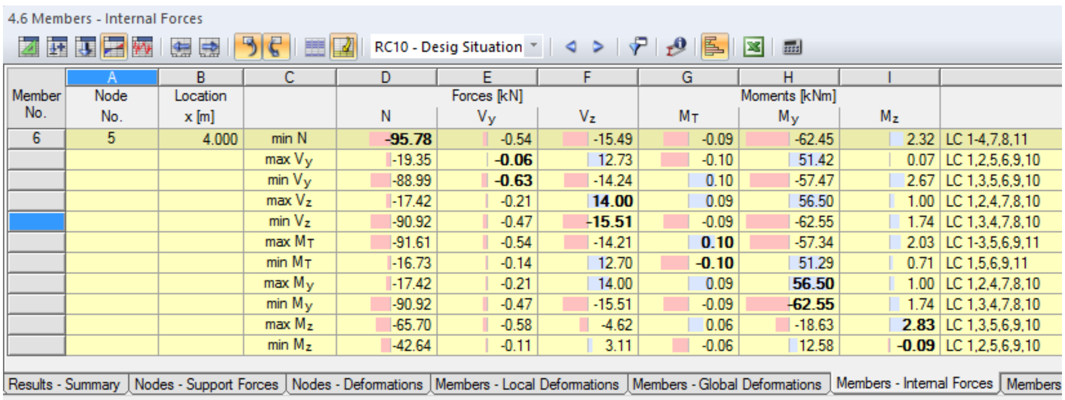



Figure 5.32: Results of *RC10* for seismic design situation showing moments $M_Y$ and support forces on selected members

The internal forces are listed in Table 4.6 as shown in Figure 5.33 for member *6*.



4.6 Members - Internal Forces

RC10 - Desig Situation

| Member No. | Node No. | Location x [m] | | N [kN] | $V_y$ [kN] | $V_z$ [kN] | $M_T$ [kNm] | $M_y$ [kNm] | $M_z$ [kNm] | |
|---|---|---|---|---|---|---|---|---|---|---|
| 6 | 5 | 4.000 | min N | **-95.78** | -0.54 | -15.49 | -0.09 | -62.45 | 2.32 | LC 1-4,7,8,11 |
| | | | max $V_y$ | -19.35 | **-0.06** | 12.73 | -0.10 | 51.42 | 0.07 | LC 1,2,5,6,9,10 |
| | | | min $V_y$ | -88.99 | **-0.63** | -14.24 | 0.10 | -57.47 | 2.67 | LC 1,3,5,6,9,10 |
| | | | max $V_z$ | -17.42 | -0.21 | **14.00** | 0.09 | 56.50 | 1.00 | LC 1,2,4,7,8,10 |
| | | | min $V_z$ | -90.92 | -0.47 | **-15.51** | -0.09 | -62.55 | 1.74 | LC 1,3,4,7,8,10 |
| | | | max $M_T$ | -91.61 | -0.54 | -14.21 | **0.10** | -57.34 | 2.03 | LC 1-3,5,6,9,11 |
| | | | min $M_T$ | -16.73 | -0.14 | 12.70 | **-0.10** | 51.29 | 0.71 | LC 1,5,6,9,11 |
| | | | max $M_y$ | -17.42 | -0.21 | 14.00 | 0.09 | **56.50** | 1.00 | LC 1,2,4,7,8,10 |
| | | | min $M_y$ | -90.92 | -0.47 | -15.51 | -0.09 | **-62.55** | 1.74 | LC 1,3,4,7,8,10 |
| | | | max $M_z$ | -65.70 | -0.58 | -4.62 | 0.06 | -18.63 | **2.83** | LC 1,3,5,6,9,10 |
| | | | min $M_z$ | -42.64 | -0.11 | 3.11 | -0.06 | 12.58 | **-0.09** | LC 1,2,5,6,9,10 |

Results - Summary | Nodes - Support Forces | Nodes - Deformations | Members - Local Deformations | Members - Global Deformations | Members - Internal Forces | Members

Figure 5.33: Table *4.6 Member - Internal Forces* showing results of *RC10* for seismic design situation for member *6* at node *5*

The modal combination rule was applied in form of the equivalent linear combination. The effect of this can be seen on the corresponding internal forces – they are smaller compared to the maximum values (displayed bold on the main diagonal) and have varying signs.

The second-order effects were taken into account (1) by an approximate method with the factor $1/(1-\theta)$ (*DLC2* in RF-DYNAM Pro) and (2) by a more exact method with the consideration of the geometric stiffness matrix (*DLC3*). For both methods, the final seismic design combination was built as described above. Figure 5.34 shows a comparison of those internal forces.

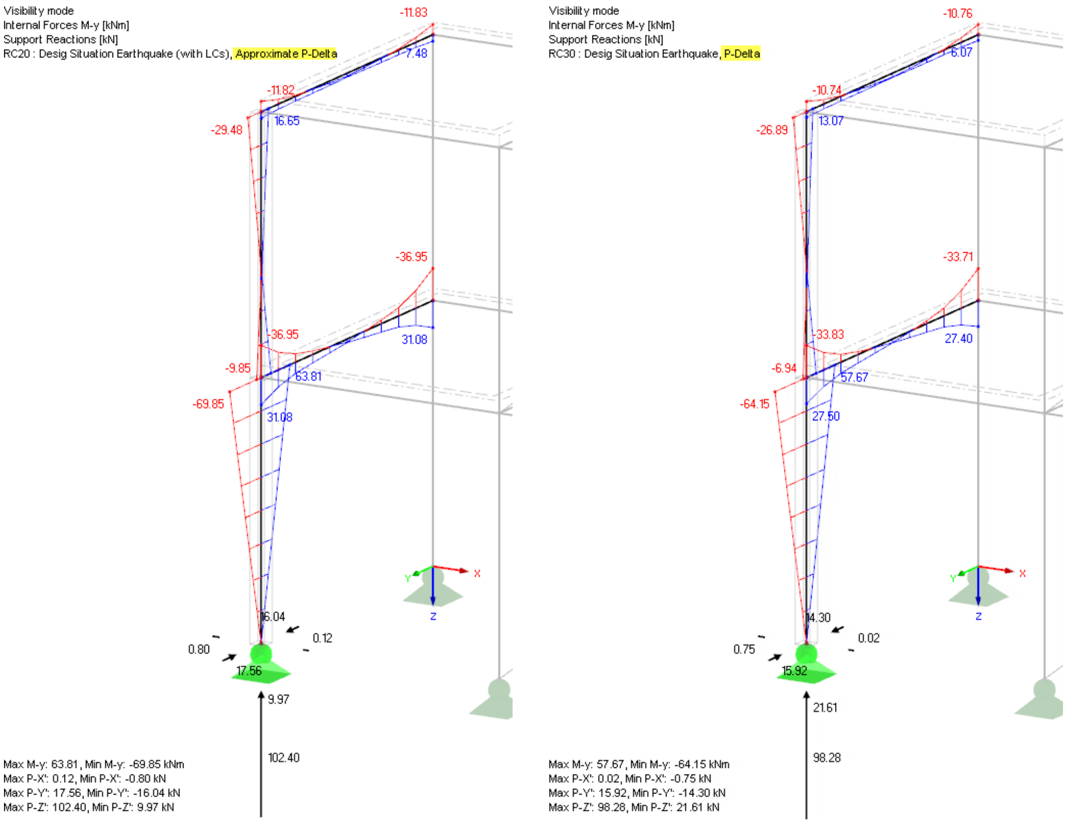



Figure 5.34: Results of seismic design situations considering P-$\Delta$ effect by a factor (*RC20*) and by the geometric stiffness matrix (*RC30*) showing moments $M_Y$ and support forces on selected members

The approximate method using the factor $1/(1-\theta)$ leads to results on the safe side.

With those result combinations for the seismic design situation, the design of the single construction elements can be performed, for example with the add-on modules RF-STEEL, RF-STEEL EC3, or RF-CONCRETE.

## 5.2 Machine-Induced Vibrations

In the second example, the dynamic excitation of two rotating machines is analysed by a linear time history analysis in the add-on module **RF-DYNAM Pro - Forced Vibrations**.

The steel framework that supports the two machines is excited in resonance first, which means that the frequency of the structure is the same as the frequency of the excitation. Resulting from resonance, the systems experiences unacceptable large deformations and accelerations. Changing the cross-sections and thus modifiying the natural frequency of the structure is compulsory.

### 5.2.1 Structural System, Machines, and Load Cases in RFEM

Figure 5.35 shows the structural system and the applied materials and cross-sections.

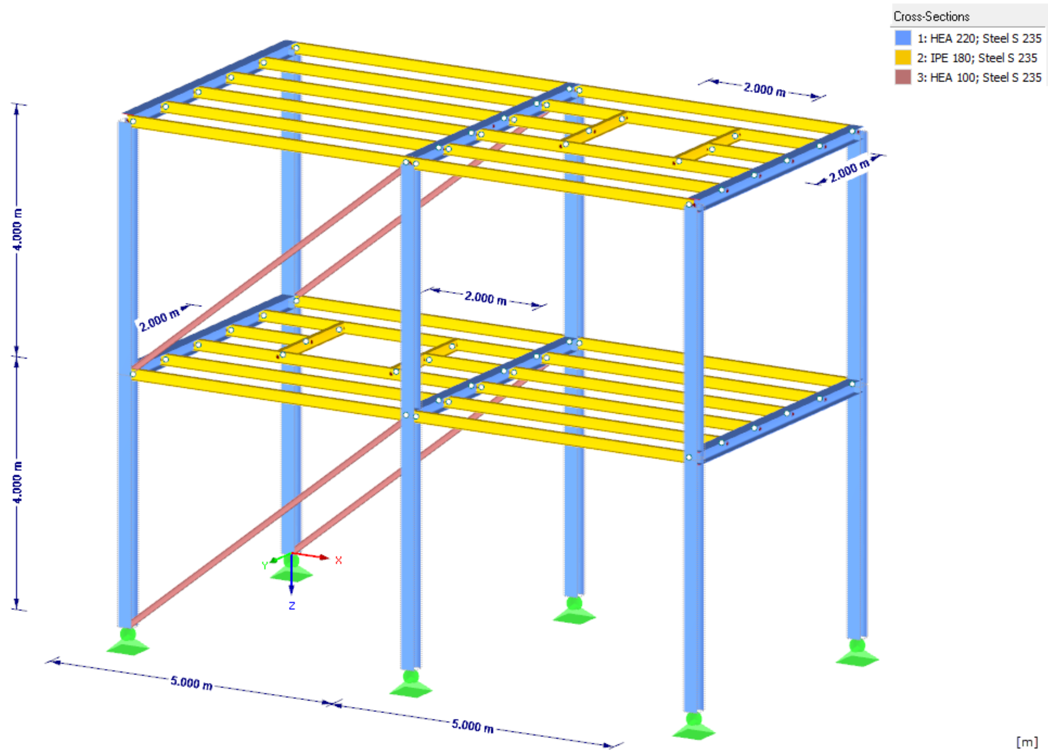



Figure 5.35: Structural system with coloured cross-sections and materials and openings where machines are located

In the openings in the lower and upper levels, two machines are located. The machines themselves are not modelled – they are represented by their dead loads and the centrifugal forces that they produce.

The self-weight of the steel frame in the $+$Z-direction and an additional surface load of 0.5 kN/m$^2$ are defined in *LC1*. This load case will be imported as mass in RF-DYNAM Pro.

Each of the machines has a self-weight of $m = 1.84$ t. This dead load acts as line loads on the adjacent members. It is defined in *LC2* as illustrated in Figure 5.36.

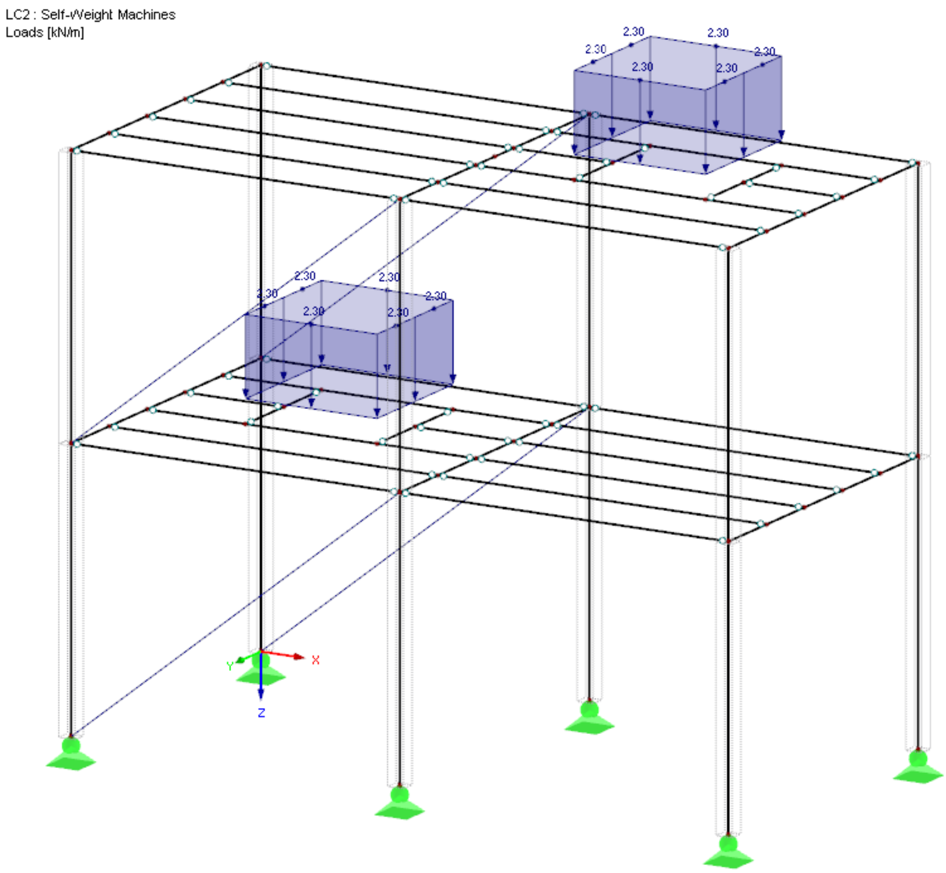



Figure 5.36: Self-weight of machines, defined as line loads on members around openings

The mass of the rotor $m_R = 0.5$ t rotates with 310 rpm with an eccentricity of $e = 19$ mm. This movement is illustrated in Figure 5.37. The centrifugal force $F_r$ acts perpendicularly to the rotational movement.

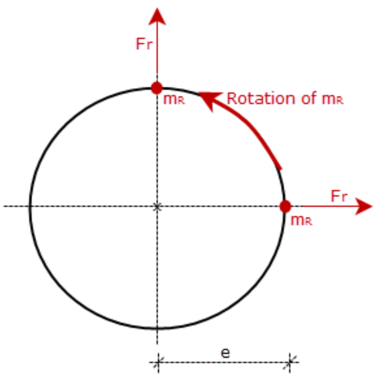



Figure 5.37: Rotation of rotor mass $m_R$ within machine and resulting centrifugal force $F_r$

The excitation frequency of the machine can be calculated with:

$$\omega_e = 2\,\pi \cdot 310/60\text{ s} = 32.463\text{ rad/s} \qquad (5.15)$$

We will define time diagrams later in RF-DYNAM Pro when the excitation frequency $\omega_e$ is entered.

The value of the centrifugal force is defined with:

$$F_r = \omega_e^2 \cdot e \cdot m_R = 10.01\text{ kN} \qquad (5.16)$$

The modelling in RFEM is simplified by defining only the horizontal and vertical centrifugal forces, which is of sufficient accuracy. The horizontal component is delayed in time and defined with a phase shift in the periodical time diagram. This will be considered when the time diagrams are created in RF-DYNAM Pro.

Again, the centrifugal forces are defined as line loads along the openings. The vertical and horizontal components are arranged in separate load cases. The loads in *LC3* and *LC4* determine the size and the position of the time-invariant excitations. Those load cases are shown in Figure 5.38.

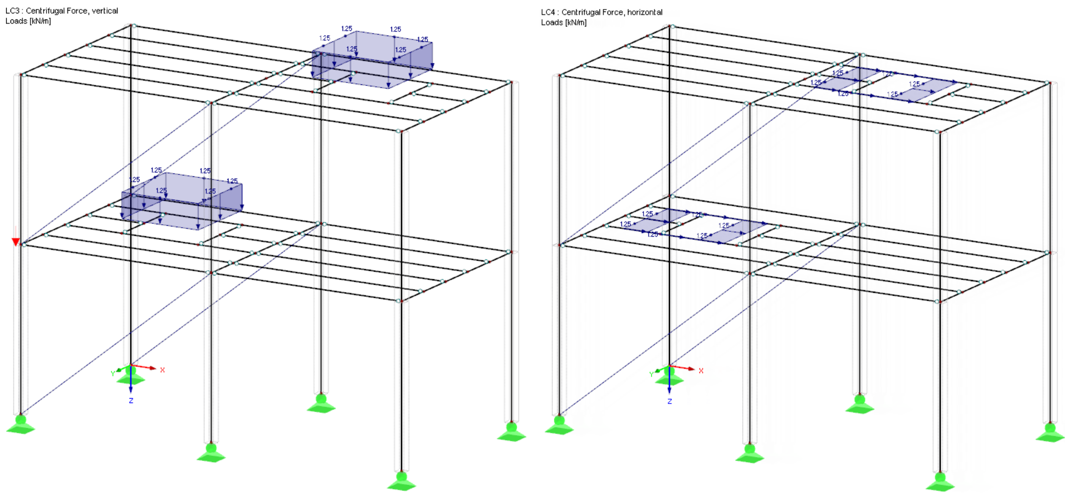



Figure 5.38: Centrifugal forces $F_r$ defined as vertical and horizontal line loads along openings in two load cases

## 5.2.2 Module RF-DYNAM Pro

In the *General* tab, the *Response spectrum analysis* and the *Time Diagrams* are activated of the *RF-DYNAM Pro - Forced Vibrations* module (see Figure 5.39).

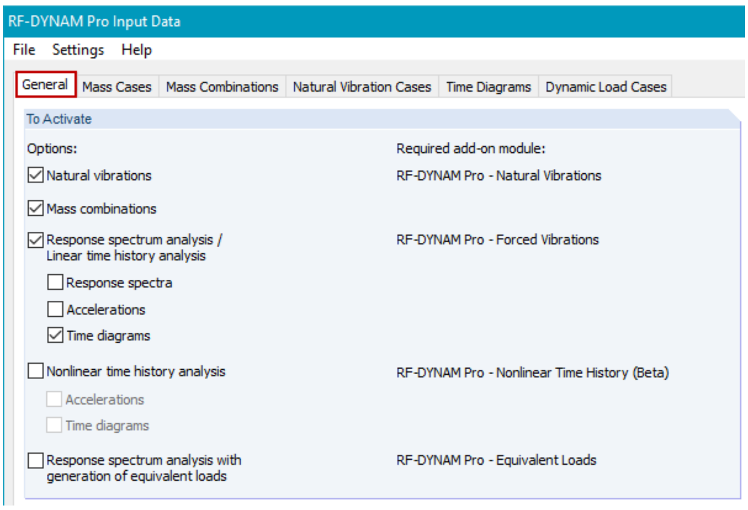



Figure 5.39: *General* tab of RF-DYNAM Pro with *Forced Vibrations* module and *Time Diagrams* selected

## Definition of Masses

In this example, only the self-weight of the steel frame and of the machines are of relevance. Thus, the two relevant load cases *LC1* and *LC2* are imported as masses into RF-DYNAM Pro. Two mass cases are defined: *MC1* is shown in Figure 5.40, *MC2* is defined analogously by importing *LC2*.

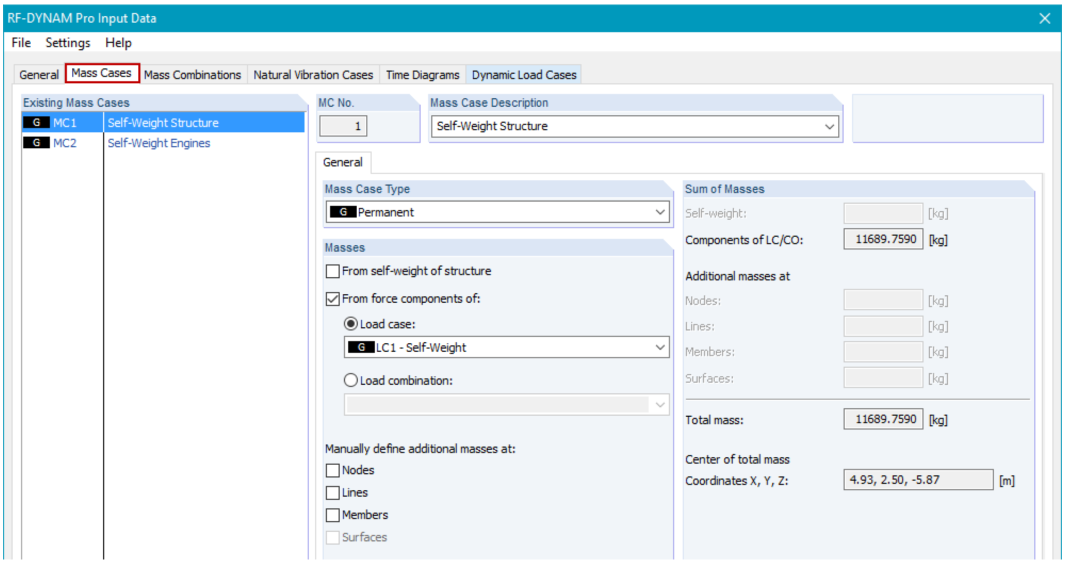



Figure 5.40: Mass case *MC1* with self-weight imported from *LC1* (option *From self-weight of structure* cleared as self-weight has been considered in *LC1*)

The two mass cases of self-weight are combined in mass combination *MCO1* using factors of 1.0. This is shown in Figure 5.41.

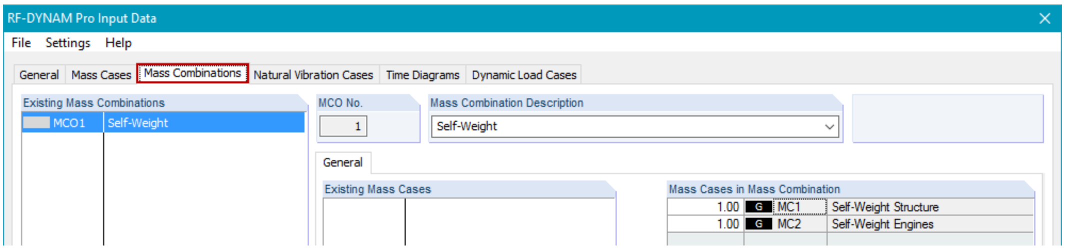



Figure 5.41: Mass combination *MCO1* with combined masses

## Natural Vibration Analysis

A time history analysis with the *linear implicit Newmark solver* does not require any eigenvalues or mode shapes – the analysis is done via a time step integration scheme. Important for the accuracy of the solution is the choice of the time step (see next section). Nevertheless, the natural frequencies are important to understand the behaviour of the structural system. They should be analysed, therefore, so that possible cases of resonance become clear.

In the natural vibration analysis, a diagonal mass matrix is considered with the masses acting in the three translational direction X, Y, and Z. The rotational degrees of freedom are taken into account as well.

The settings of the natural vibration case *NVC1* are illustrated in Figure 5.42. There are *10* eigenvalues to be calculated.

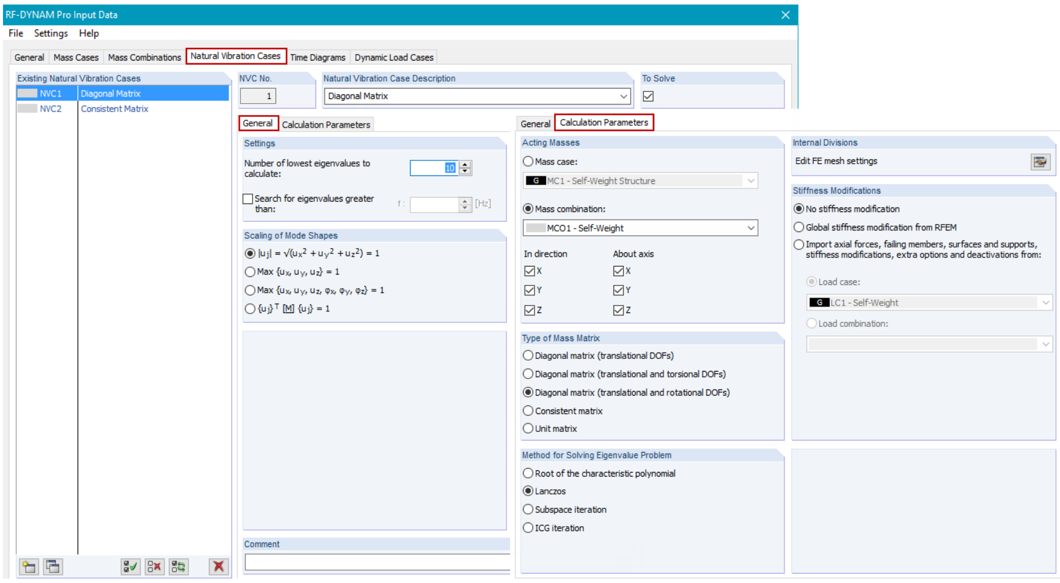
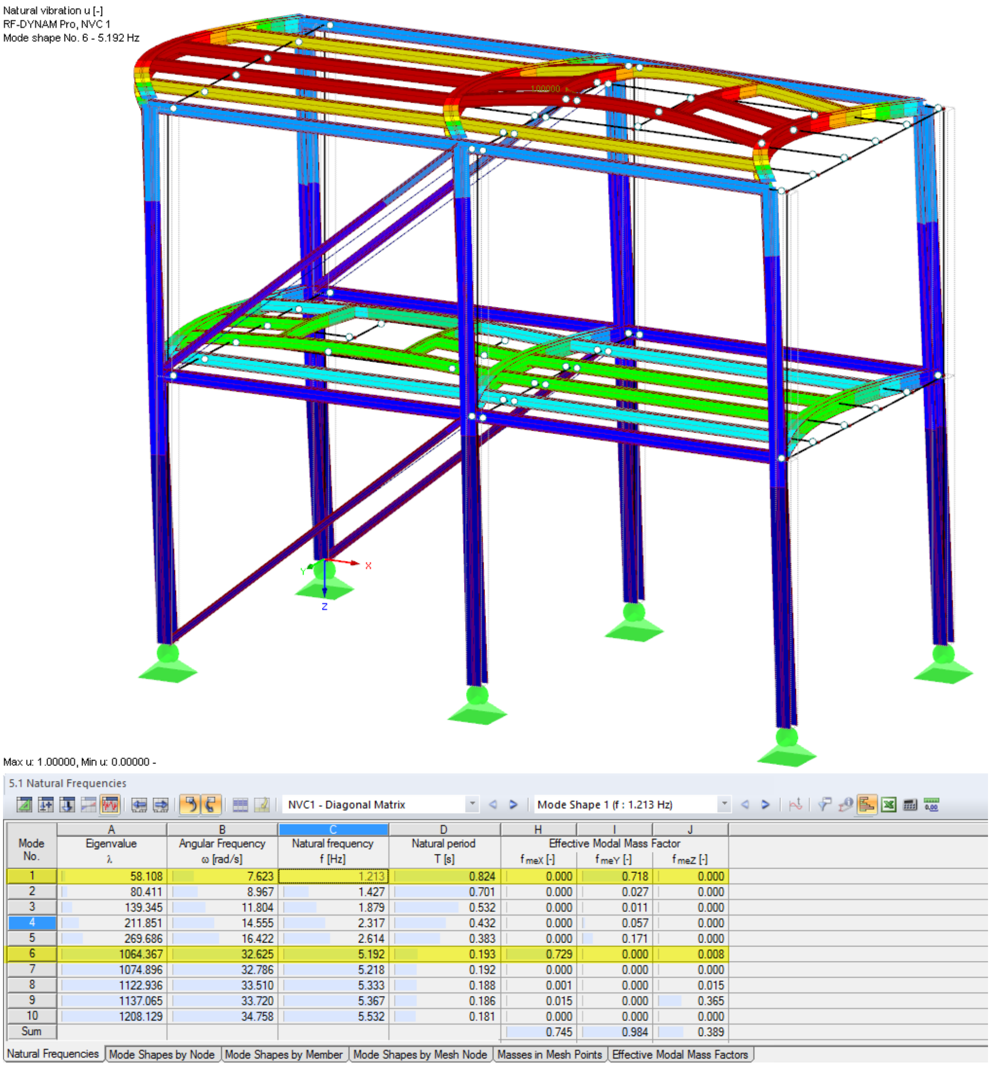



Figure 5.42: Natural vibration case *NVC1* employing a diagonal mass matrix and with masses acting in all directions and about all axes

The relevant mode shape *6* in the X-direction with a natural frequency of $f = 5.192$ Hz is illustrated Figure 5.43. The other eigenvalues are also listed together with the effective modal mass factors. The dominant eigenvalues are highlighted.




| Mode No. | A Eigenvalue λ | B Angular Frequency ω [rad/s] | C Natural frequency f [Hz] | D Natural period T [s] | H Effective Modal Mass Factor $f_{meX}$ [-] | I $f_{meY}$ [-] | J $f_{meZ}$ [-] |
|---|---|---|---|---|---|---|---|
| 1 | 58.108 | 7.623 | 1.213 | 0.824 | 0.000 | 0.718 | 0.000 |
| 2 | 80.411 | 8.967 | 1.427 | 0.701 | 0.000 | 0.027 | 0.000 |
| 3 | 139.345 | 11.804 | 1.879 | 0.532 | 0.000 | 0.011 | 0.000 |
| 4 | 211.851 | 14.555 | 2.317 | 0.432 | 0.000 | 0.057 | 0.000 |
| 5 | 269.686 | 16.422 | 2.614 | 0.383 | 0.000 | 0.171 | 0.000 |
| 6 | 1064.367 | 32.625 | 5.192 | 0.193 | 0.729 | 0.000 | 0.008 |
| 7 | 1074.896 | 32.786 | 5.218 | 0.192 | 0.000 | 0.000 | 0.000 |
| 8 | 1122.936 | 33.510 | 5.333 | 0.188 | 0.001 | 0.000 | 0.015 |
| 9 | 1137.065 | 33.720 | 5.367 | 0.186 | 0.015 | 0.000 | 0.365 |
| 10 | 1208.129 | 34.758 | 5.532 | 0.181 | 0.000 | 0.000 | 0.000 |
| Sum | | | | | 0.745 | 0.984 | 0.389 |

Figure 5.43: Relevant mode shape *6* in X-direction illustrated in work window and all calculated eigenvalues tabulated with effective modal mass factors

With its frequency of $f = 5.192$ Hz, the illustrated mode shape *6* is very close to the excitation frequency of the machines which is $f_e = 5.167$ Hz.

For time history analyses, the use of the consistent mass matrix is recommended in general. The consistent matrix distributes the masses more equally over the members, which results in more local mode shapes to be identified. In this example, however, a diagonal mass matrix is sufficient.

## Time Diagrams

The centrifugal forces as defined in *LC3* and *LC4* shall act over time as harmonic excitations. Therefore, the definition of time diagrams is required in RF-DYNAM Pro. The excitation frequency is $\omega = 32.463$ rad/s (see Equation 5.15). For the vertical component, no phase shift is defined in the periodic time diagram. The horizontal component has to be shifted to express the time delay, however. This is done with a phase shift of $\varphi = \pi/2$. The factor *k* is set to 1 because the magnitude of the load was defined in load cases *LC3* and *LC4*.

The definition of the time diagrams is shown in Figure 5.44 and Figure 5.45.

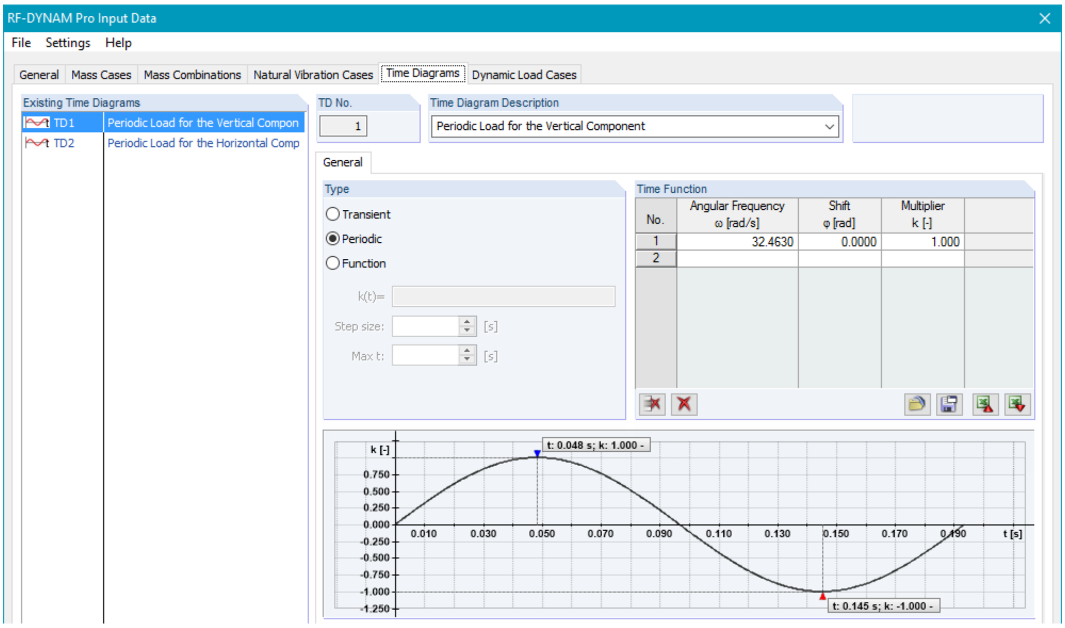



Figure 5.44: Definition of periodic time diagram for vertical component of centrifugal force

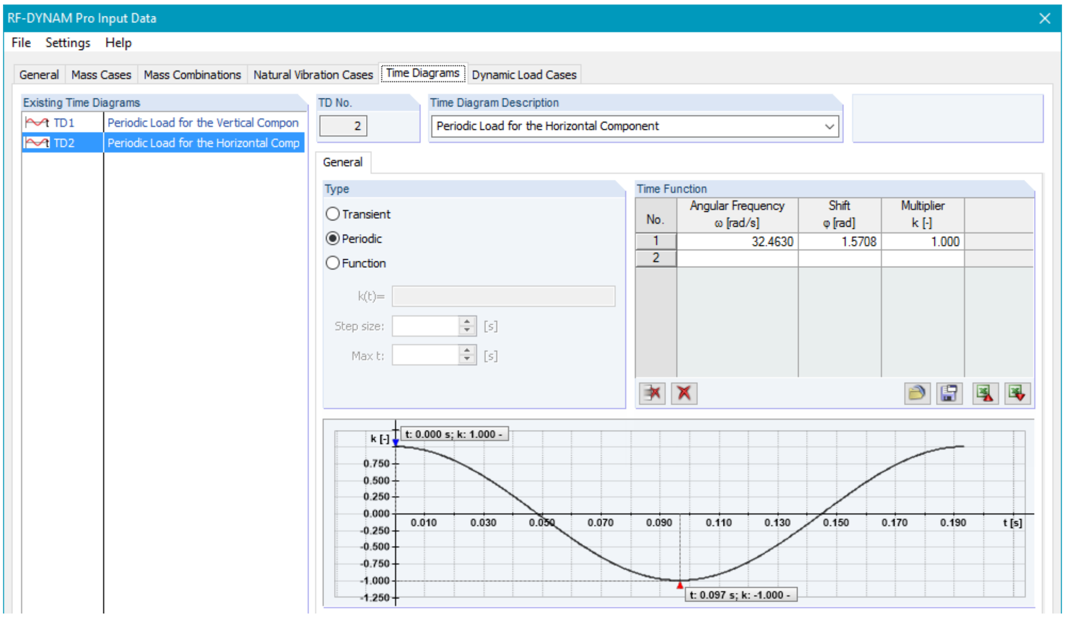



Figure 5.45: Definition of periodic time diagram for horizontal component of centrifugal force

## Time History Analysis

In the dynamic load case *DLC1*, the load cases *LC3* and *LC4* of RFEM are connected to the time diagrams *TD1* and *TD2* that have been defined in RF-DYNAM Pro. The *linear implicit Newmark solver* is chosen as the solving algorithm.

In the *Time History Analysis* tab, the time steps, the maximum time for calculation, and the export options are defined as shown in Figure 5.46.

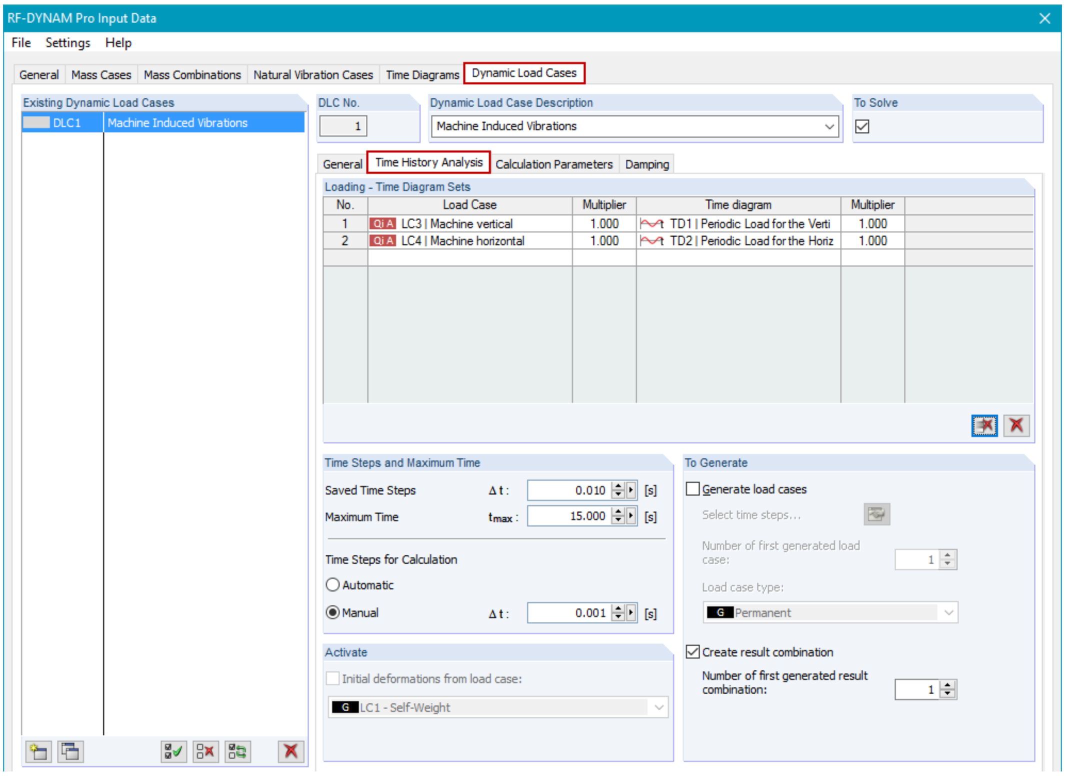



Figure 5.46: Settings for linear time history analysis of *DLC1* with connected load cases and time diagrams, time steps, and export options

The time step for calculation is set to $\Delta t = 1$ ms to achieve the required accuracy of the results.

The relevant frequency of the structure and those of the excitation are about 5.2 Hz. The required time step can be estimated with the following formula:

$$\Delta t = 1/(20\,f) = 9.6 \text{ ms} \tag{5.17}$$

To ensure also exact velocities and acceleration, the time step is chosen even smaller for this example.

In the *Calculation Parameters* sub-tab, the acting masses and the mass matrix options are set. This is shown in Figure 5.47. The *Diagonal matrix* considering all degrees of freedom is selected.

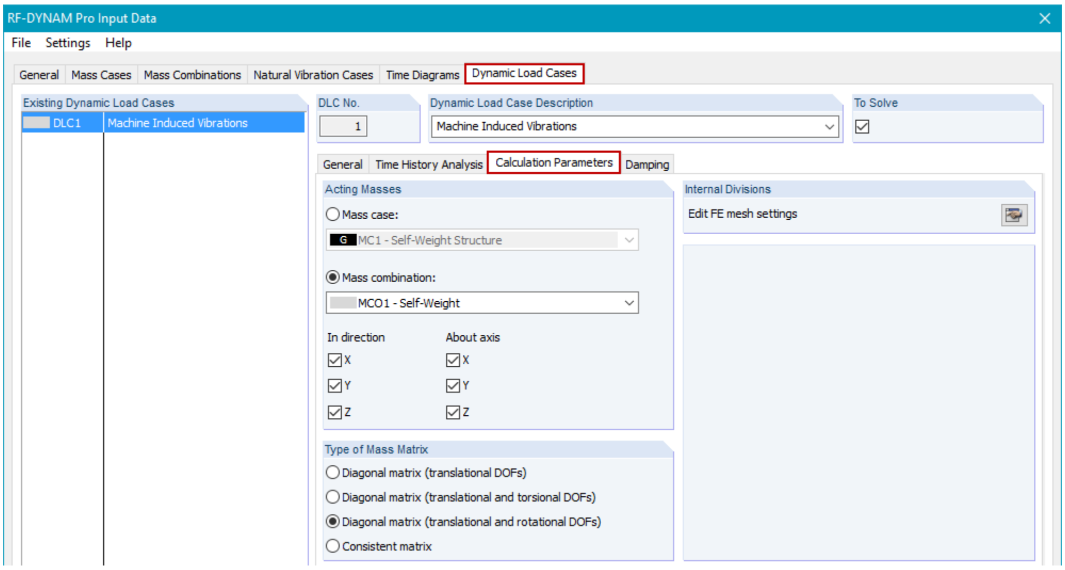



Figure 5.47: Settings of *DLC1* in *Calculation Parameters* tab for acting masses and mass matrix

In the *Damping* sub-tab, the structural damping is defined. The settings for damping used in this example are shown in Figure 5.48.

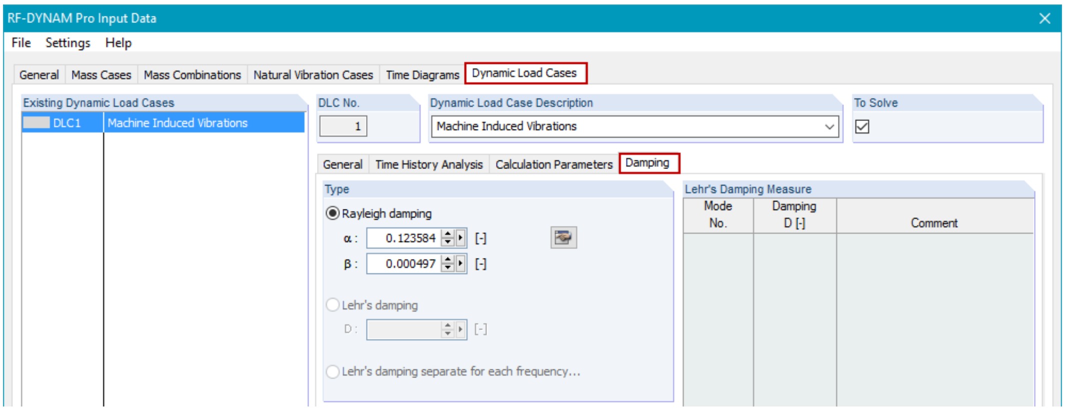



Figure 5.48: Settings of *DLC1* in *Damping* tab for Rayleigh coefficients $\alpha$ and $\beta$

The *linear implicit Newmark solver* requires *Rayleigh damping* coefficients. To convert available Lehr's damping values to Rayleigh coefficients, the following relation can be used:

$$D_i = \frac{\alpha}{2\,\omega_i} + \frac{\beta\,\omega_i}{2} \tag{5.18}$$

In this example, the most important eigenvalues in the X- and Y-directions shall be damped with D = 0.01. The relevant eigenvalues (*1* and *6*) have angular frequencies of $\omega_1 = 7.623$ rad/s and $\omega_6 = 32.620$ rad/s. In the *Lehr's to Rayleigh damping calculator* which can be accessed via the button, the angular frequencies and the damping values are defined as seen in Figure 5.49.

After [OK], the corresponding Rayleigh coefficients $\alpha = 1.235839\text{E}-01$ and $\beta = 4.9692\text{E}-04$ are set in the *Damping* tab (see Figure 5.48).

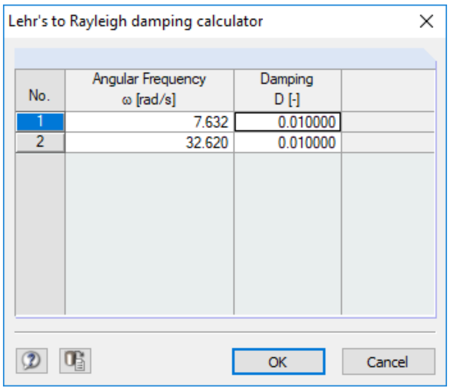


Figure 5.49: Settings in *Lehr's to Rayleigh damping calculator*

## Results in Case of Resonance

As stated earlier, the natural frequency in X-direction is very close to the excitation frequency, which causes resonance. The results are expected to be very large. The deformation of the structural system are illustrated in Figure 5.50. In the *Time Course Monitor*, the displacements in X-direction at Node 73 are displayed versus the time. The resonance can be clearly seen. The maximum displacement is about u = 145.2 mm

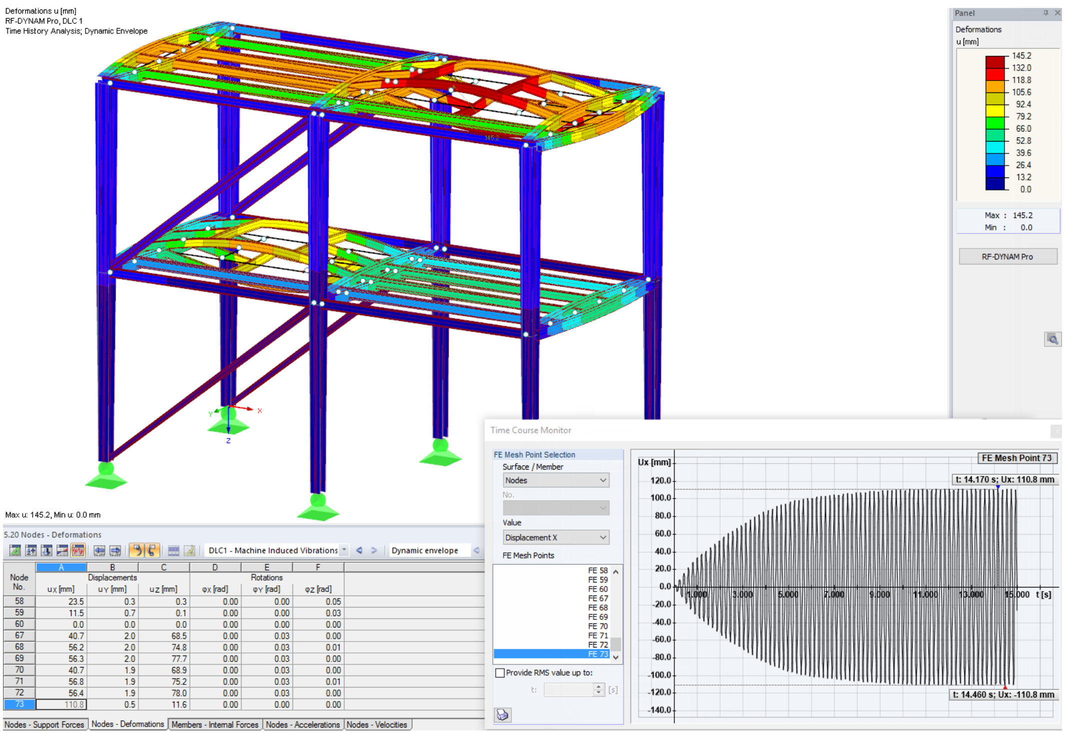


Figure 5.50: Displacement $u$ resulting from time history analysis with dynamic envelope in work window and displacement $u_X$ at node 73 displayed versus time in *Time Course Monitor* showing resonance

## Adjustment of Structural System

To avoid that kind of resonance case as seen in Figure 5.50, the cross-sections of the beams and columns are modified as illustrated in Figure 5.51. This is done to achieve a considerable shift of the dominant natural frequency.

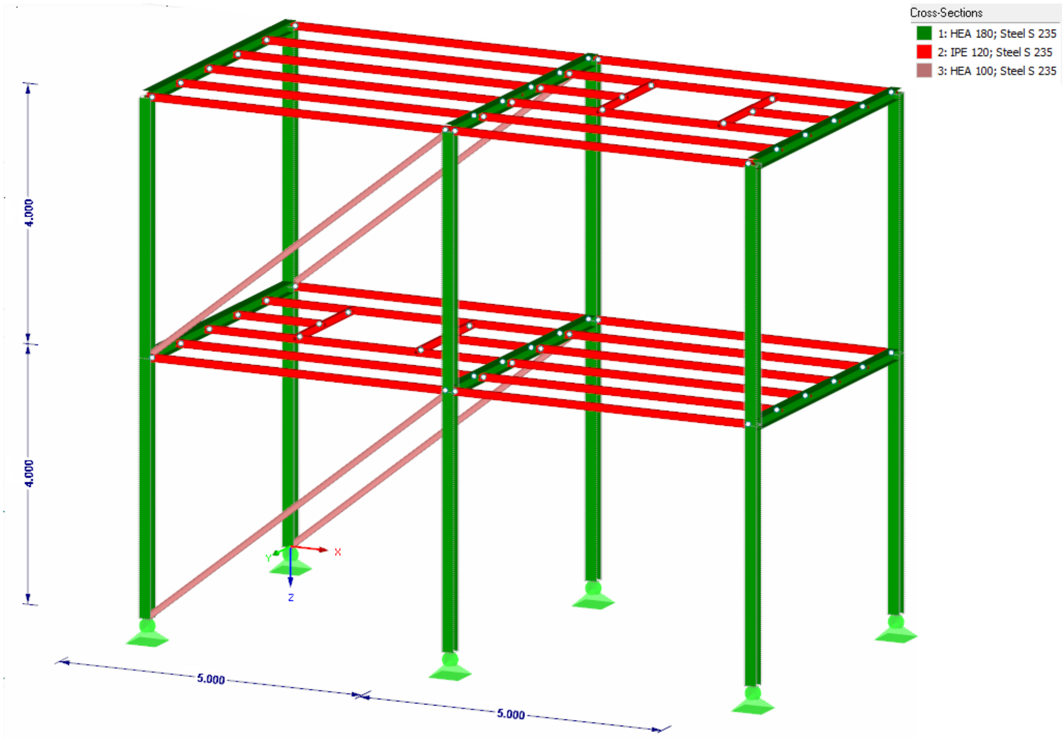



Figure 5.51: Structural system with modified cross-section of beams and columns

The changed frequencies of the structure are listed in Figure 5.52. The dominant eigenvalues are highlighted.

5.1 Natural Frequencies

NVC1 - Diagonal Matrix | Mode Shape 1 (f : 0.736 Hz)

| Mode No. | A<br>Eigenvalue<br>λ | B<br>Angular Frequency<br>ω [rad/s] | C<br>Natural frequency<br>f [Hz] | D<br>Natural period<br>T [s] | H<br>Effective Modal Mass Factor<br>$f_{meX}$ [-] | I<br><br>$f_{meY}$ [-] | J<br><br>$f_{meZ}$ [-] |
|---|---|---|---|---|---|---|---|
| 1 | 21.400 | 4.626 | 0.736 | 1.358 | 0.000 | 0.620 | 0.000 |
| 2 | 25.444 | 5.044 | 0.803 | 1.246 | 0.000 | 0.020 | 0.000 |
| 3 | 71.362 | 8.448 | 1.344 | 0.744 | 0.000 | 0.020 | 0.000 |
| 4 | 111.051 | 10.538 | 1.677 | 0.596 | 0.000 | 0.106 | 0.000 |
| 5 | 132.464 | 11.509 | 1.832 | 0.546 | 0.000 | 0.219 | 0.000 |
| 6 | 299.071 | 17.294 | 2.752 | 0.363 | 0.000 | 0.000 | 0.140 |
| 7 | 304.556 | 17.452 | 2.777 | 0.360 | 0.000 | 0.000 | 0.268 |
| 8 | 322.441 | 17.957 | 2.858 | 0.350 | 0.000 | 0.000 | 0.000 |
| 9 | 364.591 | 19.094 | 3.039 | 0.329 | 0.000 | 0.001 | 0.000 |
| 10 | 405.248 | 20.131 | 3.204 | 0.312 | 0.000 | 0.000 | 0.000 |
| 11 | 421.940 | 20.541 | 3.269 | 0.306 | 0.000 | 0.000 | 0.000 |
| 12 | 633.773 | 25.175 | 4.007 | 0.250 | 0.701 | 0.000 | 0.000 |
| Sum | | | | | 0.701 | 0.985 | 0.408 |

Natural Frequencies | Mode Shapes by Node | Mode Shapes by Member | Nodal Masses | Effective Modal Mass Factors

Figure 5.52: Eigenvalues of modified structural system showing dominant mode shape *12* in X-direction with frequency $f_{12} = 4.00$ Hz

The dominant frequency is now about $f = 4.00$ Hz and, therefore, shifted by 1.2 Hz compared to the critical frequency.

Additionnally, the Rayleigh damping coefficients need to be adjusted so that the dominant frequencies are damped with $D = 0.01$. With the modified frequencies, the coefficients are determined as $\alpha = 7.82 \times 10^{-02}$ and $\beta = 6.71 \times 10^{-04}$.

The other settings are kept as before. Then the time history analysis is performed again.

As a result, the displacements are much smaller. In the *Time Course Monitor*, the effect of damping can be well seen compared to the case of resonance. There the root mean square value (*RMS*), which is calculated from the displacement values from t = 0 s to t = 10 s, is shown as well.

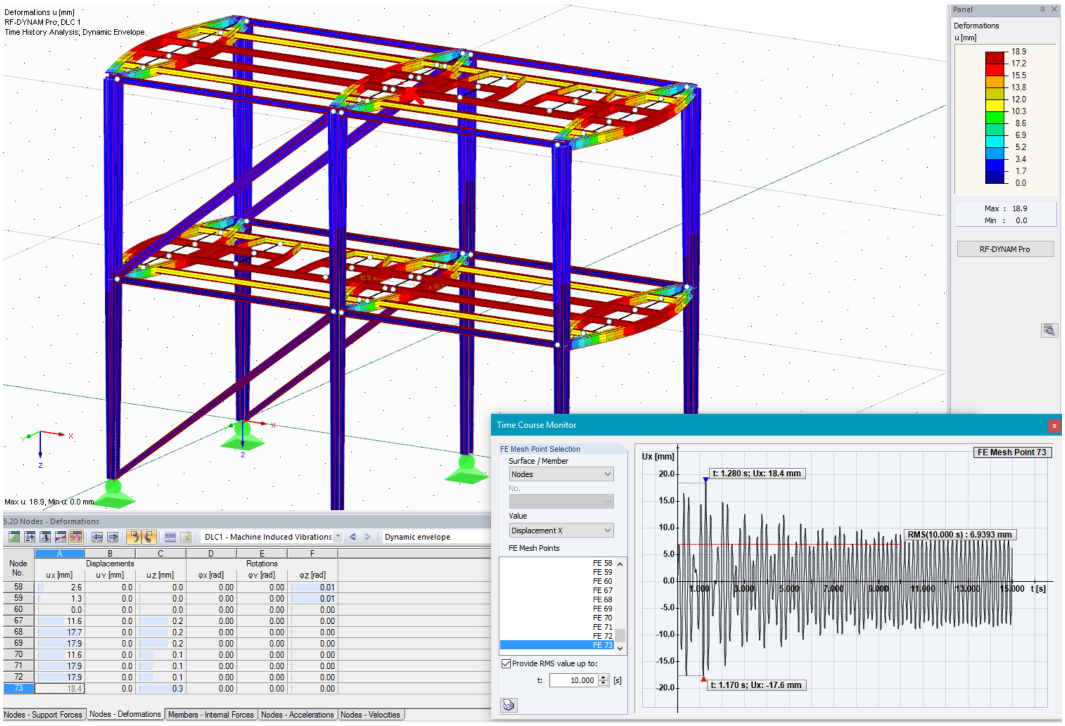


Figure 5.53: Displacement $u$ of altered structural system with dynamic envelope in work window and displacement $u_X$ at node 73 displayed versus time in *Time Course Monitor* and *RMS* value of 6.9 mm showing no resonance

This example demonstrates how to analyse machine-induced vibrations by RFEM and the add-on module RF-DYNAM Pro. It shows how important it is to analyse the frequencies of the structure in order to eliminate resonance cases.

# Literature


[1] *EN 1998-1: Design of structures for earthquake resistance Part 1: General rules, seismic actions and rules for buildings*. CEN, Brussels, 2004.

[2] *DIN 4149: Buildings in German earthquake areas - Design loads, analysis and structural design of buildings*. Beuth Verlag GmbH, Berlin, 2005.

[3] *IBC2012: International Building Code*. International Code Council, 2011.

[4] *EN 1990: Basis of structural design*. CEN, Brussels, 2002.

[5] Christian Barth and Walter Rustler. *Finite Elemente in der Baustatik-Praxis*. Beuth, Berlin, 2. edition, 2013.

[6] Klaus-Jürgen Bathe. *Finite Element Procedures*. Prentice Hall, 1996.

[7] Hans-Günter Natke. *Baudynamik*. B. G. Teubner, Stuttgart, 1989.

[8] Edward L. Wilson. *Three-Dimensional Static and Dynamic Analysis of Structures*. Computer and Structures, Inc., Berkeley, California, USA, 3rd edition, 2002.

[9] Joseph W. Tedesco, William G. McDougal and C. Allen Ross. *Structural Dynamics : Theory and Applications*. Addison-Wesley, 1st edition, 1999.

[10] A.K. Chopra. *Dynamics of Structures - Theory and Applications to Earthquake Engineering*. Prentice Hall, 2001.

[11] C. Katz. *Anmerkung zur Überlagerung von Antwortspektren*. D-A-CH Mitteilungsblatt, 2009.

[12] A.K. Gupta. *Response Spectrum Method in Seismic Analysis and Design of Structures*. CRC Press, 1992.

[13] E.L. Wilson, A. Der Kiureghian and E.P. Bayo. *A replacement for the SRSS method in seismic analysis.*, 1981.

[14] A. Der Kiureghian. *A response spectrum method for random vibration analysis of MDF system.*, 1981.

[15] Abaqus. *Abaqus 6.14 - Theory Guide*. Simulia., 2014.

[16] Abaqus. *Lecture 1 - Overview of ABAQUS/Explicit*. Abaqus Inc., 2001.

[17] Ivan Němec and Vladimír Kolář. *Finite Element Analysis of Structures - Principles and Praxis*. Shaker Verlag, Aachen, 2010.

[18] U.Stelzmann, C.Groth and G.Müller. *FEM für Praktiker - Band 2: Strukturdynamik*. Expert Verlag, 2008.

[19] ANSYS Inc.. *Theory Reference for the Mechanical APDL and Mechanical Applications, Release 15.0.*, 2013.

[20] K. Meskouris, K.-G. Hinzen, C. Butenweg and M.Mistler. *Bauwerke und Erdbeben*. Vieweg und Teubner, Berlin, 3. edition, 2011.

[21] *EN 1990: Grundlagen der Tragwerksplanung*. Beuth Verlag GmbH, Berlin, 2010.

[22] *Program Description RFEM 5*. DLUBAL GmbH, 2013.

[23] C. Petersen. *Dynamik der Baukonstruktionen*. Vieweg & Sohn, 1996.

[24] Manual SAP2000. *CSI Analysis Reference Manual - For SAP2000, ETABS, and SAFE2007.*, 2007.

[25] V. Červenka, L. Jendele and J. Červenka. *ATENA Program Documentation Part 1 - Theory*. Červenka Consulting s.r.o., 2014.

[26] *OENORM B4015:2007: Design loads in buildings - Accidental actions: Seismic actions*. Austrian Standards Institute, 2007.

[27] *NBC2010: National Building Code of Canada*. Canadian Commission on Building and Fire Codes, National Research Council of Canada, Thirteenth edition, 2010.

# Index