Version
September 2015

Add-on Module

RF-DYNAM PRO

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

Program Description

© Dlubal Software GmbH 2015
Am Zellweg 2
D-93464 Tiefenbach
Germany

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

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 three 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 and time history analysis of the given structure can be performed with the module *RF-DYNAM Pro - Forced Vibrations*. The add-on module *RF-DYNAM Pro - Equivalent Loads* offers 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 team from DLUBAL SOFTWARE GMBH

Natural Vibrations

RF-DYNAM Pro - Natural Vibrations determines the lowest eigenvalues of the structure. 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, linear, surface masses as well as inertia masses)
- Combination of masses in different mass cases and mass combinations
- Preset combination factors according to EN 1998-1 CEN
- Optional import of normal force distributions (for example for considering prestress)
- 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 analyze 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 RFEM 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 analyzed using the modal analysis or the direct integration. Furthermore, multi-modal and multi-point response spectra analysis can be performed. The required spectra can be created according to the standards or user-defined. 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 or direct integration in the time history analysis available
- Structural damping using the Rayleigh damping coefficients or the Lehr's damping values
- Direct import of initial deformations from a load case or a load combination possible
- 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], 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 EC 8 can be applied)
- Calculation is performed within *RF-DYNAM Pro* and is therefore linear
- 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 history diagram. 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 Excel. In case of a time history analysis, you can export results of a single time step or filter the most unfavorable results of all time steps. In case of a response spectra 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 into *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 RFEM add-on module *RF-DYNAM Pro - Natural Vibrations*. You can perform seismic analyses 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.

- Response spectra of numerous standards (EN 1998 [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 EC 8 can be applied)
- Generated equivalent static loads are exported into 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.
- Combination of the modal responses (*SRSS* rule) and combination of the results from different excitation directions (*SRSS* or 100% / 30% rule)

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 subsequently superimposed in result combinations. Combination of the modal results takes place first. Afterwards, the results of different excitation directions are combined.

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

1.2 RF-DYNAM Pro Team

The following people were involved in the development of *RF-DYNAM Pro*:

Program coordination

Dipl.-Ing. Georg Dlubal
Dipl.-Ing. (FH) Younes El Frem
Ing. Pavel Bartoš
Dr. M.Sc. Dipl.-Ing. (FH) Gerlind Schubert
Dipl.-Ing. Stefan Frenzel

Programming

Doc. Dr.-Ing. Ivan Němec
Dr.-Ing. Radoslav Rusina
Dr.-Ing. Zbyněk Vlk
Dr.-Ing. Ivan Ševčik
Ing. Petr Horák
Ing. Radek Dubina
Ing. Jiří Buček
Ondřej Vydra
Ján Juranko
Michal Zelenka
Ing. Michal Brabec
RNDr. Jan Gregor

Program design

Dipl.-Ing. Georg Dlubal
MgA. Robert Kolouch
Ondrej Vydra

Program testing

M.Eng. Dipl.-Ing. (FH) Walter Rustler
M.Eng. Dipl.-Ing. (BA) Andreas Niemeier
Ing. Jonáš Bartoň
Dr. M.Sc. Dipl.-Ing. (FH) Gerlind Schubert
Dipl.-Ing. Stefan Frenzel

Manual, Help System, and Translation

Dr. M.Sc. Dipl.-Ing. (FH) Gerlind Schubert
BSc Eng Chelsea Jennings
M.A. Melanie Most
Dipl.-Ü. Gundel Pietzcker
Dipl.-Ing. (FH) Robert Vogl
Oliver Vogl
Ing. Nikola Tomšů
Ing. Ladislav Kábrt
Ing. Fabio Borriello
Ing. Dmitry Bystrov
Eng. Rafael Duarte
Ing. Jana Duníková
Ing. Lara Caballero Freyer
Ing. Ph.D. Alessandra Grosso
Ing. Aleksandra Kociołek
Eng. Nilton Lopes Fernandes
Mgr. Ing. Hana Macková
Ing. Téc. Ind. José Martínez Hernández
Mgr. Petra Pokorná
Ing. Michaela Prokopová
Ing. Marcela Svitáková

Technical support and final testing

M.Eng. Cosme Asseya
Dipl.-Ing. (BA) Markus Baumgärtel
Dipl.-Ing. Moritz Bertram
M.Sc. Sonja von Bloh
Dipl.-Ing. (FH) Steffen Clauß
Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) René Flori
Dipl.-Ing. (FH) Stefan Frenzel
Dipl.-Ing. (FH) Walter Fröhlich
Dipl.-Ing. (FH) Wieland Götzler
Dipl.-Ing. Thomas Günthel
Dipl.-Ing. (FH) Sebastian Hawranke
Dipl.-Ing. (FH) Paul Kieloch
Dipl.-Ing. (FH) Bastian Kuhn
Dipl.-Ing. (FH) Adrian Langhammer
Dipl.-Ing. (FH) Ulrich Lex
Dipl.-Ing. (BA) Sandy Matula
Dipl.-Ing. (FH) Alexander Meierhofer
M.Eng. Dipl.-Ing. (BA) Andreas Niemeier
M.Eng. Dipl.-Ing. (FH) Walter Rustler
M.Sc. Dipl.-Ing. (FH) Frank Sonntag
Dr. M.Sc. Dipl.-Ing. (FH) Gerlind Schubert
Dipl.-Ing. (FH) Christian Stautner
Dipl.-Ing. (FH) Lukas Sühnel
Dipl.-Ing. (FH) Robert Vogl

1.3 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. The present manual 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 dialog boxes, windows, and menus are set in *italics* making the connection between the explanations in the manual and the program clearer.

At the end of the manual, you find the index. If you still cannot find what you are looking for, check our website www.dlubal.com, where you can go through the *FAQ* pages, watch the recorded webinars, or study the examples provided. The *Dlubal Blog* is also highly recommended, you find that under www.dlubal.com/blog/en/.

1.4 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, you can select on the RFEM menu **Add-on Modules** → **Dynamic** → **RF-DYNAM Pro**, shown in Figure 1.1.

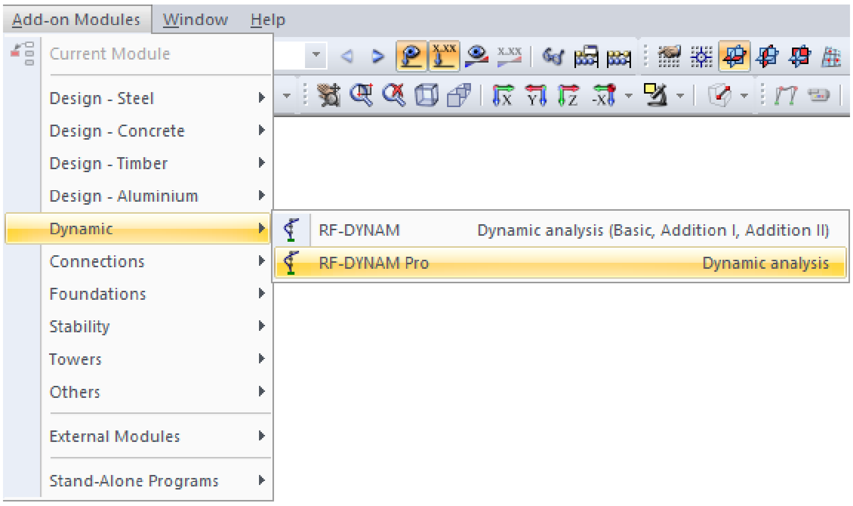


Figure 1.1: Menu **Add-on Modules** → **Dynamic** → **RF-DYNAM Pro** to open the module *RF-DYNAM Pro*

When *RF-DYNAM Pro* was open before and is your current module, you can also use the menu **Add-on Modules** → **Current Module**.

Navigator

Alternatively, you can open the add-on module in the *Data Navigator* by clicking

Add-on Modules → **RF-DYNAM Pro**

When you right-click on the add-on module you can add it to your favorites. The project navigator is shown in Figure 1.2.

Panel

When the results from *RF-DYNAM Pro* are already available, you can also open the add-on module from the panel.

To see the panel you have to choose the *RF-DYNAM Pro* case from the drop-down menu in the main program RFEM, make the results visible with the [Show Results] button and display the panel by clicking the [Panel] button.

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

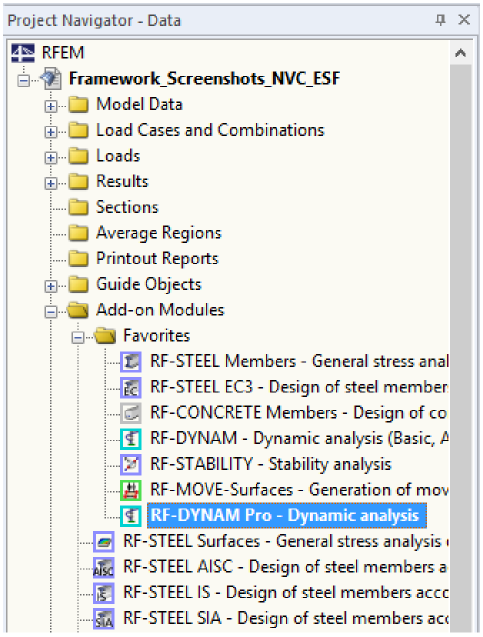


Figure 1.2: Data navigator **Add-on Modules** → **RF-DYNAM Pro** to open the module *RF-DYNAM Pro*

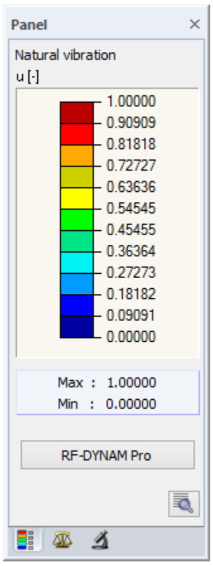


Figure 1.3: Panel with [RF-DYNAM Pro] button shown in Figure 1.3 to re-open the add-on module.

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 your input 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 select the corresponding check boxes.

The first window that appears when you open the module is shown in Figure 2.1.

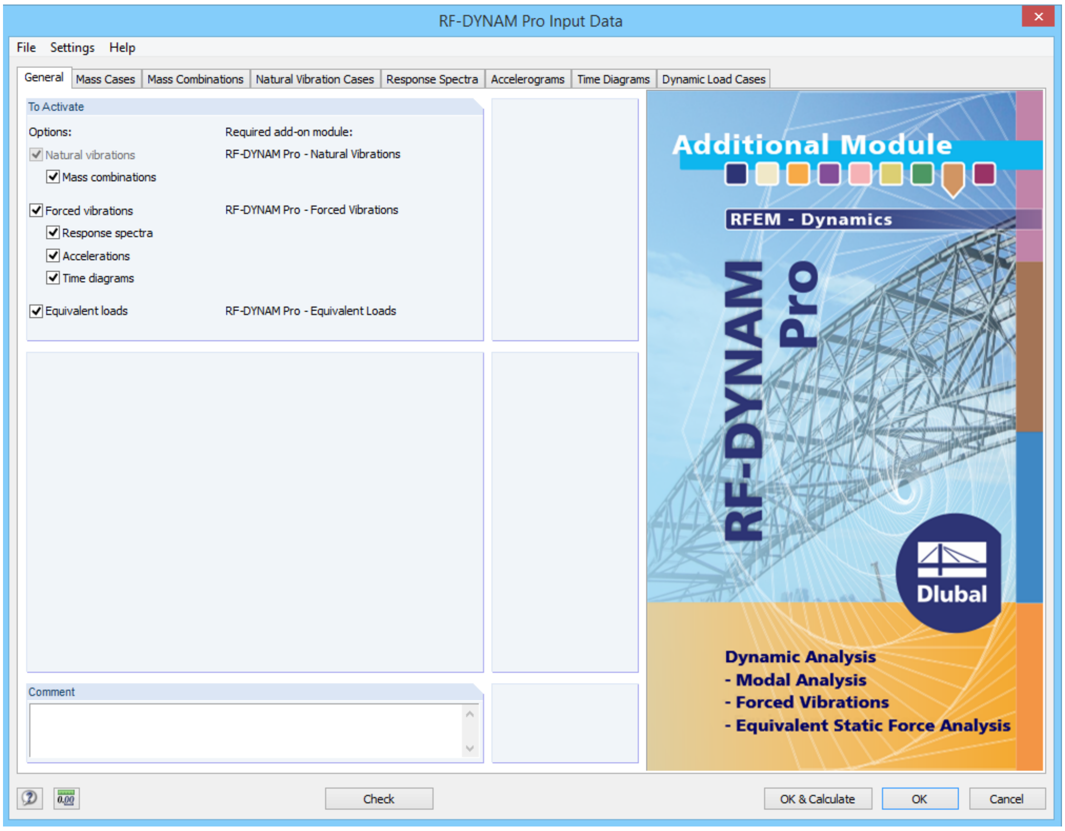


Figure 2.1: Module window *General Data*.

In the *General* tab you decide whether you want to analyze only natural frequencies and mode shapes of the structural system (*RF-DYNAM Pro - Natural Vibrations*) or whether you want to analyze the response of the system to forced vibrations (*RF-DYNAM Pro - Forced Vibrations*) or whether you want to generate equivalent static forces (*RF-DYNAM Pro - Equivalent Loads*). Add-on modules that were not purchased can still be opened but run only as a demo version. You can also activate a 30-day trial version of those add-on modules.

Natural Vibrations

This option is always selected because a natural vibration analysis of the structure is mandatory. The tabs *Mass Cases* and *Natural Vibration Cases* belong to this option by default. When you select the *Mass combination* check box another tab appears as shown in Figure 2.1.

Forced Vibrations

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

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* tab belong to the module *RF-DYNAM Pro - Equivalent Loads*.

2.1 Structure of the Main Tab Windows

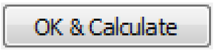
The windows of *RF-DYNAM Pro* always contain the following buttons:

The [Help] button gives direct access to the manual and further online 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* window as known from the main *RFEM* program. This window can also be reached by the **Settings** → **Units and Decimal Places...** menu. This is further discussed in Sections 2.10 and 4.6.

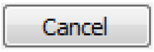
In the *Details* dialog box you define global settings that apply to whole dynamic calculation performed in *RF-DYNAM Pro*. This dialog box is also accessible by **Settings** → **Details**. These settings are explained in Section 2.9.

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 but not perform the calculation press [OK]. By doing so you exit the *RF-DYNAM Pro* module and return to the main program.



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 description, this is marked with an orange box in Figure 2.2. These 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 buttons to create new cases, copy existing cases or delete existing cases.

You have 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 with 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 this, you find the main entering area for your input data which firstly opens in the *General* sub-tab. Eventually more sub-tabs might appear depending on the selected check boxes.

In the *General* sub-tab you have some space to enter comments; this space is marked with 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.

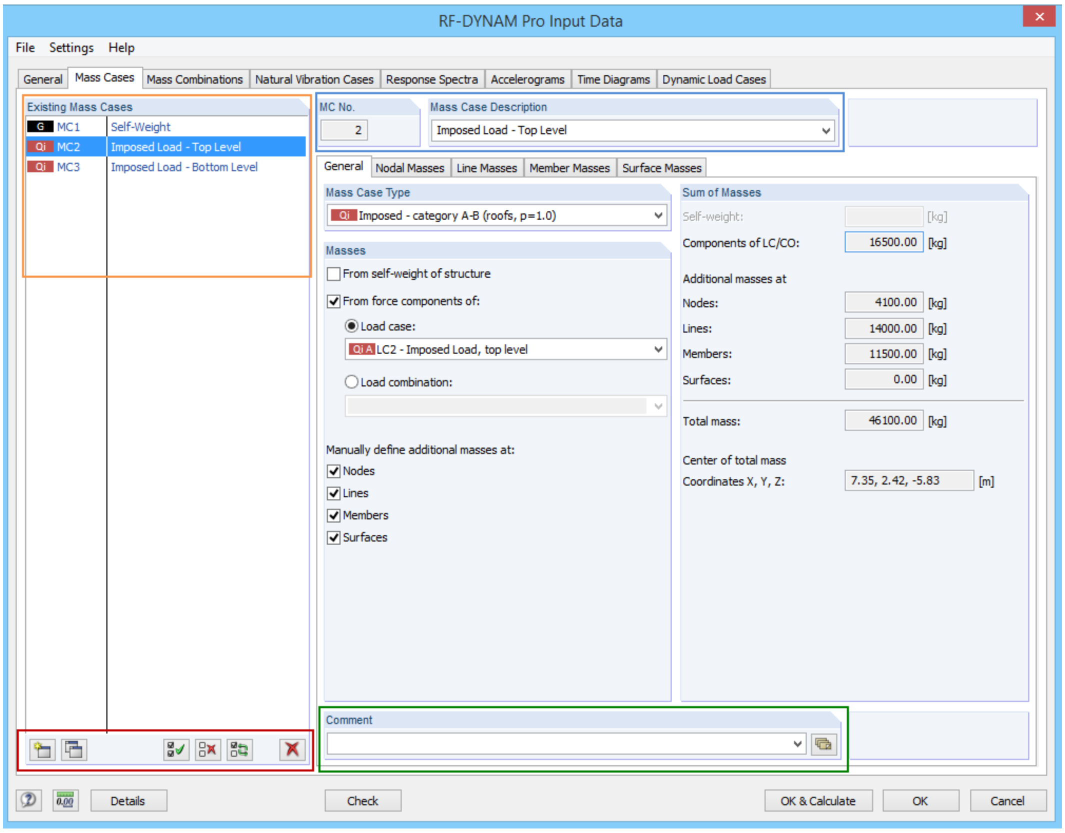
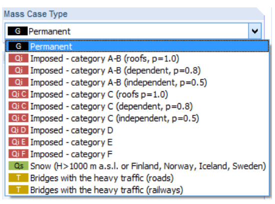


Figure 2.2: Module window *Mass Cases* with the *General* tab open. The MC2 is selected and all four options to define additional masses are selected to show the appearing sub-tabs.

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.

Mass Case Type

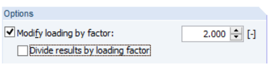


Choose one of the mass case types from the drop-down list. This is especially important when you use *Mass Combinations* (see Section 2.3). Combination factors are then preset in accordance with **EC0** [4] and **EC8** [1].

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 activating the check box *from force components of load case / load combination*. Choose the load case or load combination to be imported from the drop-down menu.



The factor to modify loads of load cases, which can be set in the calculation parameters of RFEM, is not overtaken into *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 structure make sure the `*from self-weight of structure*' check box in *RF-DYNAM Pro* is cleared. Otherwise you double the self-weight of the structure.

In **Settings** → **Details** 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 *Details* dialog box is explained in Section 2.9.

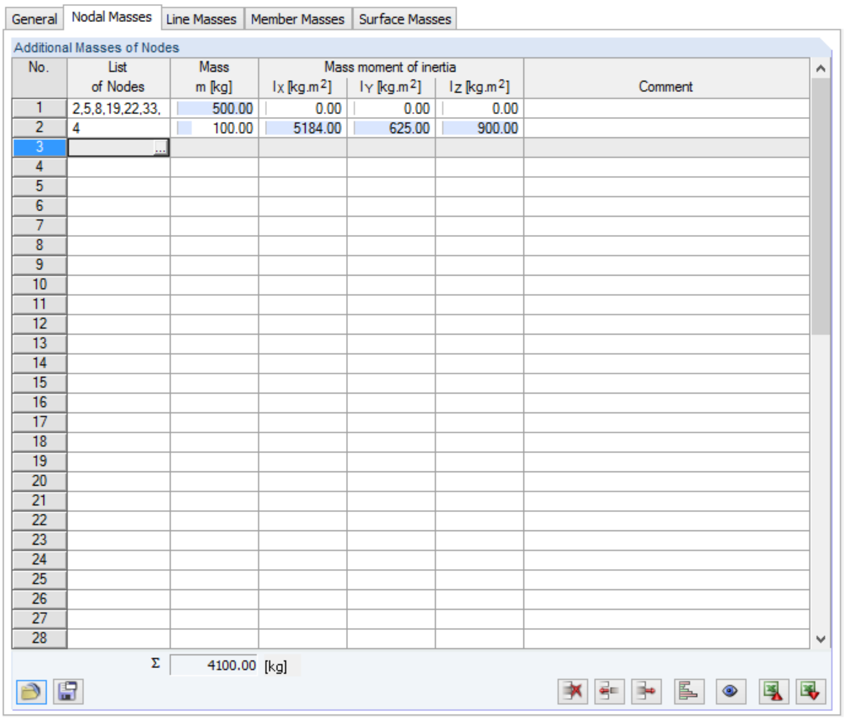
In addition, or as an alternative to the previously described options to import masses, it is possible to define nodal, line, member, or surface masses. Depending on the check boxes you select additional sub-tabs appear as shown in Figure 2.2. The settings within those sub-tabs are detailed in the Sections 2.2.1 and 2.2.2.

Sum of Masses

On the right hand side of the window the sums of masses are provided to double-check the input of self-weight, imported masses from load cases or combinations, and additional masses. The total mass and the resulting center of mass are provided.

2.2.1 Additional Nodal Masses

The nodal mass tab is shown in Figure 2.3.



No.	List of Nodes	Mass m [kg]	Mass moment of inertia Ix [kg.m²]	Iy [kg.m²]	Iz [kg.m²]	Comment
1	2,5,8,19,22,33,	500.00	0.00	0.00	0.00	
2	4	100.00	5184.00	625.00	900.00	
3						
4						
5						
6						
7						
8						
9						
10						
11						
12						
13						
14						
15						
16						
17						
18						
19						
20						
21						
22						
23						
24						
25						
26						
27						
28						
Σ		4100.00 [kg]				

Figure 2.3: Module window *Nodal Masses* which is available when *additional masses at nodes* in the *General* tab are selected.

You can provide a list of nodes manually or by using the [Multiple Selection] button shown on the left to select nodes in the graphic.

The masses $m\ [kg]$ can only be defined manually and act in the directions defined in the *Natural Vibration Cases* (see Section 2.4).

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

The buttons on the bottom of the table provide usual table functions as described in the ***RFEM* manual** in **Section 11.5**.

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

2.2.2 Additional Line, Member and Surface Masses

In addition to nodal masses you can provide line, member or surface masses manually. The corresponding sub-tabs appear when the check boxes *Lines*, *Members* or *Surfaces* in the *Mass Case* tab shown in Figure 2.2 are selected. The table to enter the line masses is shown in Figure 2.4.

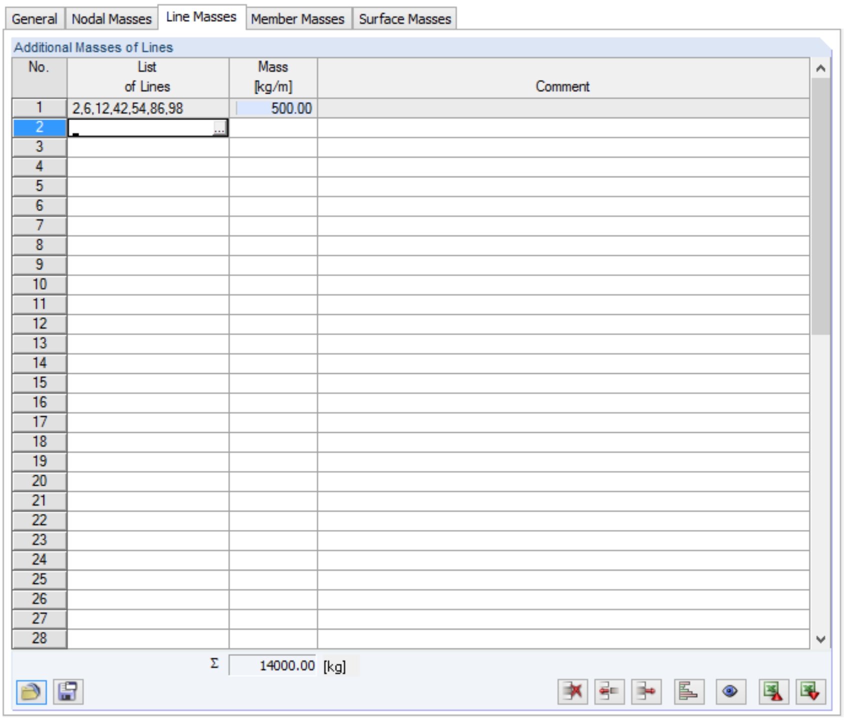


Figure 2.4: Module window *Line Masses* which is available when *additional masses at lines* in the *General* tab are selected.

The tables to enter member and surface masses appear very similar and are not explicitly shown here.

A list of lines (members or surfaces) can be entered manually or by using the [Multiple Selection] button shown on the left to select lines (members or surfaces) graphically.

Line and member masses are provided in kg per unit length. Surface masses are given in kg per unit area. They can only be defined manually and act in the directions defined in the *Natural Vibration Cases* (see Section 2.4).

Masses are only considered in translational directions when a diagonal mass matrix is chosen for the calculation (except the mass moments of inertia as detailed in Section 2.2.1). Rotation about the longitudinal axis of a line or member is taken into account when a diagonal mass matrix with torsional elements is chosen. Only with the consistent mass matrix, masses can act in translational and rotational directions. See Section 2.4 for further details about the mass matrices.

2.3 Mass Combinations

Mass cases can be combined to mass combinations. This is done in analogy to load cases and load combinations in the main program *RFEM* as is described in the ***RFEM* manual** in **Section 5.5.1**. The *Mass Combination* tab only exist when the corresponding check box in the *General* tab (see Figure 2.1) is selected. When you open the *Mass Combination* window the first time, a mass combination is preset and the existing mass cases are listed.

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 preset 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 module window with mass combination *MCO1* containing self-weight and imposed load mass cases is illustrated in Figure 2.5.

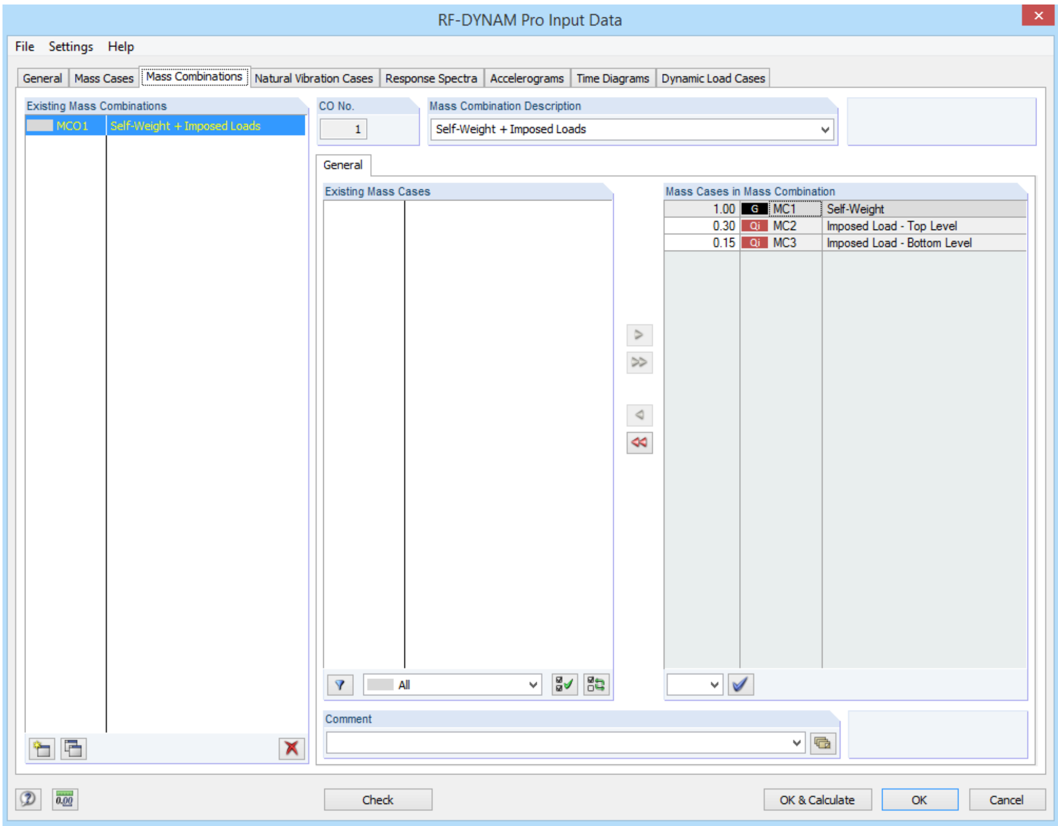
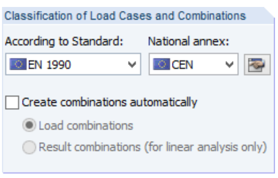


Figure 2.5: Module window *Mass Combinations* with all existing mass cases selected for the mass combination. The combination factors are preset by *RF-DYNAM Pro*.



The combination factors are preset in accordance to **EC0** [4] and **EC8** [1]. Those factors can be adjusted manually by entering a value or choosing a value from the drop-down list. As regulated in **EN 1998-1** in **Section 3.2.4** [1] additional masses beside the self-weight have to be considered to calculate inertia effects.

$$\sum G_{k,j} \text{``+''} \sum \psi_{E,i} \cdot Q_{k,i} \tag{2.1}$$

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

$$\psi_{E,i} = \varphi \cdot \psi_{2,i} \tag{2.2}$$

where $\psi_{2,i}$ are regulated in **EN1990 Table A.1.1** [4] and φ are provided in **EN 1998-1 Section 4.2.4** [1].

These combination factors might be regulated differently in each national annex of **EC8** and in other international building standards.

2.4 Natural Vibration Cases

The *Natural Vibration Case* tab is the centerpiece of the module *RF-DYNAM Pro - Natural Vibrations* and a very important basis for the two additional modules. Here, 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 any stiffness modification or import axial forces as an initial condition.

The *General* tab of the *Natural Vibration Case* window is shown in Figure 2.6.

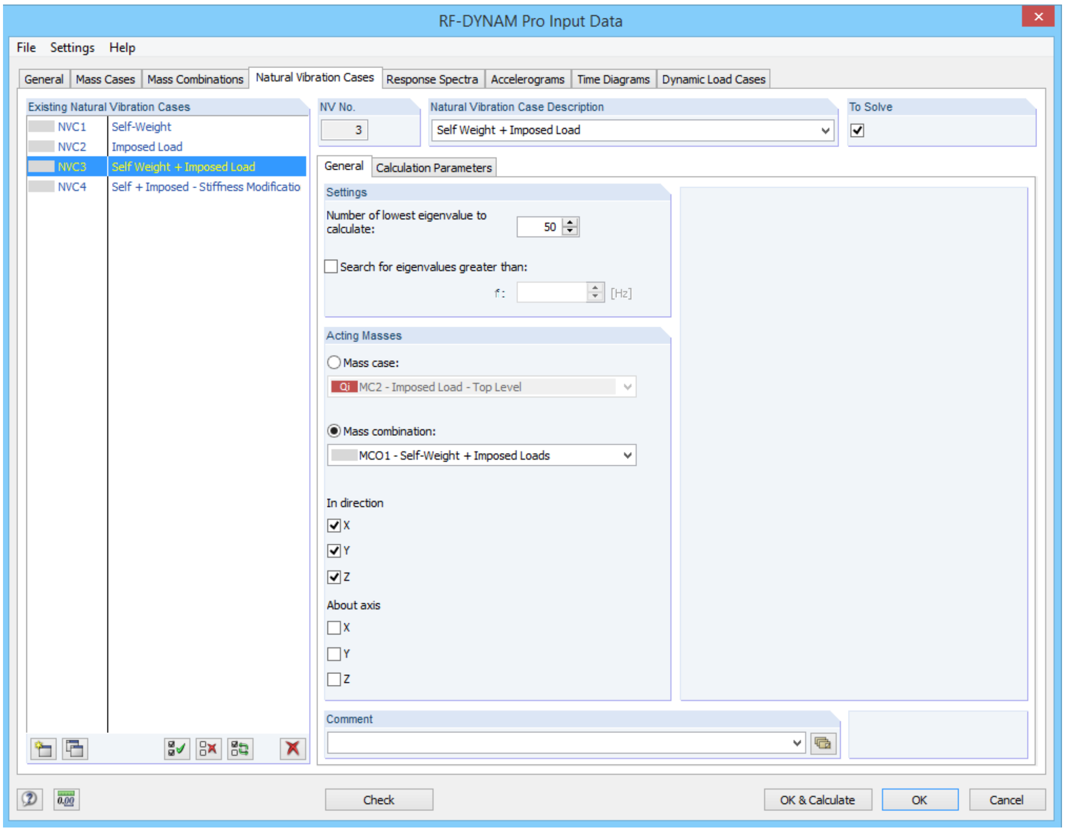
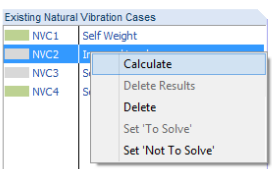


Figure 2.6: Module window *Natural Vibration Cases* with the *General* tab open.

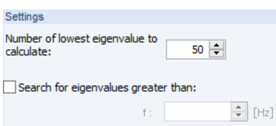
For each *Natural Vibration Case* you can set various calculation parameters as shown in Figure 2.7

The *Natural Vibration Case* number is set automatically and cannot be edited. When a case is deleted later the numbers do not change. You can decide whether or not the specific *Natural Vibration Case* is solved by selecting or clearing the check box *To Solve*.



You can calculate each *Natural Vibration Case* separately by using the context menu and apply *Calculate*. The colour of a *NVC* is grey when no results of this *NVC* are available and turns green as soon as the calculation finished and results are available, more information are provided in Section 3.3.

2.4.1 Number of Eigenvalues



In the *General* tab you set the number of lowest eigenvalues to be calculated. The maximum number of eigenvalues is limited to 9999 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).

You can calculate eigenvalues only above a certain value of natural frequency f to reduce the number of produced results.

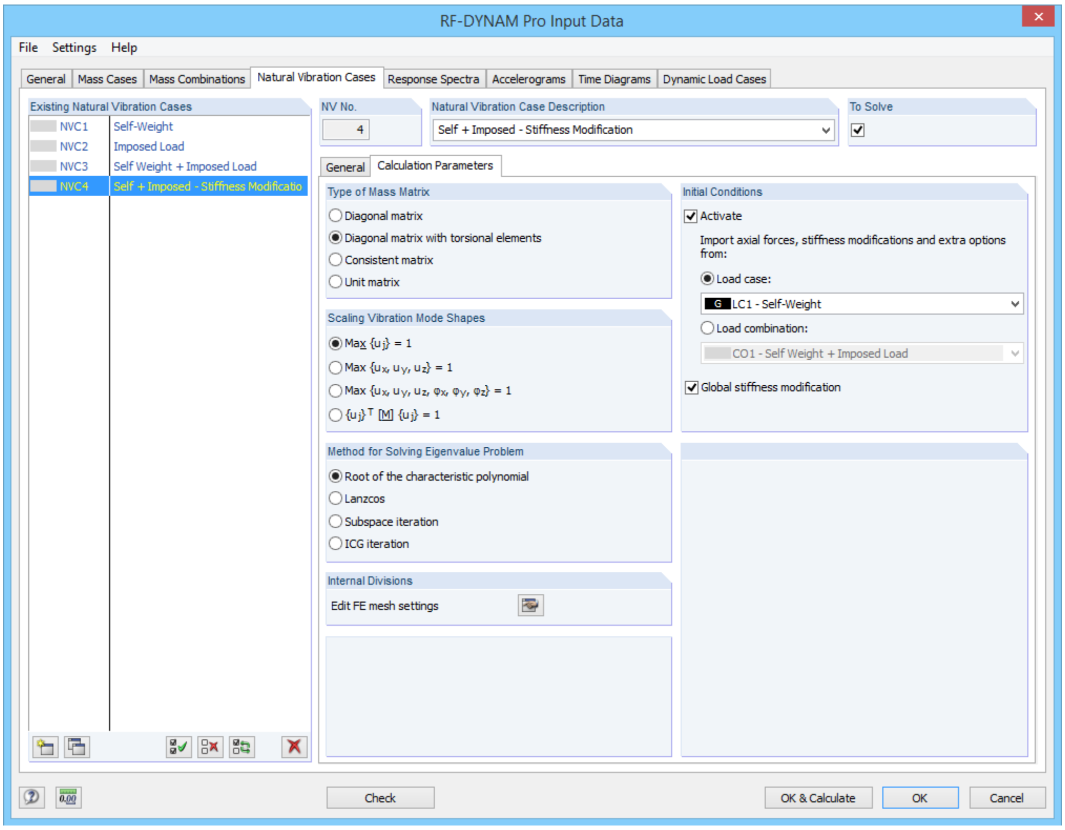
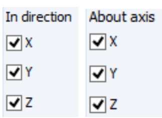


Figure 2.7: Module window *Natural Vibration Cases* with the *Calculation Parameter* tab open.

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).

2.4.2 Acting Masses

For each *Natural Vibration Case* you can import different *Mass Cases* or *Mass Combinations*. Please choose from the drop-down menu in the *General* tab.

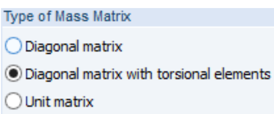


You must define the direction in which the masses are acting. The masses can act in the global translational X, Y, or Z-direction by activating the corresponding check boxes. Masses can act rotationally about the global X, Y, and Z-axis by activating the corresponding check boxes.

These settings change your mass matrix, and you 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 direction. Only when your are interested in rotational mode shapes you should consider rotating masses.

Rotational masses are only considered when the type of mass matrix is set to *diagonal with torsional elements* or *consistent*. When using a diagonal mass matrix, the activation of *about axis* has no influence on the results unless you defined nodal mass moments of inertia (see Section 2.2.1).

2.4.3 Type of Mass Matrix



Four different types of mass matrices are available in the *Calculation Parameter* tab as shown in Figure 2.7.

Diagonal Mass Matrix: When the type of the mass matrix $\mathbf{M}$ is chosen to be diagonal the masses are lumped to the FE-nodes. The entries in the matrix are the masses acting on each FE-node in the translational directions X, Y, and Z. Masses rotating about the X, Y or Z- axis are neglected even when the *about axis* check boxes in the *General* tab are selected. Only mass moments of inertia I_X, I_Y and I_Z as provided in the *Nodal Mass* table (see Section 2.2.1) are considered. The diagonal mass matrix $\mathbf{M}$ is structured as follows,

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

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

Diagonal Mass Matrix with Torsional Elements: This matrix is still a diagonal matrix where the masses are lumped to the FE-nodes but the masses acting about the longitudinal axis of a member or surface are taken into account automatically when acting masses about the X, Y and Z-axis are activated. The direction that is taken into account depends on the local longitudinal direction of the members and surfaces. The diagonal mass matrix $\mathbf{M}$ on a specific FE-point with mass m is structured as follows:

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

with X, Y, or Z being the distances to the center of total mass provided in the *Mass Case* tab described in Section 2.2.

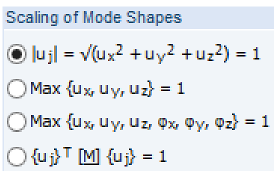
Consistent Mass Matrix: The consistent mass matrix is a full mass matrix of the finite elements, so the masses are not simply lumped to the FE-nodes, instead shape functions are used for a more realistic distribution of the masses within the FE-elements. With the consistent mass matrix non-diagonal entries 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):

$$\mathbf{M} = m \cdot \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & Y^2 + Z^2 & -XY & -XZ \\ 0 & 0 & 0 & -XY & X^2 + Z^2 & -YZ \\ 0 & 0 & 0 & -XZ & -YZ & X^2 + Y^2 \end{bmatrix} \tag{2.5}$$

with the mass m and the distances X, Y, or Z to the center 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 (see Section 2.4.2). 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].

2.4.4 Scaling of Mode Shapes

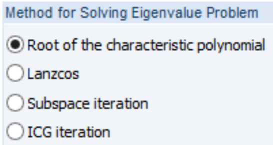


The mode shapes can be scaled to any arbitrary value. The first three options are all a good choice for a satisfying illustration of the mode shapes. The option $|\mathbf{u}_j| = \sqrt{u_x^2 + u_y^2 + u_z^2} = 1$ scales the magnitude of the mode shape vector $\mathbf{u}_j$ (only translational parts) 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 j.

The option $\mathbf{u}_j^T \mathbf{M} \mathbf{u}_j = 1\ kg$ is always used internally for time history or response spectrum analysis 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 φ_X, φ_Y and φ_Z are provided in $[1/m]$. The resulting mode shapes are discussed in Section 4.1.

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.

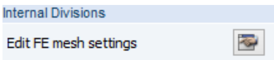
Root of the characteristic polynomial: This is the analytic solution of an eigenvalue problem, see for example [6] or [7] and is solved using a direct method. The main advantage of this method is the precision of higher eigenvalues and that all eigenvalues of a structure can be determined. For large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

Method by Lanczos: The method by Lanczos is an iterative method to determine the p lowest eigenvalues and corresponding mode shapes of large structures. 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 with this method. For further details see Bathe [6].

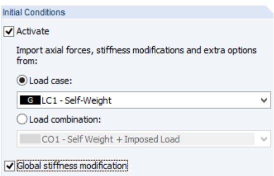
ICG Iteration: This method is also suitable for large structures where only a few eigenvalues are required. But here the eigenvalues are calculated successively. Therefore, the number of required eigenvalues is proportional to the computing time. Theoretically, all eigenvalues of a structure can be calculated.

2.4.6 FE Mesh Settings



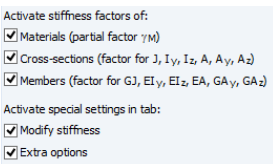
The button shown on the left links directly to the FE mesh settings of the main *RFEM* program. The *FE Mesh Settings* dialog box is also available under **Calculation** $\rightarrow$ **FE Mesh Settings...** in the main program *RFEM*. Further details about the FE mesh and parameters that can be adjusted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

2.4.7 Stiffness Modifications and Axial Forces as Initial Conditions



Axial forces, stiffness modifications and extra options as set in any load case or load combinations are imported as initial condition into *RF-DYNAM Pro* when the *Initial Conditions* in Figure 2.6 are *activated*.

The *global stiffness modifications* are only available when no load case is imported or when the imported load case or combination does not contain any stiffness modifications. *Global stiffness modifications* import the material partial factor γ_M set in the *Edit Material* dialog box in the main program *RFEM* (see ***RFEM* manual Section 4.3**), cross-section modifications as set in the *Edit Cross-Section* dialog box in the main program *RFEM* (see ***RFEM* manual Section 4.13**), and stiffness modifications of members as set in the *Edit Member* dialog box (see ***RFEM* manual Section 4.17**).



The *Calculation Parameters* of a load case or combination that modify the stiffness are shown on the left hand side. The check boxes *Materials*, *Cross-Sections*, and *Members* activate the modifications made in the material, cross-section, and member dialog boxes as described above for the *global stiffness modifications*. When you select *modify stiffness* an additional tab in the *Calculation Parameter* dialog box opens that allows you to enter factors for materials, cross-sections, supports and hinges. In addition to that, members, surfaces and solids can be deactivated which leads to a different structural system that is imported into *RF-DYNAM Pro*. When *extra options* are selected the stiffness from the add-on module *RF-CONCRETE* can be imported. Further details about the calculation parameters of load cases are given in the ***RFEM* manual** in **Section 7.3**.

Axial forces and also failing supports, members, surfaces or solids are imported automatically (without activating any stiffness modifications in the *LCs / COs*) as soon as the *initial conditions* are selected and the corresponding load case or combination is imported.

To summarize, *RF-DYNAM Pro* imports the stiffness matrix of the structural system after the load case or the load combination has been calculated in *RFEM*. This is an initial condition for the eigenvalue analysis performed in *RF-DYNAM Pro*, the calculation itself is still linear. With this option, you can approximately consider nonlinear effects, as for example, failing members, supports and hinges. If no results for the load case or load combination exist, the calculation is performed automatically before the *RF-DYNAM Pro* module calculation.

When you want to consider axial forces only (for example prestress in cables) make sure that no stiffness modifications are activated in the calculation parameters of the imported load case or combination.

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

A complex figure of this feature is provided in the *Dlubal Blog* (www.dlubal.com/blog/16774/dlubal-rfem-5-rstab-8-import-of-axial-forces-stiffness-modifications-and-extra-options-in-rf-dynam-pro-natural-vibrations/).

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 still 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* is also able to 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* are 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*.

The *Response Spectra* tab is illustrated in Figure 2.8.

In *RF-DYNAM Pro* you have three options to enter response spectra: according to a building standard, user-defined or generated from an accelerogram.

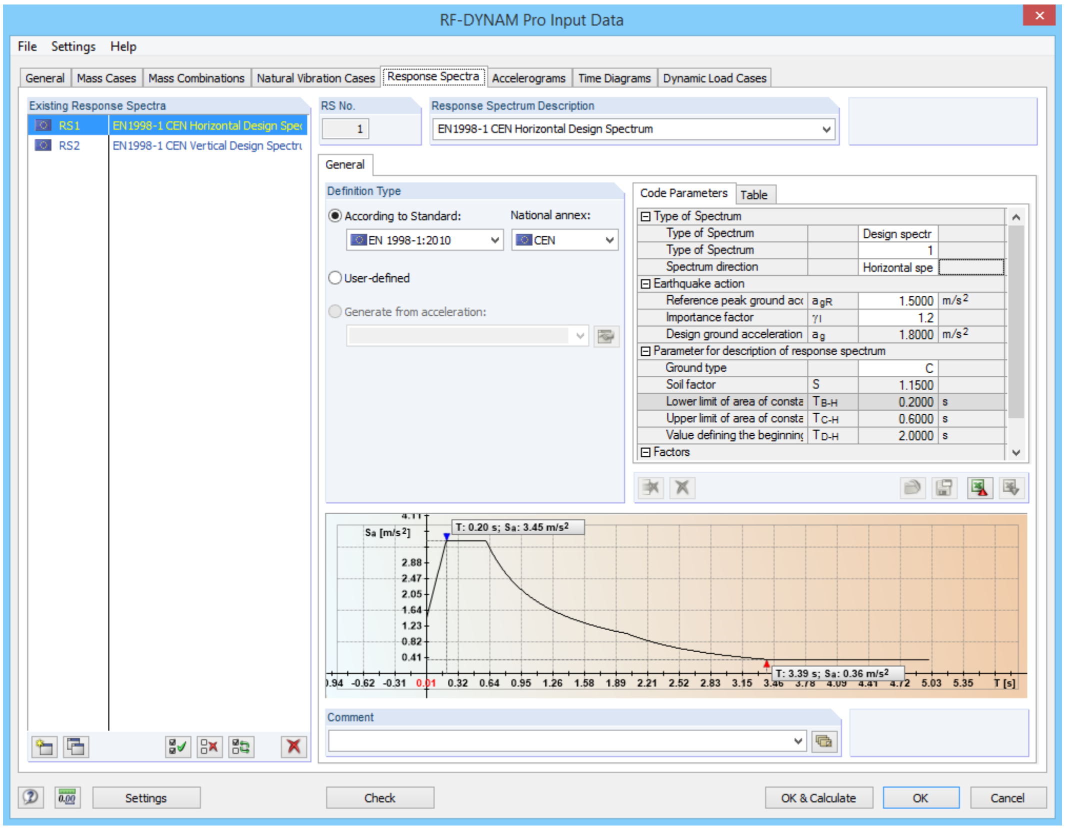
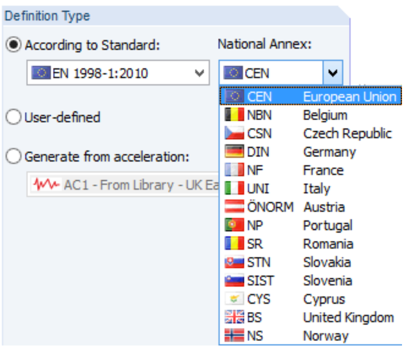


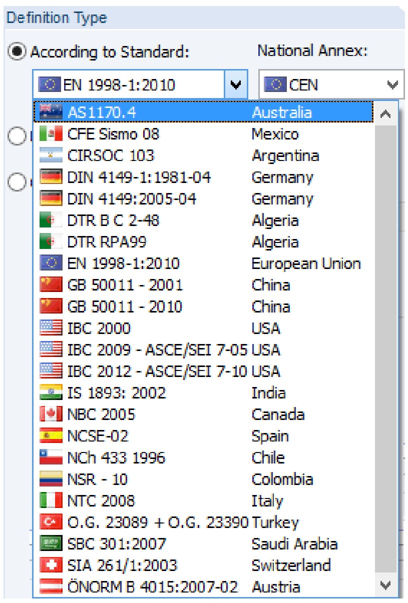
Figure 2.8: Module window *Response Spectra* with the *General* tab open. Response spectra *according to standard* is chosen, the *code parameters* are shown.

Response Spectra according to building standards

The parameters according to a building standard can be entered to generate a response spectrum. The available list of building standards is provided in Figures 2.9a and 2.9b.



a) National annexes of the EN 1998-1 [1].



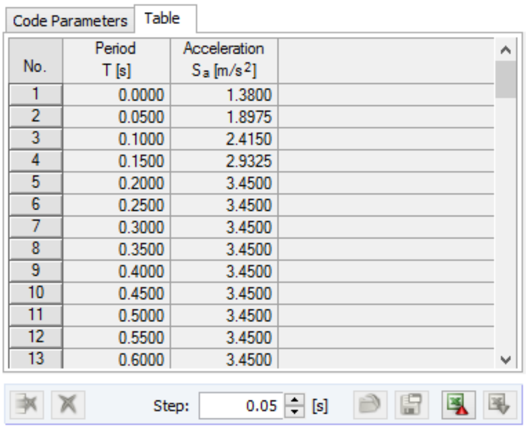
b) International building standards.

Figure 2.9: List of implemented international building standard that regulate the earthquake design of structures and provide formulas for response spectra.

Choose a standard from the drop-down list; the *code parameters* and their default values (see Figure 2.8) change depending on the chosen standard. The parameters are selected from drop-down lists or can be entered manually depending on the type of parameter and depending on the chosen building standard. The fields 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 *Others* you can adjust the parameters T_B, T_C and T_D manually.

The resulting response spectrum is illustrated in the graphic seen in Figure 2.8. You can use the mouse cursor to get information about the displayed values. The values of the generated response spectra are listed in the *table* tab as shown in Figure 2.10.



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 2.10: Tabulated values of the generated response spectrum.

Those tabulated values can be exported to *Excel*. The time step shall be adjusted before exporting the data.

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 2.11.

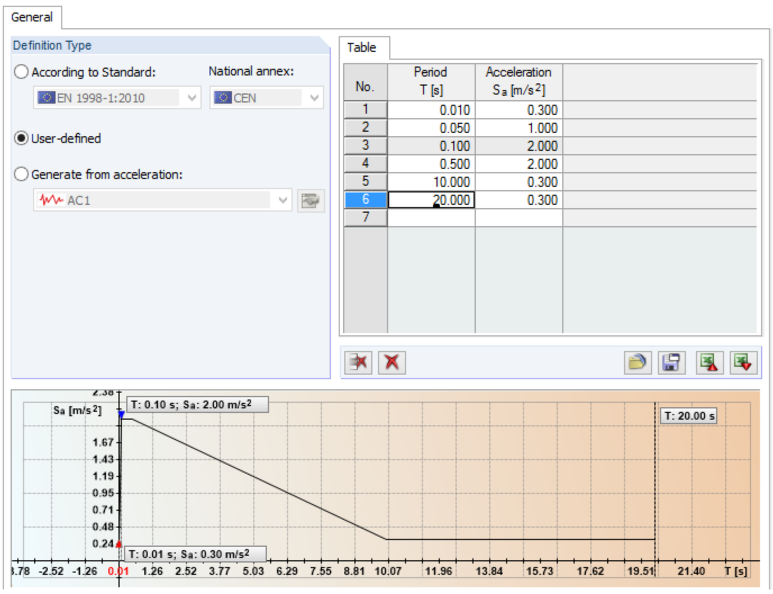


Figure 2.11: User-defined entry of a response spectrum. Period T and acceleration S_a $[m/s^2]$ must be provided.

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

You can export your user-defined response spectra to *Excel* or can import a response spectra from *Excel*.

Response spectra generated from accelerograms

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

This option is only available when *accelerations* are 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 a response spectra from a given acceleration are shown in Figure 2.12.

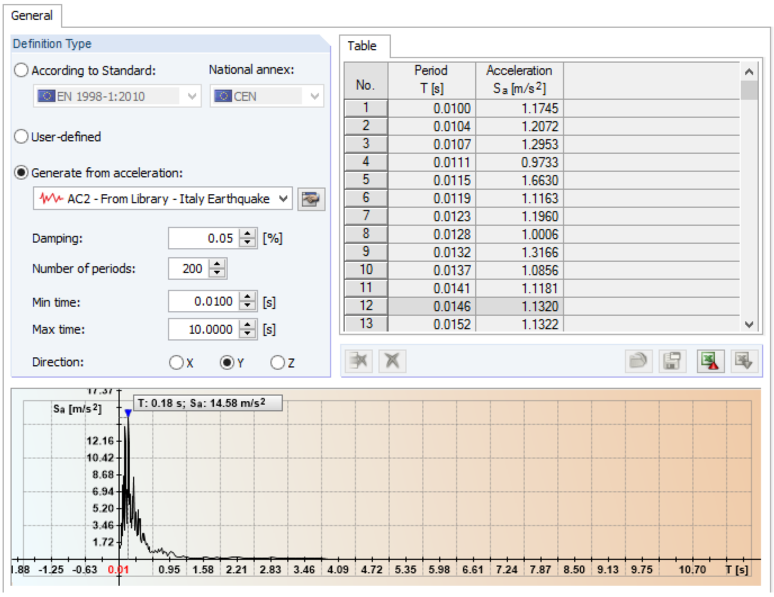
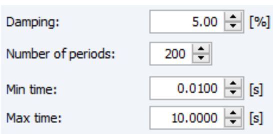


Figure 2.12: Generated response spectrum from a given acceleration. The response spectrum is shown in the graphic and the values are listed in the table.

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



You can adjust the parameters shown 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 2.12). Both gets 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 generated accelerations are smaller the larger the viscous damping is set.

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 minimum 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 2.12.

The maximum 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 2.12.

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 *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 2.13.

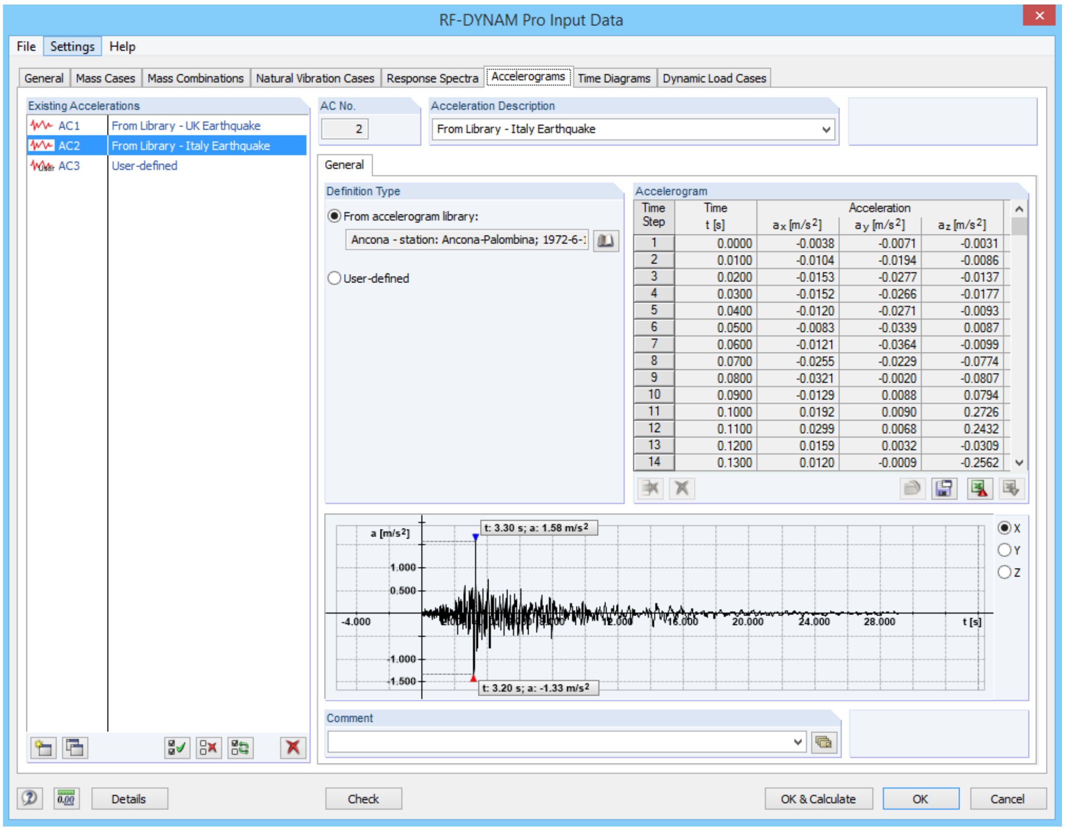


Figure 2.13: Module window *Accelerograms* with the *General* tab open.

The accelerogram tab is only available when *accelerations* are selected in the *General* tab shown in Figure 2.1. This tab belongs to the add-on module *RF-DYNAM Pro - Forced Vibrations*.

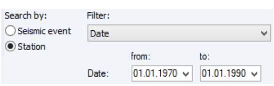
In *RF-DYNAM Pro* you can load accelerograms from a library or enter them manually.

Library containing earthquake recordings

In *RF-DYNAM Pro* a library is available offering a very large number of existing and measured accelerograms. By now, there are more than 1018 accelerograms 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 2.14.

You can select the accelerograms in a tree structure on the left.



It is possible to sort the entries by *measuring station* or *seismic event*. 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 parameters. It is characterized by, for example, the country, latitude and longitude and by the distance to the epicenter. These data are provided on the right-hand side of the library shown in Figure 2.14.

The chosen accelerogram is displayed in the graphic at the bottom of the dialog box shown in Figure 2.13. Most of the accelerograms differ in the three directions X, Y, and Z. You can switch between those directions in the graphic.

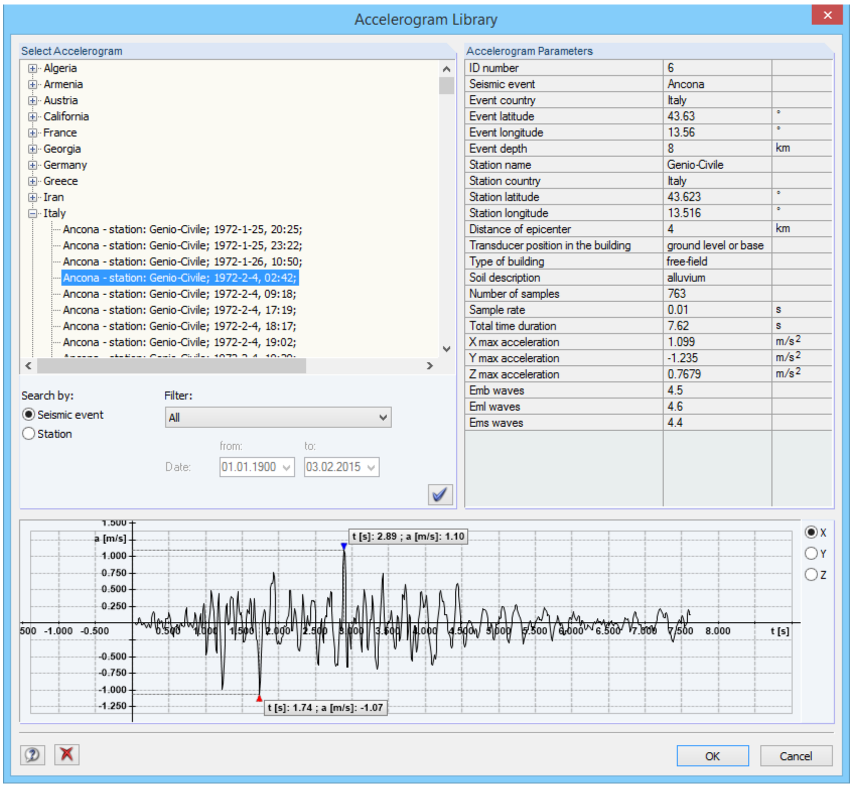


Figure 2.14: Library offering a large number of earthquake recordings.

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

User-defined acceleration-time diagrams

Any kind of acceleration-time diagram can be defined by entering the required values in the table shown in Figure 2.15.

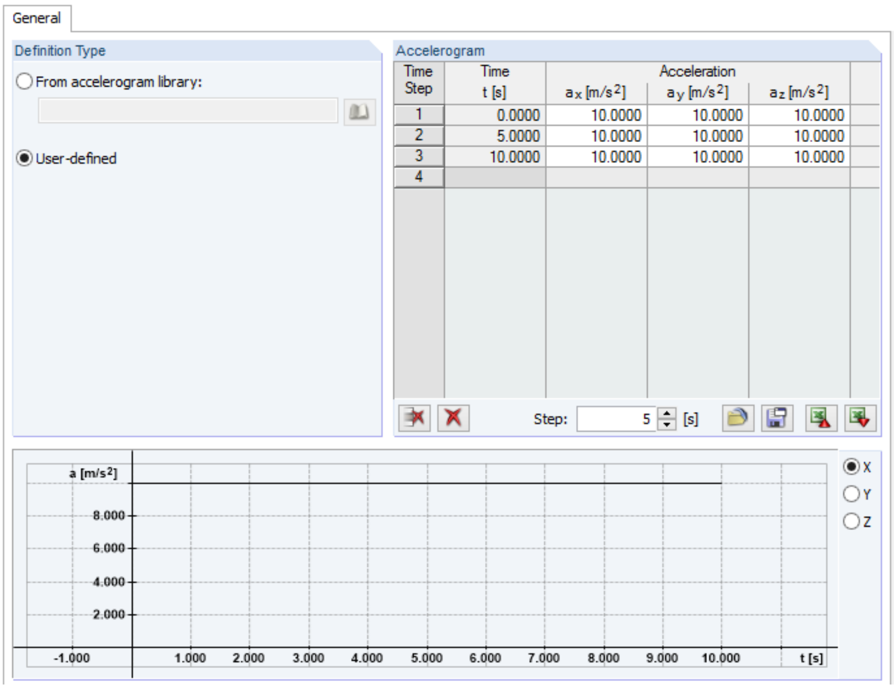


Figure 2.15: User-defined entry of an acceleration-time diagram.

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

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

The defined accelerogram is displayed in the graphic (see Figure 2.13). You can switch between the X, Y, and Z-direction in the graphic.

2.7 Time Diagrams

Time diagrams can be defined either transient, periodic or can be entered as function. They excite the system at a specific position. The load position is defined in static load cases (*LCs*), where any type of load can be entered. The static load cases (*LCs*) are connected to the time diagrams (*TDs*) in the *Dynamic Load Cases* (see Section 2.8), and the multiplier k is used to determine the final quantity of the excitation force.

The time diagram tab is only available when *time diagrams* are selected in the *General* tab shown in Figure 2.1. This tab belongs to the add-on module *RF-DYNAM Pro - Forced Vibrations*.

Transient Time Diagrams

The *Time Diagram* tab is illustrated in Figure 2.16 with an example of a transient time diagram. The values are entered in the table and the resulting time diagram is illustrated in the graphic at the bottom.

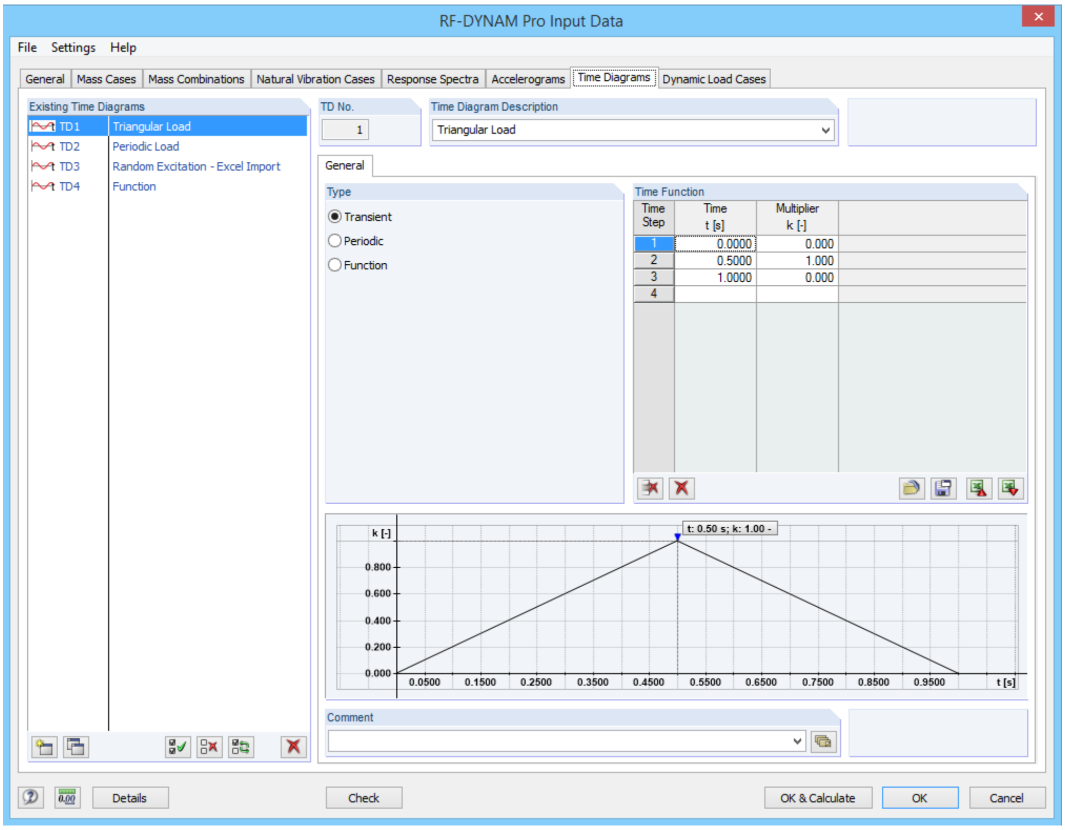


Figure 2.16: Module window *Time Diagrams* showing the input of transient time diagrams.

The table entries can be stored in a library and can be opened whenever needed, an import and export from and to *Excel* is possible.

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 2.17.

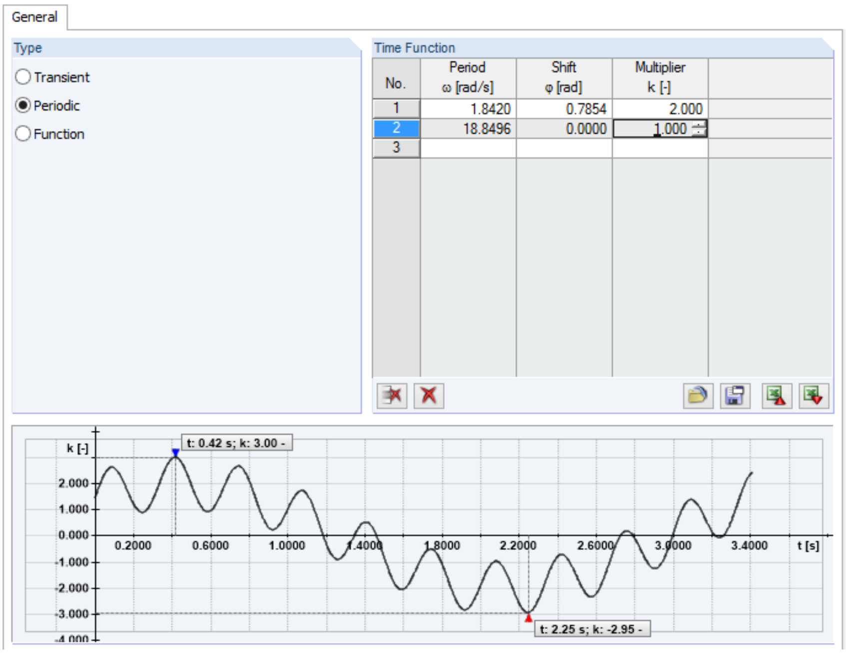


Figure 2.17: Module window *Time Diagrams* showing the input of periodic time diagrams.

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

$$f(t) = k_1 \cdot sin\,(\omega_1 \cdot t + \varphi_1) + k_2 \cdot sin\,(\omega_2 \cdot t + \varphi_2) + ... \tag{2.6}$$

The usual open, save, import and export buttons are available.

Functions

You can enter a function $k(t)$ directly in an edit field to define your time diagram. The parameter t is reserved for the time. The resulting time diagram is displayed in the graphic as shown in Figure 2.18. The generated tabulated values are listed on the right hand side.

The maximum time t_{max} can be adjusted, which influences the graphic and also the tabulated values.

All operators and functions that are available in RFEM can be used also in *RF-DYNAM Pro*. You can use the parameters that you defined in the main program RFEM. For more information about the parametric input please consult the **RFEM manual** in **Section 11.6**.

The tabulated values can be saved to the library or exported to *Excel*. The time steps shall be adjusted before exporting the data.

2.8 Dynamic Load Cases

The *Dynamic Load Cases* combine the input that has been made so far and set calculation parameters for the analysis to be performed. Four different types of *Dynamic Load Cases* are available: Response spectrum analysis, time history analysis of accelerograms, time history analysis of time diagrams, and the equivalent static force analysis. The first three types belong to the add-on module *RF-DYNAM Pro - Forced Vibrations* and the last option belongs to the add-on module *RF-DYNAM Pro - Equivalent Loads*. The *General* tab of the *Dynamic Load Case* tab is shown in Figure 2.19.

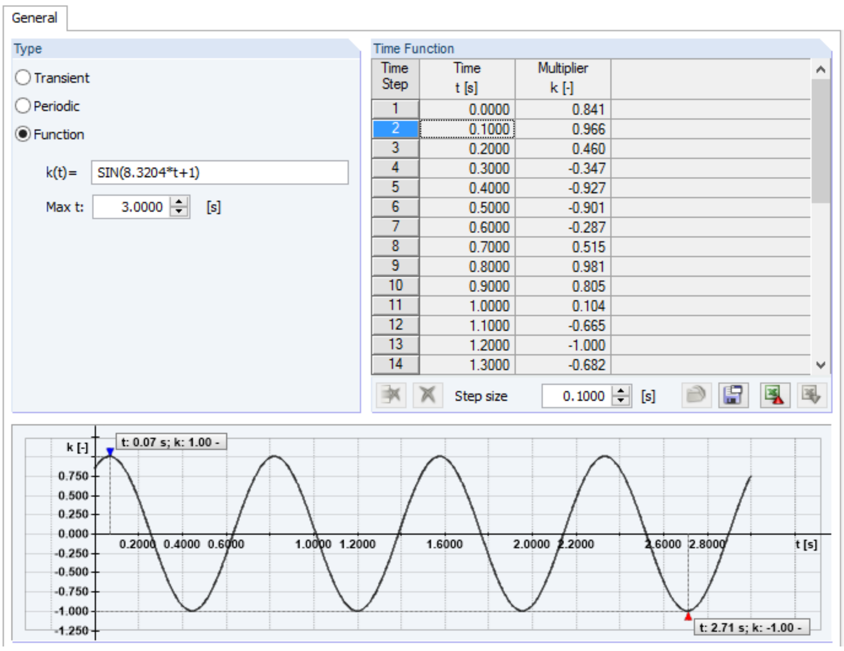


Figure 2.18: Module window *Time Diagrams* showing the input of functions as time diagrams. The parameter t is reserved for the time.

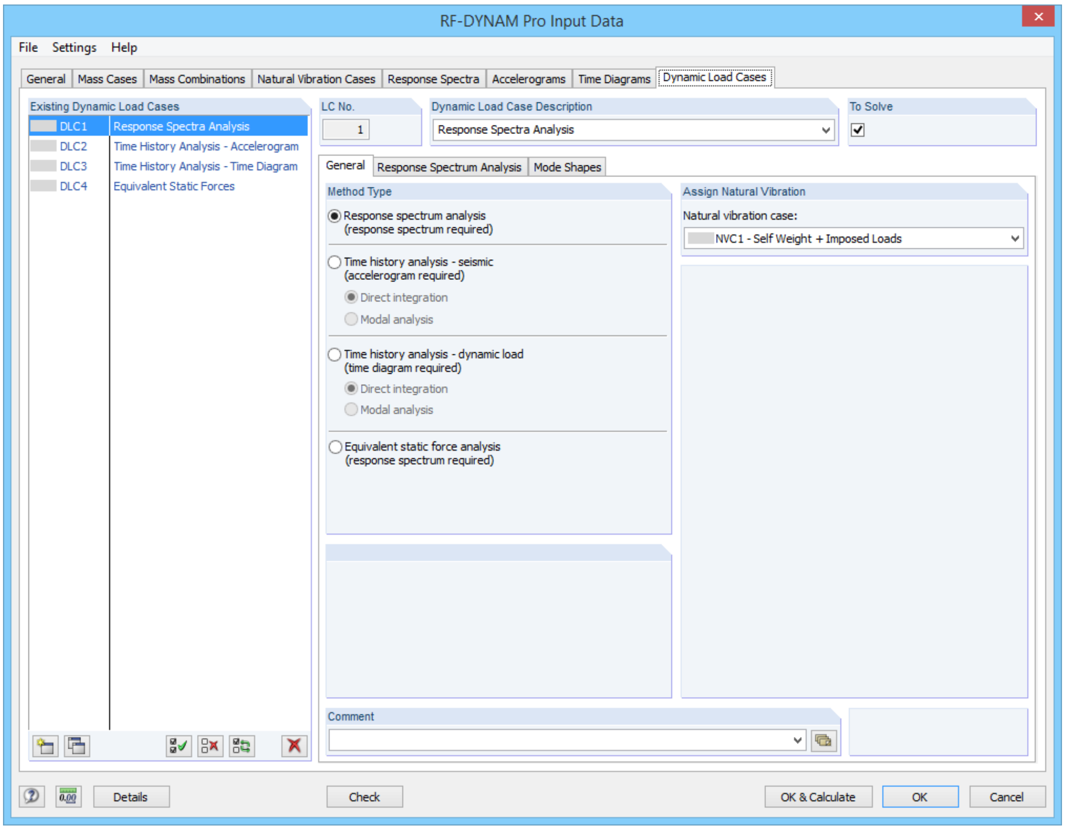
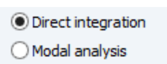
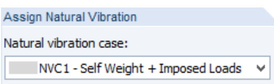


Figure 2.19: Module window *Dynamic Load Cases* with the *General* tab open.

The options are only available when the required input data (as stated in brackets) is provided. When some input data is missing, the options in the *General* tab (Figure 2.19) are grayed out.

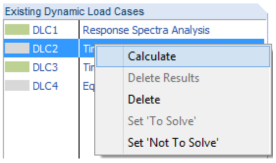


You can choose between the *Direct Integration* and *Modal Analysis* when performing a time history analysis. Further details about those time history solvers are provided in Sections 2.8.2 and 2.8.3.



One specific *Natural Vibration Case* (discussed in Section 2.4) needs to be assigned to the dynamic load case (*DLC*).

The *Dynamic Load Case* number is set automatically and cannot be edited. When a case is deleted later the numbers do not change. You can decide whether or not the specific *Dynamic Load Case* is solved by selecting or clearing the check box *To Solve*. You can also access this switch in the context menu (right-click on the existing *DLCs*).



You can calculate each *Dynamic Load Case* separately by using the context menu and apply *Calculate*. The color of a *DLC* is grey when no results of this *DLC* are available and turns green as soon as the calculation finished and results are available, for more information see Section 3.3.

Load cases (*LCs*) and / or result combinations (*RCs*) are generated in each of the *Dynamic Load Cases* when the check boxes in the *To Generate* frame are selected (in the tab *Response Spectra Analysis* in Figure 2.20, the tab *Time History Analysis* in Figure 2.23 and 2.26, and tab *Equivalent Force Analysis* in Figure 2.27). Those *LCs* and *RCs* are generated automatically and are overwritten when the *RF-DYNAM Pro* calculation is performed again. The names of the *LCs* and *RCs* refer clearly to the originating *DLCs*. Optionally, you can enter the number of the first generated LC / RC. *RF-DYNAM Pro* does not overwrite existing *LCs* or *RCs* (static or generated from other add-on modules), it chooses the first unused LC- / RC-number available. Load cases and results combinations from *RF-DYNAM Pro* are deleted when the results of the corresponding *Dynamic Load Case* are deleted.

2.8.1 Response Spectra Analysis

A response spectra analysis is performed when the corresponding radio button in Figure 2.19 is selected; this is only available in *RF-DYNAM Pro - Forced Vibrations*.

The corresponding sub-tabs when performing a response spectra analysis are called *General*, *Response Spectra Analysis, Damping* and *Mode Shapes*. The *Response Spectra Analysis* tab is illustrated in Figure 2.20.

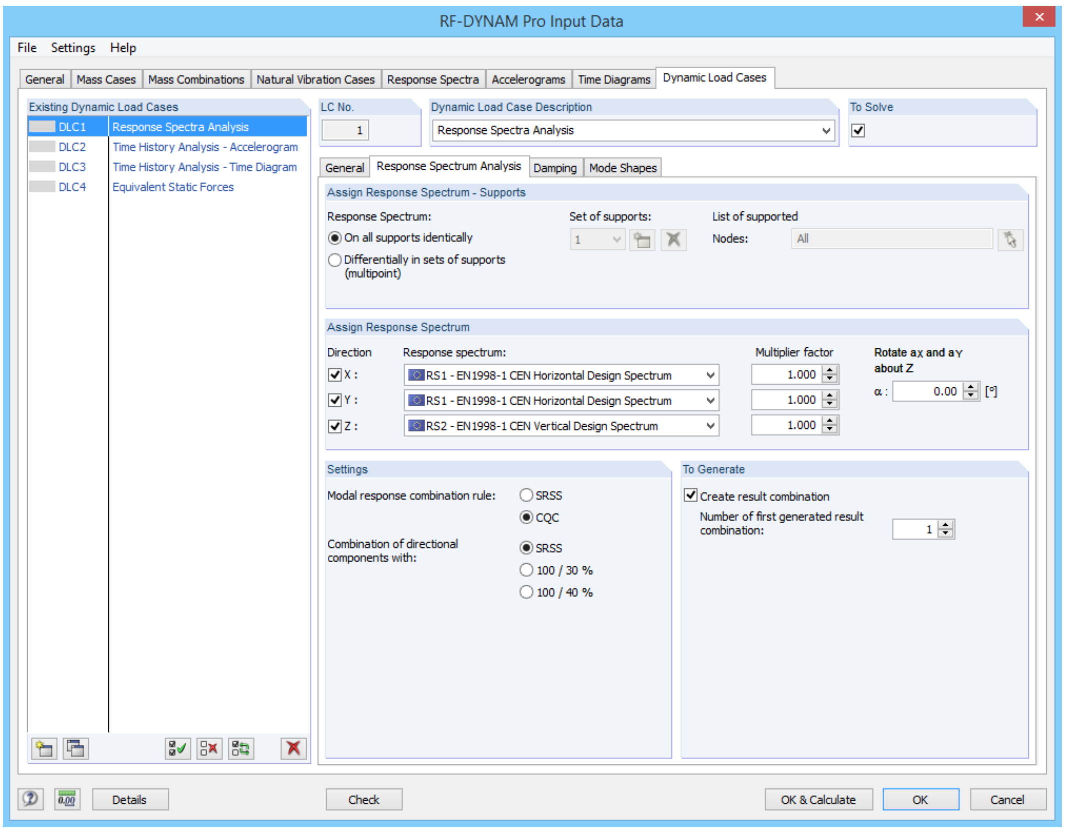
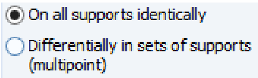


Figure 2.20: Module window *Dynamic Load Cases* with the *Response Spectra Analysis* tab open.

Assign Supports

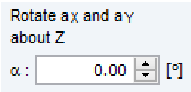


The supports that are excited by the response spectrum have to be assigned. When you consider all supports to be identically excited then those supports can be nodal, line or surface supports. The multi-point option is available for nodal supports only at the moment.

You can create several sets of supports by clicking the [New] button. You can switch in between the available sets by using the drop-down menu. Use the [Delete] button to delete a specific set. For each set, a list of supported nodes must be provided. This can be entered manually or can be selected in the graphic by using the button.

Assign Response Spectrum

In each direction a different response spectrum can be assigned, at least one of the directions must be selected. The available response spectra are chosen from the drop-down menu. The multiplier factor can be adjusted independently for each direction.



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

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

Combination Rules

In the settings you define how the responses resulting from different modes of the structure and resulting from different excitation directions are combined.

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. The results R are combined with the *SRSS* rule as follows,

$$R_{SRSS} = \sqrt{R_1^2 + R_2^2 + ... + R_p^2} \tag{2.7}$$

where R_p are the contributions resulting from p modes of the structure.

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

$$\frac{T_i}{T_j} < 0.9 \tag{2.8}$$

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

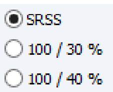
$$R_{CQC} = \sqrt{\sum_{i=1}^{p} \sum_{j=1}^{p} R_i \varepsilon_{ij} R_j} \tag{2.9}$$

with the correlation coefficient ε

$$\varepsilon_{ij} = \frac{8 \cdot \sqrt{D_i D_j}\left(D_i + rD_j\right) r^{3/2}}{(1-r^2)^2 + 4D_i D_j r\left(1+r^2\right) + 4\left(D_i^2 + D_j^2\right) r^2} \quad \text{with } r = \frac{\omega_j}{\omega_i} \tag{2.10}$$

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

$$\varepsilon_{ij} = \frac{8 \cdot D^2(1+r)r^{3/2}}{(1-r^2)^2 + 4D^2 r(1+r^2)} \tag{2.11}$$



The internal forces resulting from different excitation directions can be combined quadratically with the *SRSS* rule, or using the 100% / 30% (40%) rule as known from **EN 1998-1 Section 4.3.3.5** [1]. The *SRSS* rule is applied as defined in Equation 2.7 but now $i = 1..p$ are the excitation directions X, Y and Z.

Export Result Combinations

When you select *create result combinations*, *RF-DYNAM Pro* automatically generates and overwrites result combinations during the calculation. The *RCs* are tight to the result 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 *RCs* are generated when you choose the 100% / 30% (40%) rule.

Damping for the *CQC* rule

The *Damping* tab is only available when the *CQC* rule is chosen for the combination of mode shapes (defined in Equation 2.10).

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. Also the Rayleigh damping with the coefficients α and β is available and is internally converted to the Lehr's damping. The *Damping* tab is shown in Figure 2.21.

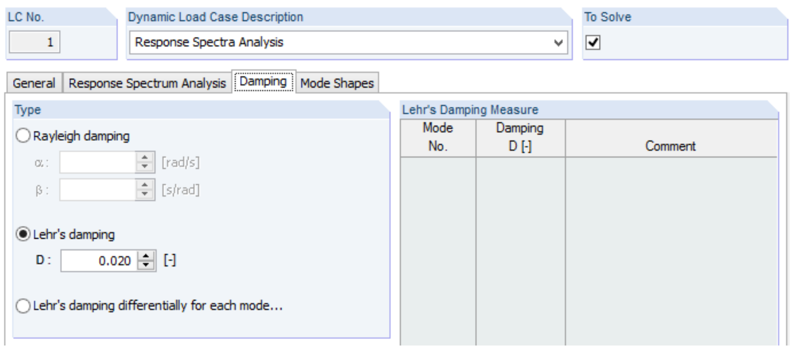


Figure 2.21: Module window *Dynamic Load Cases* with the *Damping* tab open.

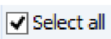
Structural viscose damping is more important for the time history analysis and is therefore detailed in Section 2.8.2.

Mode Shape Selection

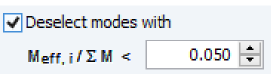
In the *Mode Shapes* tab illustrated in Figure 2.22, the natural frequencies ω and f and periods T are listed together with the corresponding accelerations S_a of the response spectrum and the effective modal mass factors f_{me} in the translational directions. The assigned response spectrum is illustrated in the graphic. Corresponding values are displayed in red when you select a row in the 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 are selected when the check box *Select all* is activated. In the table all modes are selected and this cannot be changed manually.



Using the check box *deselect modes with*, you can deactivate modes with an effective modal mass factor below a certain value. This selection is remembered when you clear the check box.

When both check boxes are cleared you can choose individual modes manually.

The sum of the effective modal mass factors f_{me} is 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 are provided in Section 4.1.

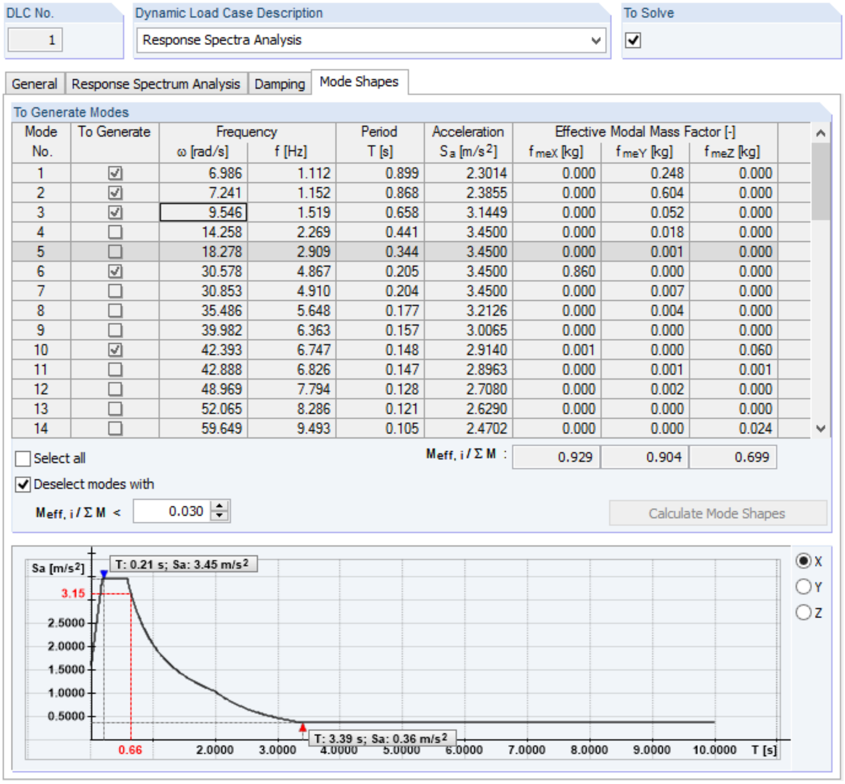


Figure 2.22: Module window *Dynamic Load Cases* with the *Modes Shapes* tab open. The third mode is selected in the table and marked red in the graphic.

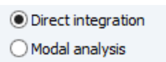
You can switch between the excitation directions X, Y and Z in the graphic. Note that also the values S_a in the table update depending on this choice.

2.8.2 Time History Analysis of Accelerograms

A time history analysis of accelerograms is performed when the corresponding radio button in Figure 2.19 is selected; this is only available in *RF-DYNAM Pro - Forced Vibrations*.

The corresponding sub-tabs when performing a time history analysis are called *General*, *Time History Analysis* and *Damping*. The *Time History Analysis* tab is illustrated in Figure 2.23.

Modal Analysis versus Direct Integration



In the *General* tab (see Figure 2.19) you can choose between the modal analysis and the direct integration.

The modal analysis uses a decoupled system based on the eigenvalues and mode shapes of the structure, determined in the assigned *Natural Vibration Case*. 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 found with an implicit (*Newmark*) solver. Once the eigenvalues are determined, the modal analysis provided in *RF-DYNAM Pro - Forced Vibrations* is slightly faster than the direct integration.

The direct integration is an implicit (*Newmark*) solver that requires sufficient small time steps to achieve accurate results. By now, this solver can only handle linear systems in *RF-DYNAM Pro - Forced Vibrations*. There is no modal analysis required, so the number of required eigenvalues can be set to 1 in the assigned *Natural Vibration Case* (see Section 2.4).

The most important when performing a time history analysis is the choice of the time step. It is a compromise between calculation time and accuracy, the smaller the time step the longer takes the

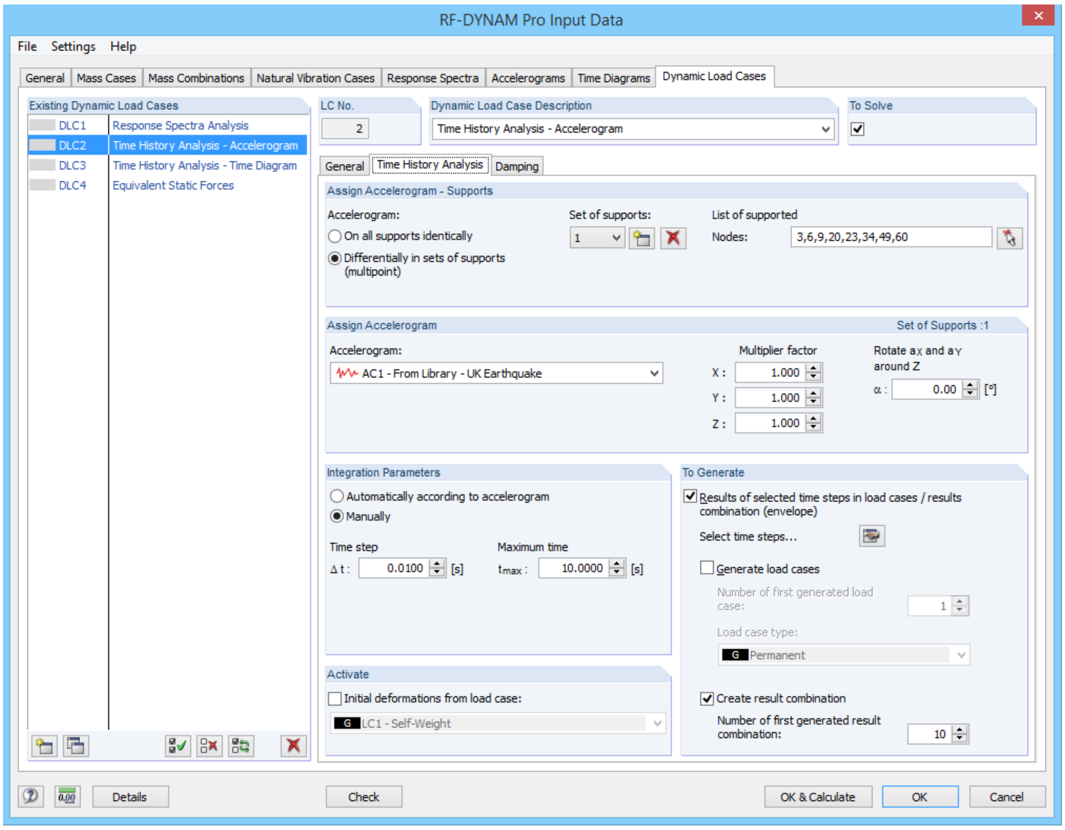


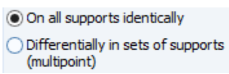
Figure 2.23: Module window *Dynamic Load Cases* with the *Time History Analysis* (*Accelerograms*) tab open.

calculation, but when the time step is chosen too large the results are not accurate. The following advice can be given [10]:

- Considering the time diagram (see Section 2.7), the shortest length of the discrete time diagram shall be split into at least 7 time steps
- The largest 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 $\Delta t \leq 1\,/\,20 f$

The recommended time steps achieve accurate deformations and forces, but when you are interested in velocities and accelerations you have to choose even smaller time steps.

Assign Supports

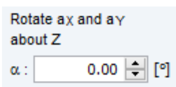


The supports that are excited by the accelerogram have to be assigned. When you consider all supports to be identically excited then those supports can be nodal, line or surface supports. The multi-point option is available for nodal supports only at the moment.

You can create several sets of supports by clicking the [New] button. You can switch in between the available sets by using the drop-down menu. Use the [Delete] button to delete a specific set. For each set, a list of supported nodes must be provided. This can be entered manually or can be selected in the graphic by using the [Pick] button.

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 X-Y plane about the Z-axis. So for example, when your accelerogram shall excite the structure $45°$ rotated about the Z-axis, activate only the X-direction with your response spectra chosen from the drop-down menu and enter $\alpha = 45°$.

For each set of supports you can assign different accelerograms.

Integration Parameters

RF-DYNAM Pro can choose the time steps automatically according to the assigned accelerogram, or you can define time steps and maximum time manually. Results for each single time step are produced.

Initial Deformation

When you select *initial deformations from load case* you can import initial conditions directly from a load case; these are the conditions at time step $t = 0\ sec$.

Export of Load Cases and Result Combinations

You can export load cases for single time steps by activating the *generate load cases* check box. Choose the load case type from the drop down list below.

In the list shown in Figure 2.24, you can select the time steps that you want to export.

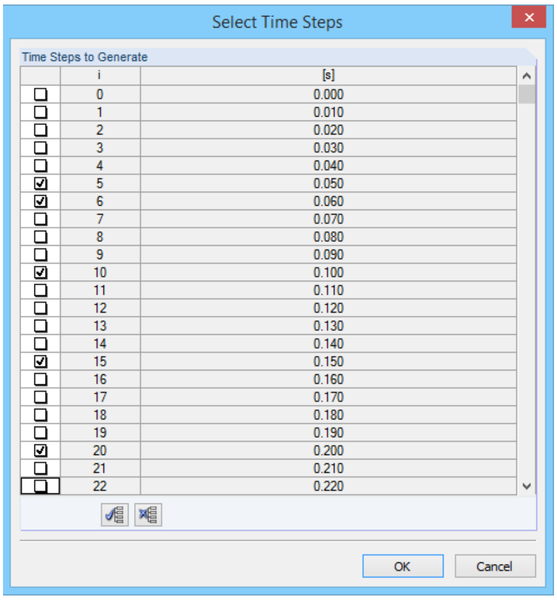


Figure 2.24: Dialog box to choose time steps for the load case export in the time history analysis.

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

Structural Damping

In *RF-DYNAM Pro* structural viscose damping is available, this can be entered using the Rayleigh coefficients α and β, or using 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* tab is shown in Figure 2.25.

With the Rayleigh coefficients, the damping matrix $\mathbf{C}$ is defined as

$$\mathbf{C} = \alpha\mathbf{M} + \beta\mathbf{K} \tag{2.12}$$

using the factors α and β. The Rayleigh damping coefficients are used for direct integration where a damping matrix $\mathbf{C}$ is required, which does not need to be a diagonal matrix. For more information and the pro and cons of the Rayleigh damping see for example [10].

The Lehr's damping values D_i are used for the modal analysis where the structure is decoupled in its single eigenvalues. Here the damping matrix $\mathbf{C}$ is 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:

$$D_i = \frac{c_i}{2 \cdot m_i \cdot \omega_i} \tag{2.13}$$

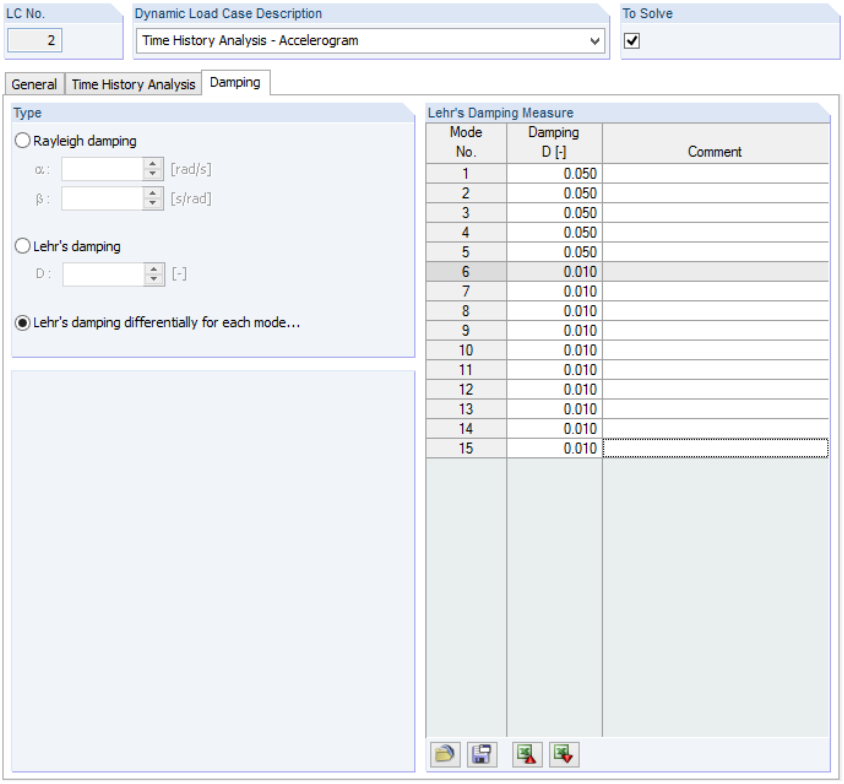


Figure 2.25: Module window *Dynamic Load Cases* with the *Damping* tab open.

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

RF-DYNAM Pro is internally converting the damping values depending on the method of analysis. For the conversion the following system of equation is solved for two independent frequencies of the system:

$$D_r = \frac{1}{2}\left(\frac{\alpha}{\omega_r} + \beta\omega_r\right) \tag{2.14}$$

When direct integration is employed for the time history analysis (set in *General Data*, see Figure 2.19), Lehr's damping values can only be entered for the first two modes of the system. This is due to the internal conversion to Rayleigh coefficients (Equation 2.14). When only one Lehr's damping value is provided or only one eigenvalue is calculated in the natural vibration cases (*NVCs*) the coefficient β is set to zero and Equation 2.14 is used to calculate α.

2.8.3 Time History Analysis of Time Diagrams

A time history analysis of time diagrams is performed when the corresponding radio button in Figure 2.19 is selected; this is only available in *RF-DYNAM Pro - Forced Vibrations*.

The corresponding sub-tabs when performing a time history analysis of time diagrams are called *General*, *Time History Analysis* and *Damping*. The *Time History Analysis* tab is illustrated in Figure 2.26. It looks slightly different to the *Time History Analysis* tab associated to accelerograms. Here, any point of the system can be excited in any direction by the defined time diagrams, not only supports.

Modal Analysis versus Direct Integration

In the *General* tab (see Figure 2.19) you can choose between the modal analysis and the direct integration. The solvers are identical to those used for the time history analysis of accelerograms; the details were provided in Section 2.8.2.

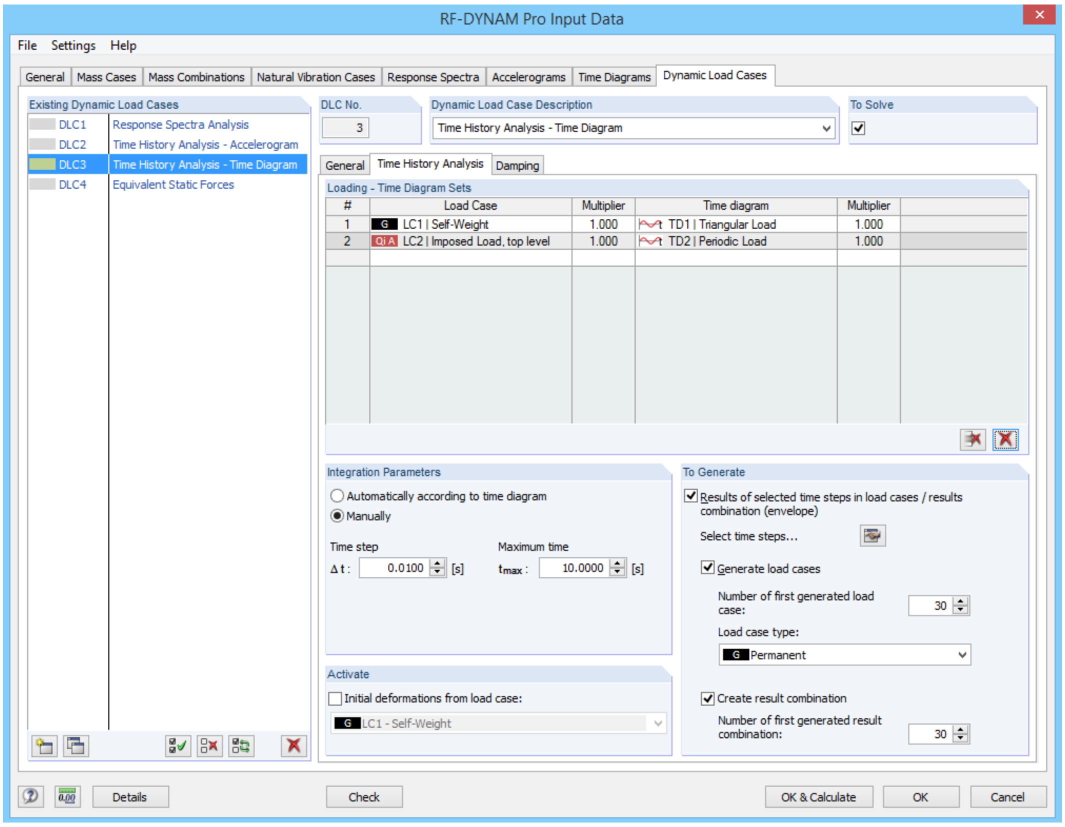


Figure 2.26: Module window *Dynamic Load Cases* with the *Time History Analysis* (*Time Diagrams*) tab open.

Loading - Time Diagram Sets

Load cases defined in the main program *RFEM* are combined with *Time Diagrams* defined earlier (see Section 2.7) to *Loading - Time Diagram Sets*. In the table, choose a load case and 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 such *Loading - Time Diagram Sets* by filling more rows in the table as shown in Figure 2.26. This is required to simulate an excitation that is varying in time but also changing its position, *i.e.* pedestrian walking on a bridge.

Integration Parameters

RF-DYNAM Pro can choose the time steps automatically according to the assigned time diagram. For harmonic time diagrams the maximum time is set to $t_{max} = 10 \cdot 2\pi/\omega$ which is ten times the period length. For transient time diagrams the maximum time is overtaken from the user-defined time diagram. When the automatic settings do not suit, you can define time steps and maximum time manually. Results for each single time step are produced.

Initial Deformations

By activating *Initial deformations from load case* you can import initial conditions directly from a load case; these are the conditions at time step $t = 0\, sec$.

Export of Load Cases and Result Combinations

You can export load cases for single time steps by activating *Generate load cases*. Choose the load case type from the drop down list below.

You can select the time steps that you want to export by choosing from the list shown in Figure 2.24 on page 32.

Result combinations (*RCs*) as a result envelope with maximum and minimum values of all time steps is generated when you select *Create result combination*.

Structural Damping

The settings for structural damping were discussed in detail in Section 2.8.2 and are the same for the time history analysis of time diagrams.

2.8.4 Equivalent Load Analysis

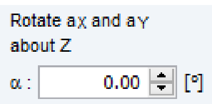
The equivalent load analysis is performed when the corresponding radio button in Figure 2.19 is selected; this belongs to the add-on module *RF-DYNAM Pro - Equivalent Loads*. The equivalent load analysis in *RF-DYNAM Pro* is based on the multi-modal response spectra analysis. The main differences to the *Response Spectra Analysis* in the add-on module *RF-DYNAM Pro - Forced Vibrations* (described in Section 2.8.1) are listed below:

- Load cases with equivalent loads are exported to *RFEM* separate for each mode and each excitation direction
- The calculation of load cases is done in the main program RFEM
- Accidental torsional actions can be considered automatically
- Base shear forces can be easily evaluated separate for each mode
- The modal responses are combined with the *SRSS* rule using a *preserve sign* option[11]
- Result combinations are produced separately for each excitation direction (combined modal responses) and for the combination of results from different excitation directions
- 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 called *General*, *Equivalent Force Analysis* and *Mode Shapes*. The *Equivalent Force Analysis* tab is illustrated in Figure 2.27.

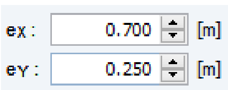
Assign Response Spectrum

In each direction a different response spectrum can be assigned, at least one of the directions must be selected. The available response spectra have to 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 X-Y plane about the Z-axis. So for example, when your response spectra shall excite the structure $45°$ rotated about the Z-axis, activate only the X-direction with your response spectra chosen from the drop-down menu and enter $\alpha = 45°$.

Accidental Torsional Actions



RF-DYNAM Pro considers accidental torsion actions automatically when you select the check box and enter the eccentricities e_X and e_Y. This is regulated for example in **EC8 Section 4.3.2 and 4.3.3.3.3** [1] and is similarly regulated in other international building standards. The eccentricities e_X and e_Y define how far the center of mass shall be considered as being displaced, to account for uncertainties in the location of masses.

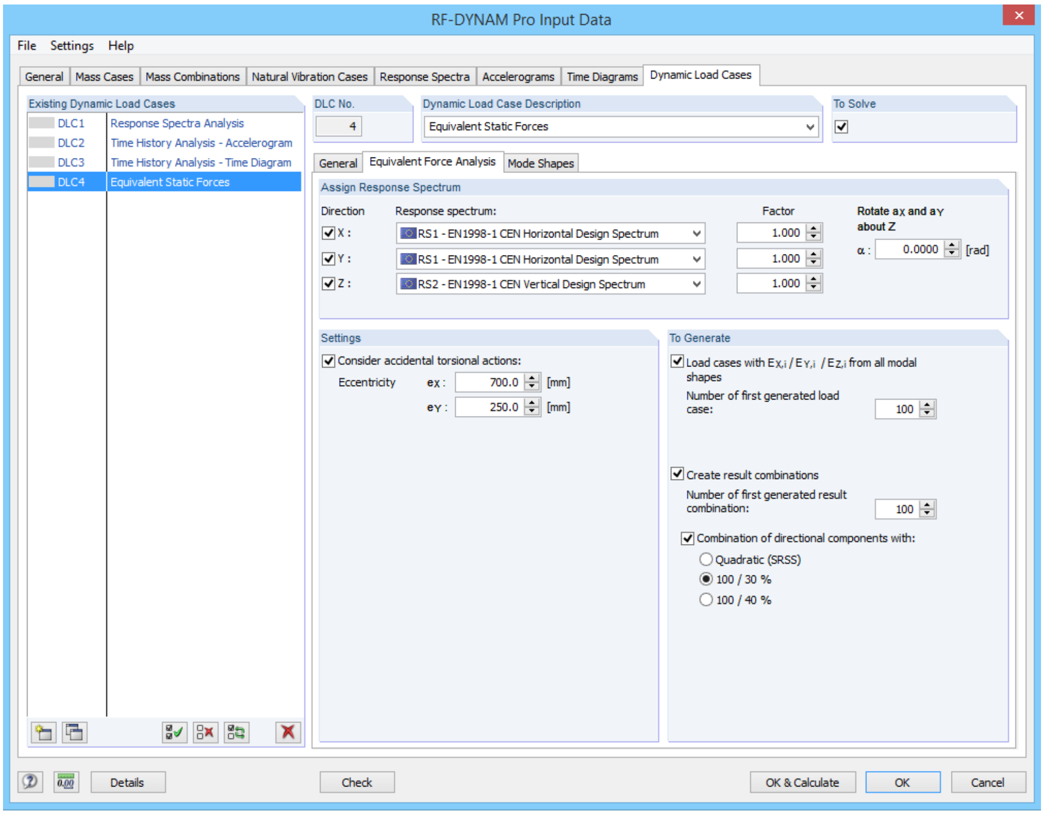


Figure 2.27: Module window *Dynamic Load Cases* with the *Equivalent Force Analysis* tab open.

When you export load cases with generated equivalent loads, two LCs are generated for each mode shape and each direction. One with positive torsional moments, and the other one with negative torsional moments. *RF-DYNAM Pro* creates an alternative combination of those two LCs.

Combination Rules

In the add-on module *RF-DYNAM Pro - Equivalent Loads* the *SRSS* rule is available to combine the different modal responses. In the *To Generate* frame you select the *RCs* to be exported. The modal responses of the structure and results from different excitation directions are combined.

The modal responses are combined using the *Square Root of the Sum of the Squares* (*SRSS*) rule. Maximum results are combined and the signs are lost in the standard form of the *SRSS* rule. The results R are combined with the *SRSS* rule as follows,

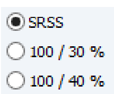
$$R_{SRSS} = \sqrt{R_1^2 + R_2^2 + ... + R_p^2} \tag{2.15}$$

where R_p are the contributions resulting from p modes of the structure.

In *RF-DYNAM Pro - Equivalent Loads* a modified form of the *SRSS* rule is used in order to preserve the signs. This modified *SRSS* rule was first published by Katz [11] and is defined as follows:

$$R_{SRSS} = \sum_{i=1}^{p} f_i \cdot R_i \quad \text{with} \quad f_i = \frac{R_i}{\sqrt{\sum\limits_{j=1}^{p} R_j^2}} \tag{2.16}$$

Using this formula the result combinations are consistent in itself. The corresponding internal forces are getting smaller.



The internal forces resulting from different excitation directions can be combined quadratically with the *SRSS* rule, or using the 100% / 30% (40%) rule as known from **EN 1998-1 Section 4.3.3.5** [1]. The *SRSS* rule is applied as defined in Equation 2.15 but now $i = 1..p$ are the excitation directions X, Y and Z.

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. When you select *create result combinations* several result combinations are automatically generated.

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

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 the settings in there were discussed in detail in Section 2.8.1 and are the same for the equivalent load analysis.

2.9 Global settings in the *Details* dialog box

The *Details* dialog box can be accessed by pressing the button [Details]. You can also reach this by **Settings** → **Details**. You can set global parameters in this dialog box that are valid for the overall dynamic calculation independent of any defined cases within the module. The dialog box is shown in Figure 2.28.

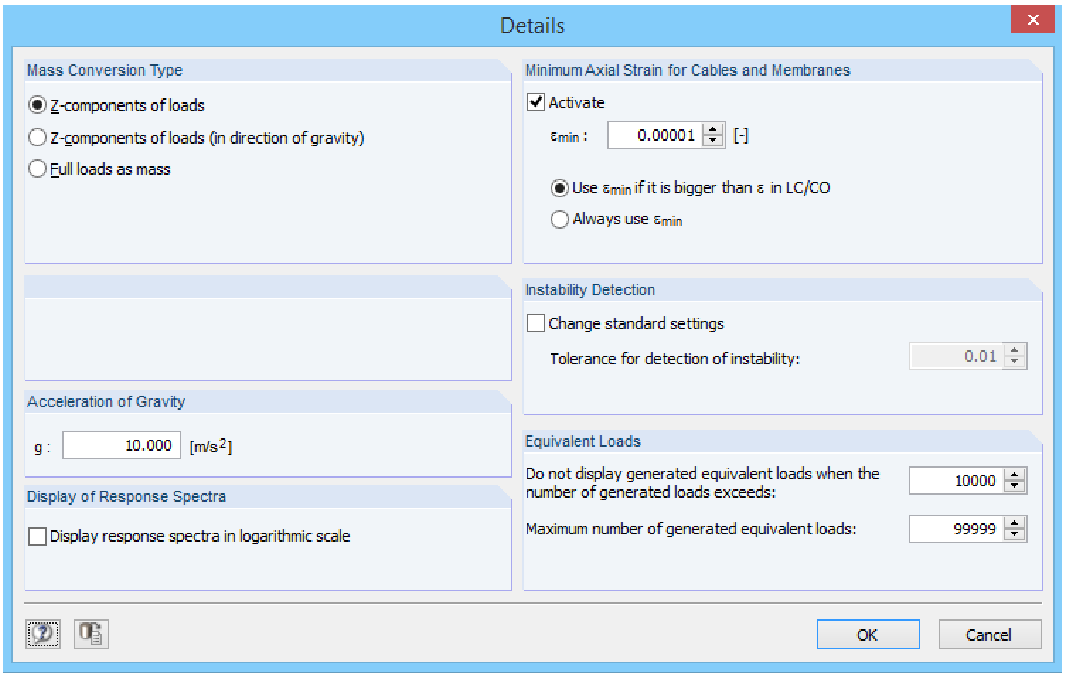
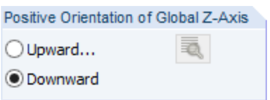


Figure 2.28: The *Details* dialog box to adjust global parameters for the *RF-DYNAM Pro* module.

Mass Conversion Type

In Section 2.2 the mass import from load cases was explained. You can modify which loads are imported by switching between the radio buttons shown in Figure 2.28. You can import only the Z-components of loads in both global Z-directions, or in the direction of gravity only. The third option imports full loads regardless their direction. Masses behave differently than loads. You have to consider them as a mass point not as a load acting in a specific direction. So, when different loads are acting on one FE-node they are added together.



The direction of gravity is defined in the direction of the global Z-axis when the positive orientation is defined downwards, and 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** $\rightarrow$ **Model Data** $\rightarrow$ **General Data**.

Acceleration of Gravity

You can change the acceleration of gravity $g\ [m/s^2]$. The default value set is $g = 10\ m/s^2$ as used also in the main program *RFEM*.

Display Response Spectra

You can change the display of the response spectra shown in the graphic in Figure 2.8 and 2.22. The default display uses a linear x-axis, but you can change this to a logarithmic scale by activating the check box.

Minimal Axial Strain for Cables and Membranes

For this special type of members a minimum axial strain is required. When this limit is set too small, the eigenvalues achieved are not realistic and only local mode shapes are determined. The default value is reasonable in most of the cases. For more details about cables have a look at the ***RFEM* manual Section 4.17**.

Instability Detection

The stability of a system can be analyzed, 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 2.8.4 load cases are exported containing the generated equivalent loads. Loads can only be viewed in the graphic when less than 10000 loads are produced. You can change this value here in the *Details* dialog box but note that the process of displaying more than 10000 is very slow. The maximum number of loads that RFEM can produce and export to *LCs* is 99999. 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 99999 FE-nodes. When your structure has more FE-nodes, the smallest equivalent loads are neglected, and only 99999 loads are produced. You can adjust the maximum number of generated equivalent loads in the *Details* dialogue.

2.10 Units and Decimal Places

You can access the *Units and Decimal Places* dialog box with the button shown on the left; the dialog box is shown in Figure 2.29.

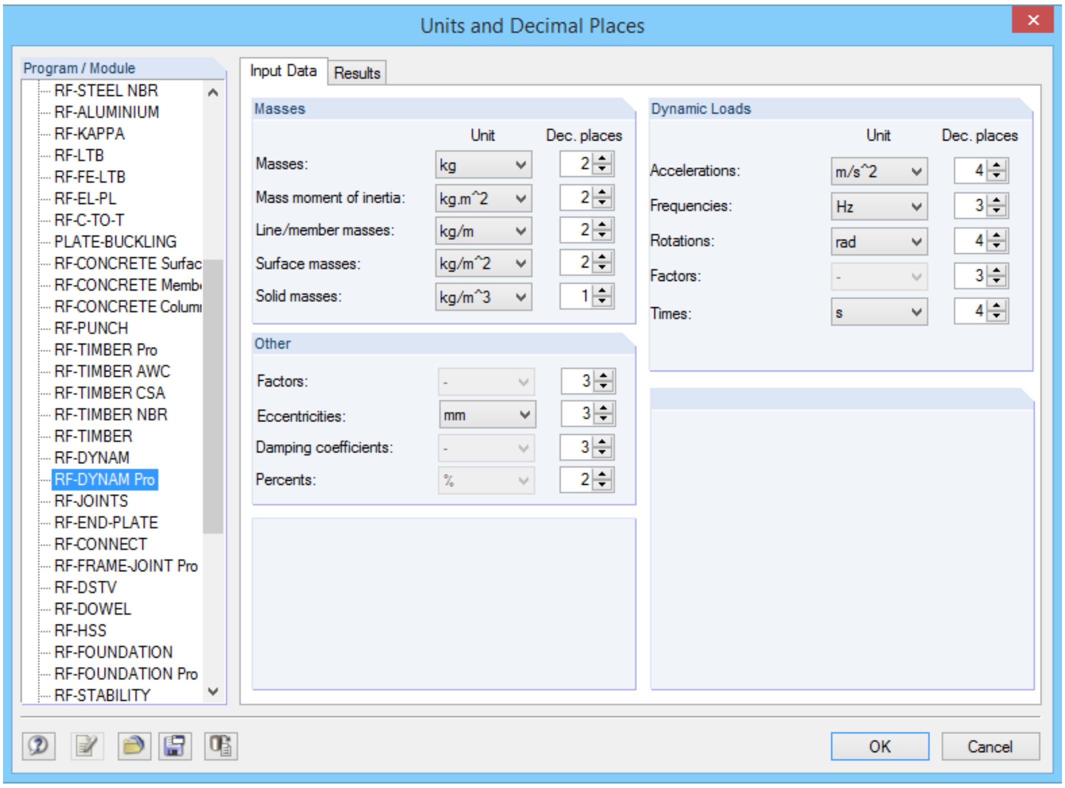


Figure 2.29: The *Units and Decimal Places* dialog box is illustrated where the units and decimal places of the *RF-DYNAM Pro* input data can be adjusted.

Search for the add-on module *RF-DYNAM Pro* in the list of modules. The *Input Data* tab is open and units can be chosen from the drop-down menus and decimal places can be adjusted.

3 Calculation

3.1 Check

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

The module shows the *Input Data Verification* dialog box displaying any warning and error messages. When no errors where found the message *no consistency errors found* is displayed.

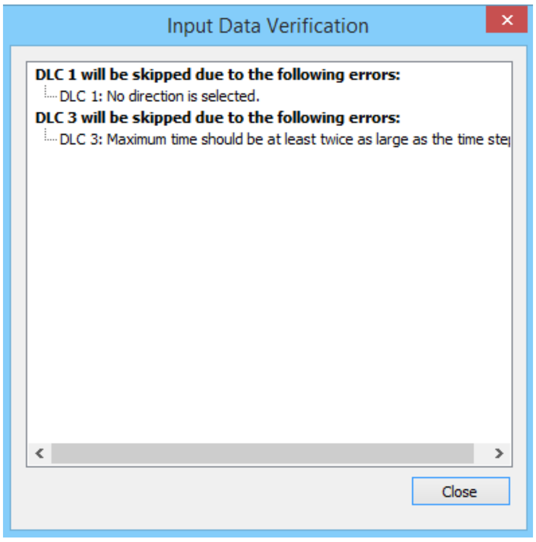
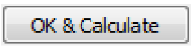


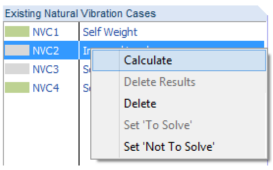
Figure 3.1: *Input Data Verification* dialog box with warning and error messages.

When you click [OK & Calculate] the check is also performed. When errors occur, the *Input Data Verification* dialog 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. Calculated are only the cases (*NVCs* and *DLCs*) where no results are available, and which are selected to be solved (*To Solve* check box). The add-on module closes once the calculation is finished.



To start the calculation of single natural vibration cases (*NVCs*) or dynamic load cases (*DLCs*) you can right-click on the specific case and press *Calculate*. Only the selected case within *RF-DYNAM Pro* is calculated. Once the calculation is finished the module stays open, the color of the selected case changes from grey to green. When you select to calculate a *DLC* then also the assigned *NVC* is calculated.

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. Calculated are only the cases (*NVCs* and *DLCs*) where no results are available, and which are selected to be solved (*To Solve* check box).

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

The *To Calculate* dialog 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.

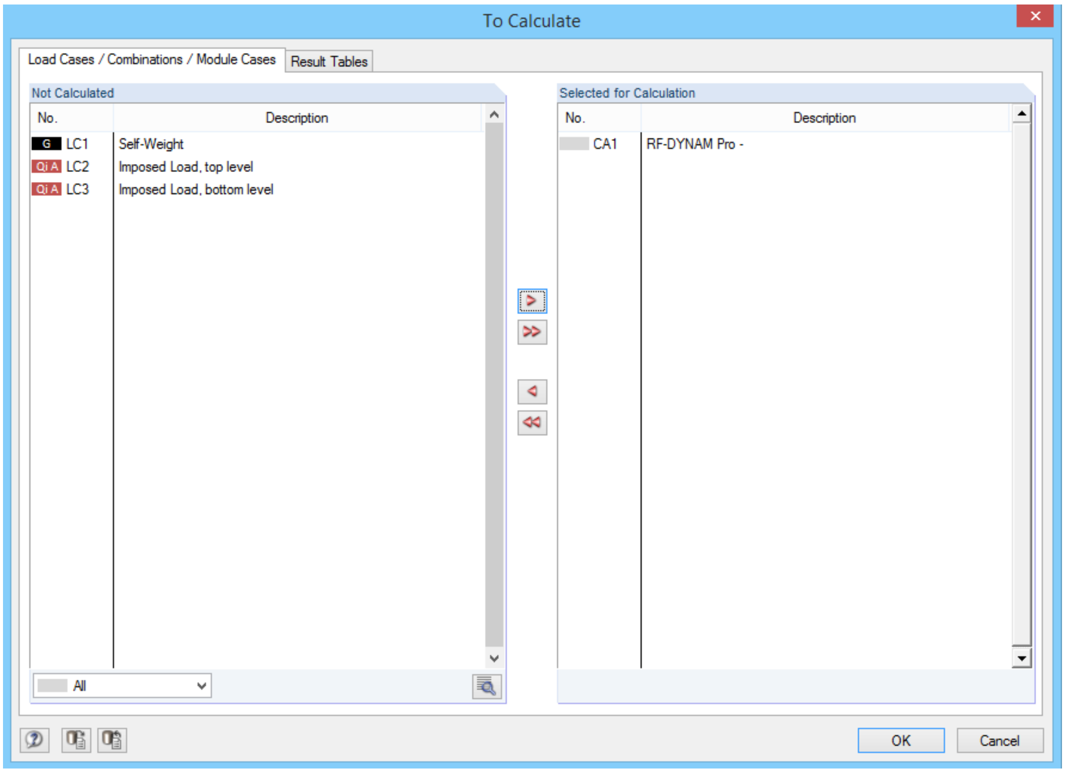
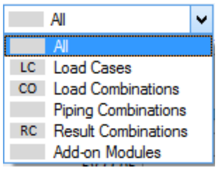


Figure 3.3: Dialog box *To Calculate*



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. Calculated are all cases (*NVCs* and *DLCs*) where no results are yet available, and which are selected to be solved (*To Solve* check box).

Once you started the calculation you can observe the analysis process in a separate dialog box shown in Figure 3.4.

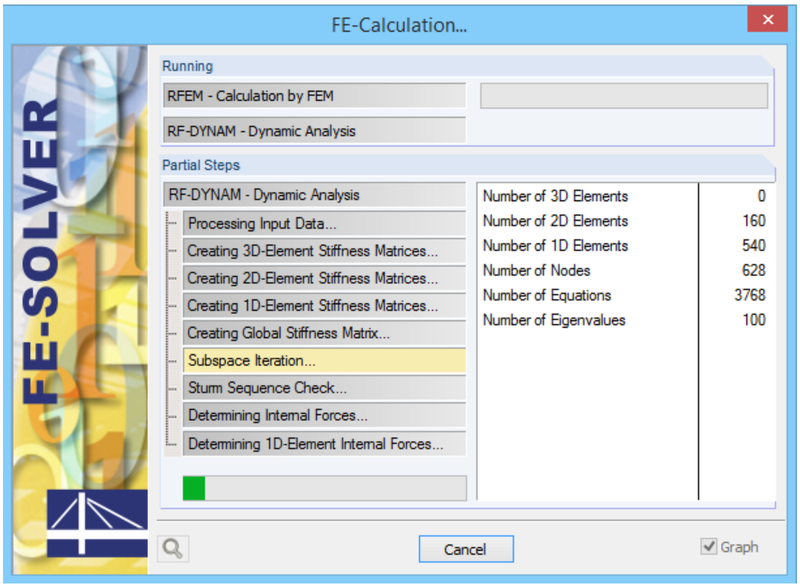


Figure 3.4: Process of the *RF-DYNAM Pro* calculation.

3.3 Availability of the results

The deletion and calculation process 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 deleted separately by using the context menu (right click on the specific case). The context menu also offers to delete the case, set the case *To Solve* or *Not to solve*.

The color 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 *LCs* and *RCs* do exist. Exported *RCs* and *LCs* are tight to the results, and are deleted as soon as the results of a case are deleted.

Results of a specific case are deleted, when

- you use the context menu to delete the results
- something is changed in the *NVCs* or *DLCs* within *RF-DYNAM Pro*
- assigned mass cases (*MCs*), mass combinations (*MCOs*), natural vibration cases (*NVCs*), response spectra (*RSs*), accelerograms (*ACs*) or time diagrams (*TDs*) change
- assigned load cases *LCs*, load combinations *COs* that are defined in *RFEM* change
- exported *LCs* or *RCs* are deleted that belong to the *DLC*

Results of a specific case are not deleted, when

- only the description of the case is modified
- in *NVCs* the scaling of mode shapes is altered

4 Results

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

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

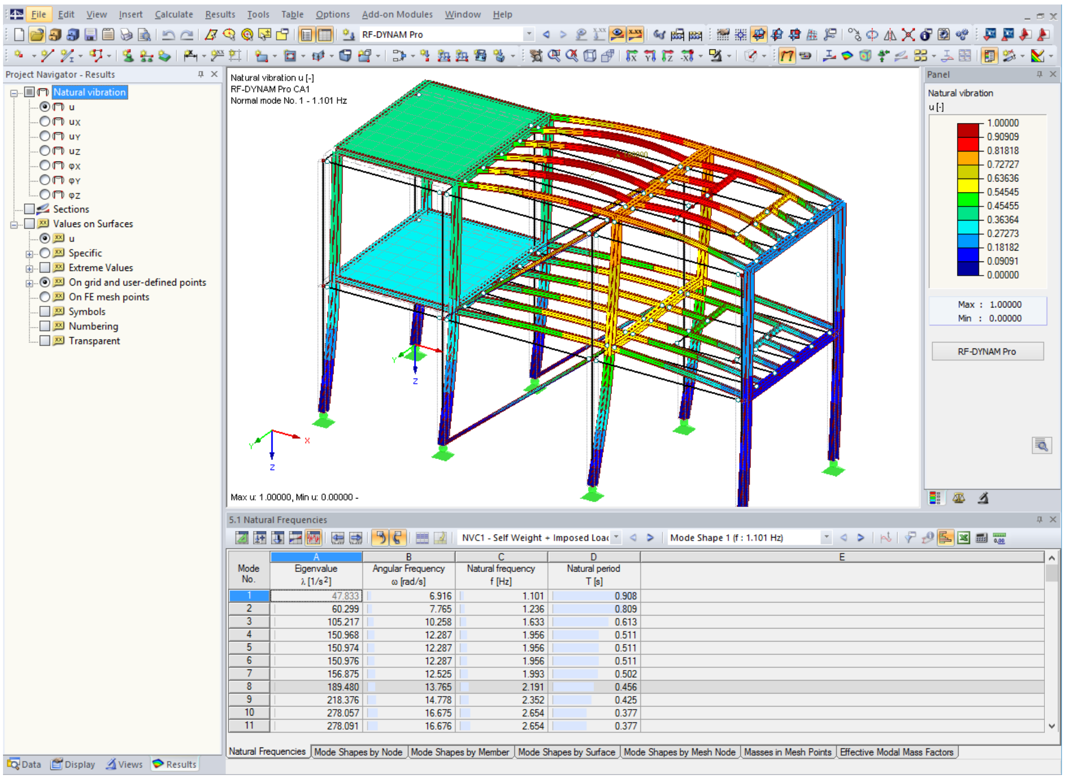
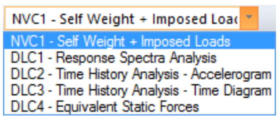


Figure 4.1: Results of an *Natural Vibration Case* in the main program RFEM.

The *Display Navigator* and *Results Navigator* are discussed in **Section 3.4.3**, the *Tables* in **Section 3.4.4** and the *Control Panel* in **Section 3.4.6** in the **RFEM manual**. Here in the *RF-DYNAM Pro* manual only the newly available results are discussed.

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

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



With the drop-down menu you can switch between the available *Natural Vibration Cases* and the *Dynamic Load Cases*. You can also switch between the cases using the and buttons. The available result tables depend on this choice. This is clarified in the following sections.

The graphic in the work area of RFEM is also updated according to this choice. 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 is working as usually in RFEM.

4.1 Natural Vibration Cases

The result tables that belong to *Natural Vibration Cases* are available when an *NVC* case is chosen in the drop-down menu. The result tables 5.1 to 5.7 belong to the *Natural Vibration Cases*. These results are always available; they belong to the module *RF-DYNAM Pro - Natural Vibrations*. The input data required for *Natural Vibration Cases* were discussed in Section 2.4.

Mode Shape 1 (f : 1.101 Hz)

You can switch between the several available mode shapes in the graphic by using the drop-down menu shown on the left, or by using ◁ and ▷ buttons. The maximum deformation displayed in the graphic depends on the scaling option; this was discussed in Section 2.4.4.

Table 5.1 Natural Frequencies

Table 5.1 provides the natural frequencies of the undamped system, the result table is shown in Figure 4.2.

5.1 Natural Frequencies

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

Mode No.	A Eigenvalue λ [1/s²]	B Angular Frequency ω [rad/s]	C Natural frequency f [Hz]	D Natural period T [s]	E
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	

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 4.2: Result 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 in Bathe [6] or Tedesco [9] for example. The equation of motion is defined with

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

where $\mathbf{M}$ is the mass matrix discussed in Section 2.4.3, $\mathbf{K}$ the stiffness matrix, and $\mathbf{u}$ 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 λ_i and corresponding mode shapes $\mathbf{u_i}$ exist for each mode i.

Table 5.2 - 5.5 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 standardized displacements u_X, u_Y, and u_Z and rotations φ_X, φ_Y, and φ_Z are sorted by nodes; this table is illustrated in Figure 4.3.

5.2 Mode Shapes by Node

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

Node No.	Mode No.	Standardized Displacements			Standardized Rotations		
		u_X [-]	u_Y [-]	u_Z [-]	φ_X [1/m]	φ_Y [1/m]	φ_Z [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 4.3: Result Table 5.2: Mode Shapes sorted by nodes. The standardized displacements u_X, u_Y, and u_Z and the standardized rotations φ_X, φ_Y, and φ_Z are provided.

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

The maximum value of u and φ is dependent on the chosen scaling option discussed in Section 2.4.4. The standardized displacements u are dimensionless, and the standardized rotations φ are given in $1/m$. These units result from the scaling procedure.

Table 5.6 Masses in Mesh Points

Depending on the imported masses (see Section 2.2), on the settings in the *Details* dialog box (see Section 2.9), and on the final assigned *Mass Case* or *Mass Combination* (see Section 2.4.2) the masses in mesh points are calculated. Provided here are only the masses in the translational directions m_X, m_Y, and m_Z. The sum of the masses is provided at the bottom of the table. The table is shown in Figure 4.4. Beside that the positions of the mesh points in the global coordinate system are listed.

5.6 Masses in Mesh Points

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

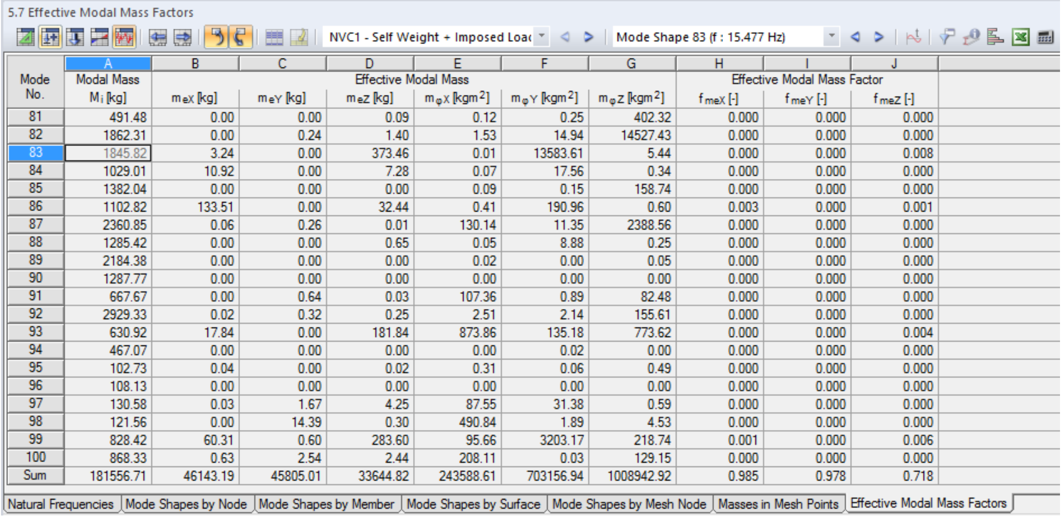
Mesh Point No.	Object Type	Object No.	Location X [m]	Location Y [m]	Location Z [m]	Mass m_X [kg]	Mass m_Y [kg]	Mass m_Z [kg]
614	Member	97	14.000	5.000	-4.500	25.24	25.24	25.24
615	Member	97	14.000	5.000	-5.000	25.24	25.24	25.24
616	Member	97	14.000	5.000	-5.500	25.24	25.24	25.24
617	Member	97	14.000	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 4.4: Result Table 5.6: The masses in the translational directions m_X, m_Y, and m_Z determined for each mesh point are listed. The sum of the masses is provided at the bottom of the table.

Table 5.7 Effective Modal Mass Factors

The Table 5.7 provides modal masses $\mathbf{M_i}$, effective modal masses m_e and effective modal mass factors f_{me}; the table is illustrated in Figure 4.5. The effective modal masses describe how much mass is activated by each eigenmode of the system in each direction.



5.7 Effective Modal Mass Factors

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

Mode No.	A Modal Mass M_i [kg]	B Effective Modal Mass m_{eX} [kg]	C m_{eY} [kg]	D m_{eZ} [kg]	E $m_{\varphi X}$ [kgm²]	F $m_{\varphi Y}$ [kgm²]	G $m_{\varphi Z}$ [kgm²]	H Effective Modal Mass Factor f_{meX} [-]	I f_{meY} [-]	J f_{meZ} [-]
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 4.5: Result Table 5.7: Modal masses M_i, effective modal masses in the translational directions m_{eX}, m_{eY}, and m_{eZ}, and around the global axis $m_{\varphi X}$, $m_{\varphi Y}$, and $m_{\varphi Z}$ are listed. Also provided are the effective modal mass factors f_{meX}, f_{meY}, and f_{meZ}.

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 $\mathbf{M}$ is the mass matrix discussed in Section 2.4.3. The modal mass $\mathbf{M_i}$ is independent of direction. The modal mass changes depending on the scaling option chosen for the mode shapes (see Section 2.4.4); 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 $\Gamma_{i,j}$.

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

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

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

with X, Y and Z being the global coordinates of the considered FE-node provided in Table 5.6 (see Figure 4.4), and X_0, Y_0 and Z_0 being the center of total mass provided in the **Mass Cases** → **General** tab in *RF-DYNAM Pro* (see Section 2.2). This matrix $\mathbf{T}$ exists for each FE-node. The definition of the participation factors defining also the rotational degrees of freedom is detailed in [12] in Section 15.7.5. The participation factor Γ_{ij} is dimensionless for translations and has the unit $[m]$ for rotations.
The effective modal masses are defined with

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

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 kgm².

The sum of the effective modal masses $\sum m_e$ is provided in Table 5.7 (see Figure 4.5) and in the translational directions these sums are equal to the total sum of the structure M_{total} (see Table

5.6 in Figure 4.4) beside masses that are not activated, *i.e.* masses in fixed supports, and assuming that all eigenvalues of the system are calculated.

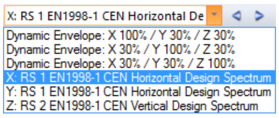
The effective modal mass factors f_{me} are needed to decide whether or not a specific mode must be taken into account for the response spectra analysis or the equivalent load analysis. The **EN 1998-1** states in **Section 4.3.3.3** that "the sum of the effective modal masses for the modes taken into 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 [1]. The effective modal mass factors f_{me} are defined as follows:

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

Further details about modal analysis can be found for example in the books written by Meskouris *et al.* [13] and Tedesco *et al.* [9].

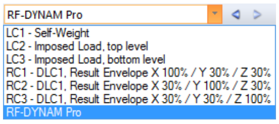
4.2 Dynamic Load Cases - Response Spectra Analysis

The result tables that belong to *Response Spectrum Analysis Cases* are available when the corresponding *DLC* case is chosen in the drop-down menu. The result 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 were discussed in Sections 2.5 and 2.8.1.



You can switch between results in the separate excitation directions X, Y and Z, and between 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 area of RFEM update according to your choice.

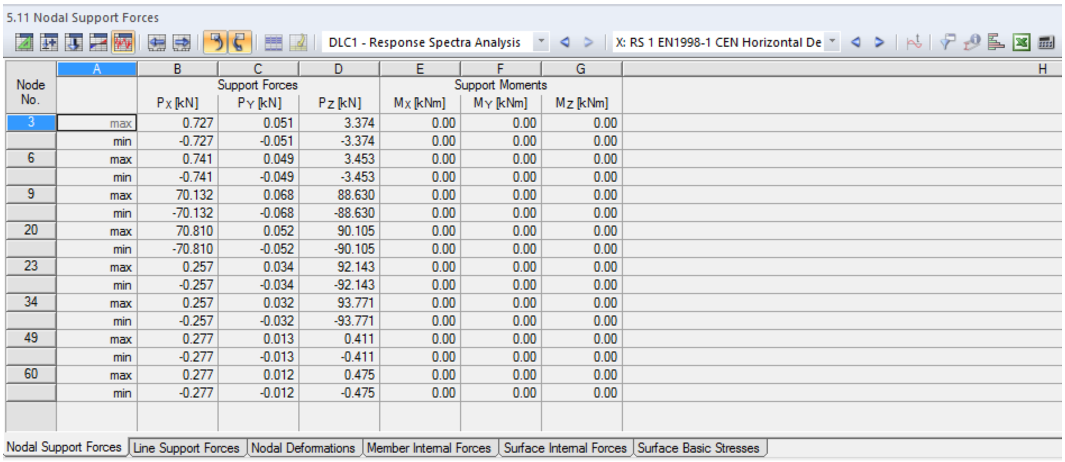
From a response spectrum analysis in the *RF-DYNAM Pro - Forced Vibrations* you get result combinations only; this means the modal responses are combined with the *SRSS* or *CQC* rule according to your choice. This is why maximum and minimum values are provided in all result tables.



When you selected the export of result combinations as discussed in Section 2.8.1, the *RCs* are available in the drop-down list in the *Toolbar* in RFEM shown on the left. You have also access to the *RCs* using **Data Project Navigator** → **Load Cases and Combinations** → **Result Combinations** or from the *Edit Load Cases and Combinations* dialog box accessed with the button.

Table 5.11 and 5.12: Support Forces

The nodal support forces are provided in Table 5.11, shown in Figure 4.6. Similarly, the line support forces are listed in Table 5.12. Maximum and minimum values are provided.



5.11 Nodal Support Forces

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

Node No.	A	B	C	D	E	F	G
		Support Forces			Support Moments		
		Px [kN]	PY [kN]	Pz [kN]	Mx [kNm]	MY [kNm]	Mz [kNm]
3	max	0.727	0.051	3.374	0.00	0.00	0.00
	min	-0.727	-0.051	-3.374	0.00	0.00	0.00
6	max	0.741	0.049	3.453	0.00	0.00	0.00
	min	-0.741	-0.049	-3.453	0.00	0.00	0.00
9	max	70.132	0.068	88.630	0.00	0.00	0.00
	min	-70.132	-0.068	-88.630	0.00	0.00	0.00
20	max	70.810	0.052	90.105	0.00	0.00	0.00
	min	-70.810	-0.052	-90.105	0.00	0.00	0.00
23	max	0.257	0.034	92.143	0.00	0.00	0.00
	min	-0.257	-0.034	-92.143	0.00	0.00	0.00
34	max	0.257	0.032	93.771	0.00	0.00	0.00
	min	-0.257	-0.032	-93.771	0.00	0.00	0.00
49	max	0.277	0.013	0.411	0.00	0.00	0.00
	min	-0.277	-0.013	-0.411	0.00	0.00	0.00
60	max	0.277	0.012	0.475	0.00	0.00	0.00
	min	-0.277	-0.012	-0.475	0.00	0.00	0.00

Nodal Support Forces | Line Support Forces | Nodal Deformations | Member Internal Forces | Surface Internal Forces | Surface Basic Stresses

Figure 4.6: Result Table 5.11: Maximum and minimum nodal support forces and moments in the three directions X, Y and Z are listed.

Table 5.13: Nodal Deformations

The nodal deformations are provided in Table 5.13 as shown in Figure 4.7. Maximum and minimum values are provided.

5.13 Nodal Deformations

Node No.	A	B Displacements u_X [mm]	C u_Y [mm]	D u_Z [mm]	E Rotations φ_X [mrad]	F φ_Y [mrad]	G φ_Z [mrad]
	min	0.0	0.0	0.0	-4.0	0.0	0.0
24	max	0.0	17.5	0.2	0.1	0.0	0.0
	min	0.0	-17.5	-0.2	-0.1	0.0	0.0
25	max	0.0	12.5	0.6	0.1	0.0	0.0
	min	0.0	-12.5	-0.6	-0.1	0.0	0.0
26	max	0.0	17.5	0.1	0.2	0.0	0.0
	min	0.0	-17.5	-0.1	-0.2	0.0	0.0
27	max	0.0	12.5	0.3	0.5	0.0	0.0
	min	0.0	-12.5	-0.3	-0.5	0.0	0.0
28	max	0.0	17.5	0.1	0.2	0.0	0.0

Nodal Support Forces | Line Support Forces | Nodal Deformations | Member Internal Forces | Surface Internal Forces | Surface Basic Stresses

Figure 4.7: Result Table 5.13: Maximum and minimum nodal displacements u_X, u_Y, u_Z, and rotations φ_X, φ_Y and φ_Z are listed.

Table 5.14: Member Internal Forces

The internal forces of members are listed in Table 5.14 as shown in Figure 4.8. Maximum and minimum values are provided.

5.14 Member Internal Forces

Member No.	A At Point Of	B Normal Force N [N]	C Shear Force V_y [N]	D V_z [N]	E Moments M_X [Nm]	F M_Y [Nm]	G M_Z [Nm]
1	min N	**-39.035**	-3.030	-75.863	-0.01	-163.65	-7.77
	max N	**39.035**	3.030	75.863	0.01	163.65	7.77
	min Vy	-38.830	**-3.174**	-76.826	-0.01	-27.50	-0.71
	max Vy	38.830	**3.174**	76.826	0.01	27.50	0.71
	min Vz	-38.784	-3.152	**-76.863**	-0.01	-65.74	-1.63
	max Vz	38.784	3.152	**76.863**	0.01	65.74	1.63
	min Mx	-38.917	-3.141	-76.579	**-0.01**	-49.56	-3.23
	max Mx	38.917	3.141	76.579	**0.01**	49.56	3.23
	min My	-39.035	-3.030	-75.863	-0.01	**-163.65**	-7.77
	max My	39.035	3.030	75.863	0.01	**163.65**	7.77
	min Mz	-39.035	-3.030	-75.863	-0.01	-163.65	**-7.77**
	max Mz	39.035	3.030	75.863	0.01	163.65	**7.77**

Nodal Support Forces | Line Support Forces | Nodal Deformations | Member Internal Forces | Surface Internal Forces | Surface Basic Stresses

Figure 4.8: Result Table 5.14: Maximum and minimum internal normal forces N, shear forces V_y and V_Z, and moments M_X, M_Y, and M_Z are listed.

Table 5.15 and 5.16: Surface Results

Table 5.15 provides the surface internal forces as shown in Figure 4.9. Maximum and minimum values are provided together with point coordinates where the maximum or minimum value of the specific internal force occurs.

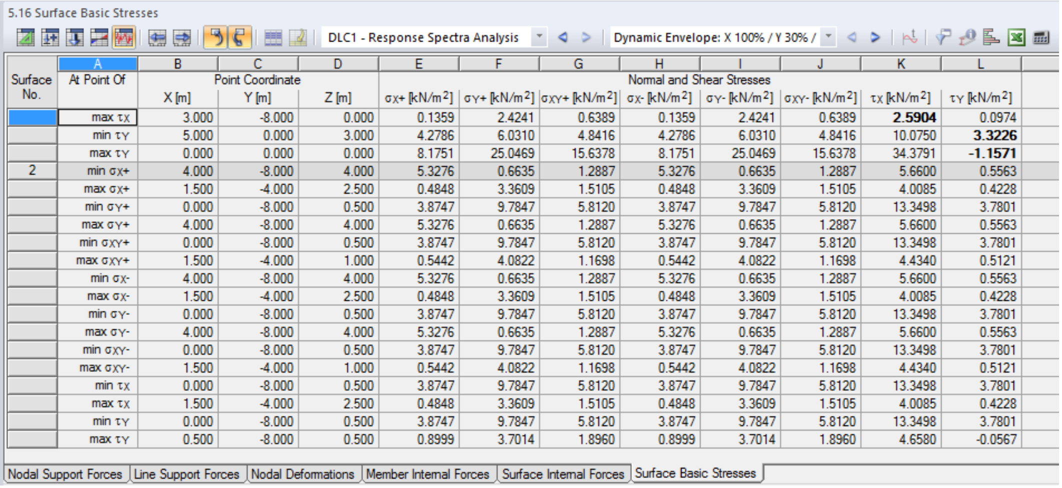
5.15 Surface Internal Forces

Surface No.	A At Point Of	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]
2	min m_x	0.000	-8.000	0.500	**15.713**	30.250	11.885	36.57	462.76	30.71	431.18	484.25
	max m_x	1.500	-4.000	2.500	**1.929**	9.916	3.716	4.38	125.77	5.24	165.90	102.32
	min m_y	0.000	-8.000	0.500	15.713	**30.250**	11.885	36.57	462.76	30.71	431.18	484.25
	max m_y	0.000	-8.000	0.000	3.455	**1.847**	11.889	199.50	178.66	552.70	600.24	190.25
	min m_{xy}	0.000	-8.000	4.000	2.132	18.799	**12.260**	84.54	90.23	161.35	31.05	75.48
	max m_{xy}	2.000	-4.000	1.500	6.508	12.766	**1.180**	3.08	4.43	60.72	123.17	175.66
	min v_x	0.000	-8.000	0.000	3.455	1.847	11.889	**199.50**	178.66	552.70	600.24	190.25
	max v_x	2.000	-4.000	3.500	7.243	13.768	1.600	**2.77**	4.81	72.72	130.97	169.69
	min v_y	0.000	-8.000	0.500	15.713	30.250	11.885	36.57	**462.76**	30.71	431.18	484.25
	max v_y	0.500	-4.000	1.500	6.690	15.426	4.177	3.30	**2.59**	70.41	106.25	183.85
	min n_x	0.000	-8.000	0.000	3.455	1.847	11.889	199.50	178.66	**552.70**	600.24	190.25
	max n_x	1.500	-4.000	1.000	2.185	14.494	1.928	4.91	136.00	**5.12**	109.63	114.85
	min n_y	0.000	-8.000	0.000	3.455	1.847	11.889	199.50	178.66	552.70	**600.24**	190.25
	max n_y	2.000	-8.000	4.000	11.040	3.364	3.850	105.53	3.99	250.90	**3.36**	57.19
	min n_{xy}	0.000	-8.000	0.500	15.713	30.250	11.885	36.57	462.76	30.71	431.18	**484.25**
	max n_{xy}	2.000	-8.000	4.000	11.040	3.364	3.850	105.53	3.99	250.90	3.36	**57.19**

Nodal Support Forces | Line Support Forces | Nodal Deformations | Member Internal Forces | Surface Internal Forces | Surface Basic Stresses

Figure 4.9: Result Table 5.15: Maximum and minimum internal normal forces n_X, n_Y and n_Z, shear forces v_X and v_Y, and moments m_X, m_Y, and m_{XY} are listed.

Surface stresses are listed in Table 5.16 as shown in Figure 4.10.


5.16 Surface Basic Stresses

DLC1 - Response Spectra Analysis | Dynamic Envelope: X 100% / Y 30% /

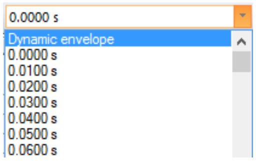
Surface No.	At Point Of	Point Coordinate X [m]	Y [m]	Z [m]	Normal and Shear Stresses σX+ [kN/m2]	σY+ [kN/m2]	σXY+ [kN/m2]	σX- [kN/m2]	σY- [kN/m2]	σXY- [kN/m2]	τX [kN/m2]	τY [kN/m2]
	max τX	3.000	-8.000	0.000	0.1359	2.4241	0.6389	0.1359	2.4241	0.6389	**2.5904**	0.0974
	min τY	5.000	0.000	3.000	4.2786	6.0310	4.8416	4.2786	6.0310	4.8416	10.0750	**3.3226**
	max τY	0.000	0.000	0.000	8.1751	25.0469	15.6378	8.1751	25.0469	15.6378	34.3791	**-1.1571**
2	min σX+	4.000	-8.000	4.000	5.3276	0.6635	1.2887	5.3276	0.6635	1.2887	5.6600	0.5563
	max σX+	1.500	-4.000	2.500	0.4848	3.3609	1.5105	0.4848	3.3609	1.5105	4.0085	0.4228
	min σY+	0.000	-8.000	0.500	3.8747	9.7847	5.8120	3.8747	9.7847	5.8120	13.3498	3.7801
	max σY+	4.000	-8.000	4.000	5.3276	0.6635	1.2887	5.3276	0.6635	1.2887	5.6600	0.5563
	min σXY+	0.000	-8.000	0.500	3.8747	9.7847	5.8120	3.8747	9.7847	5.8120	13.3498	3.7801
	max σXY+	1.500	-4.000	1.000	0.5442	4.0822	1.1698	0.5442	4.0822	1.1698	4.4340	0.5121
	min σX-	4.000	-8.000	4.000	5.3276	0.6635	1.2887	5.3276	0.6635	1.2887	5.6600	0.5563
	max σX-	1.500	-4.000	2.500	0.4848	3.3609	1.5105	0.4848	3.3609	1.5105	4.0085	0.4228
	min σY-	0.000	-8.000	0.500	3.8747	9.7847	5.8120	3.8747	9.7847	5.8120	13.3498	3.7801
	max σY-	4.000	-8.000	4.000	5.3276	0.6635	1.2887	5.3276	0.6635	1.2887	5.6600	0.5563
	min σXY-	0.000	-8.000	0.500	3.8747	9.7847	5.8120	3.8747	9.7847	5.8120	13.3498	3.7801
	max σXY-	1.500	-4.000	1.000	0.5442	4.0822	1.1698	0.5442	4.0822	1.1698	4.4340	0.5121
	min τX	0.000	-8.000	0.500	3.8747	9.7847	5.8120	3.8747	9.7847	5.8120	13.3498	3.7801
	max τX	1.500	-4.000	2.500	0.4848	3.3609	1.5105	0.4848	3.3609	1.5105	4.0085	0.4228
	min τY	0.000	-8.000	0.500	3.8747	9.7847	5.8120	3.8747	9.7847	5.8120	13.3498	3.7801
	max τY	0.500	-8.000	0.500	0.8999	3.7014	1.8960	0.8999	3.7014	1.8960	4.6580	-0.0567

Nodal Support Forces | Line Support Forces | Nodal Deformations | Member Internal Forces | Surface Internal Forces | Surface Basic Stresses

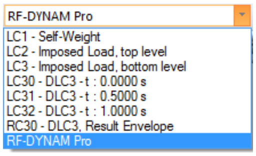
Figure 4.10: Result Table 5.16: Maximum and minimum normal and shear stresses are listed.

4.3 Dynamic Load Cases - Time History Analysis

The result tables that belong to *Time History Analysis Cases* are available when the corresponding *DLC* case is chosen in the drop-down menu. The result tables 5.17 to 5.24 belong to this type of *Dynamic Load Cases*. The *Time History Analysis* belongs to the module *RF-DYNAM Pro - Forced Vibrations*. The input data required for a time history analysis of accelerograms were discussed in Sections 2.6 and 2.8.2 and that for a time history analysis of time diagrams were discussed in Sections 2.7 and 2.8.3.



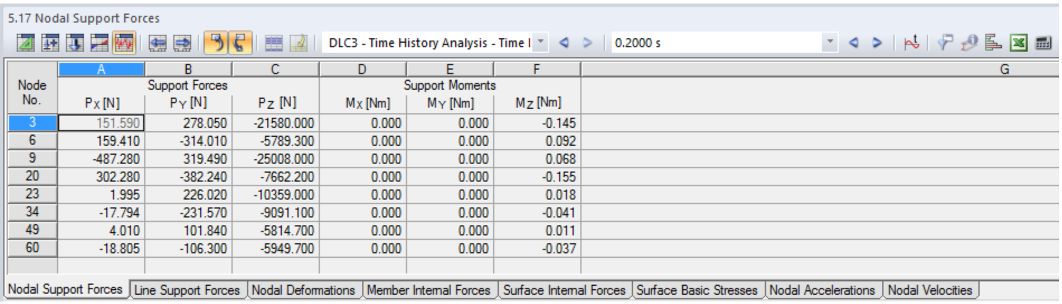
Results are available separately for each time step, also available is an result envelope providing maximum and minimum results of all time steps. You can switch between the results using the drop-down menu shown on the left. The tabulated results and also the graphic in the work area of RFEM update according to your choice. The results can also displayed versus time in the *Time Course Monitor* which is accessible with the  button.



When you selected the export of load cases and / or result combinations as discussed in Sections 2.8.2 and 2.8.3, the *LCs* and *RCs* are available in the drop-down list in the *Toolbar* in RFEM shown on the left. You have also access to the *LCs* by **Data Project Navigator** → **Load Cases and Combinations** → **Load Cases** or similarly for the *RCs* by **Result Combinations**. Load cases and result combinations are also accessible in the *Edit Load Cases and Combinations* dialog box.

Table 5.17 and 5.18: Support Forces

The nodal support forces are provided in Table 5.17, shown in Figure 4.11. Similarly, the line support forces are listed in Table 5.18. Maximum and minimum values are provided in the case of the *Dynamic Envelope*, but in case of a specific time step only one value per node is listed.


5.17 Nodal Support Forces

DLC3 - Time History Analysis - Time I | 0.2000 s

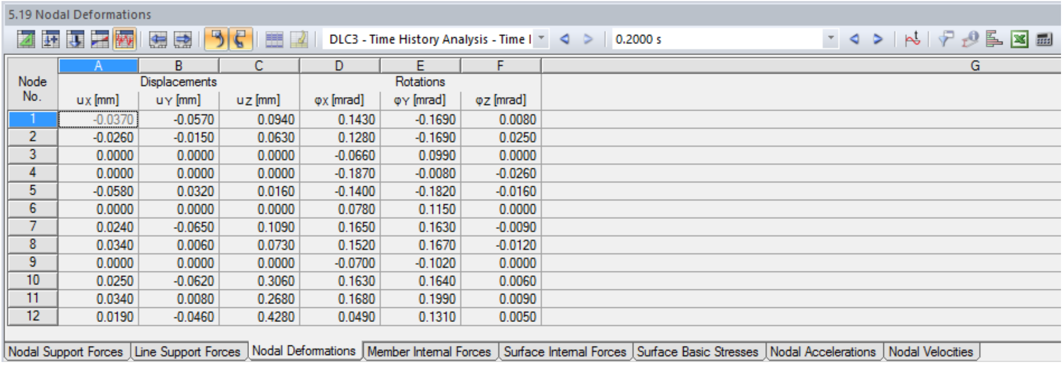
Node No.	Support Forces Px [N]	Py [N]	Pz [N]	Support Moments Mx [Nm]	My [Nm]	Mz [Nm]
3	151.590	278.050	-21580.000	0.000	0.000	-0.145
6	159.410	-314.010	-5789.300	0.000	0.000	0.092
9	-487.280	319.490	-25008.000	0.000	0.000	0.068
20	302.280	-382.240	-7662.200	0.000	0.000	-0.155
23	1.995	226.020	-10359.000	0.000	0.000	0.018
34	-17.794	-231.570	-9091.100	0.000	0.000	-0.041
49	4.010	101.840	-5814.700	0.000	0.000	0.011
60	-18.805	-106.300	-5949.700	0.000	0.000	-0.037

Nodal Support Forces | Line Support Forces | Nodal Deformations | Member Internal Forces | Surface Internal Forces | Surface Basic Stresses | Nodal Accelerations | Nodal Velocities

Figure 4.11: Result Table 5.17: Nodal support forces and moments in the three directions X, Y and Z are listed.

Table 5.19: Nodal Deformations

The nodal deformations are provided in Table 5.19 as shown in Figure 4.12. Maximum and minimum values are provided in the case of the *Dynamic Envelope*, but in case of a specific time step only one value per node is listed.

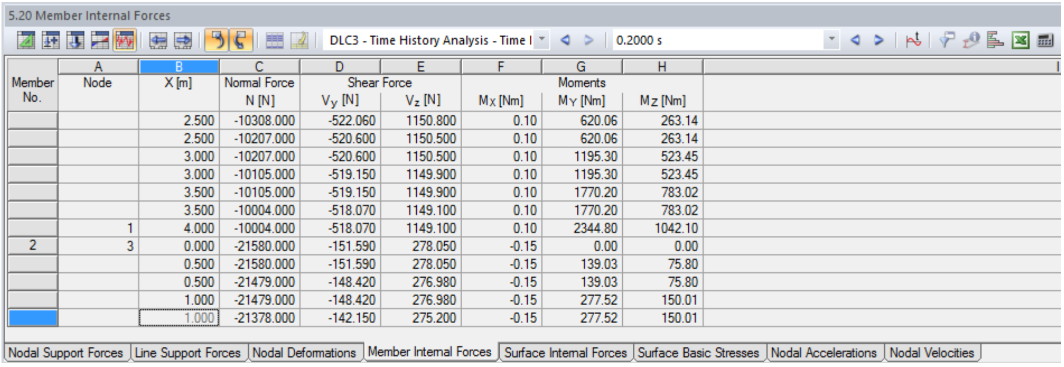


Node No.	Displacements u_X [mm]	u_Y [mm]	u_Z [mm]	Rotations φ_X [mrad]	φ_Y [mrad]	φ_Z [mrad]
1	-0.0370	-0.0570	0.0940	0.1430	-0.1690	0.0080
2	-0.0260	-0.0150	0.0630	0.1280	-0.1690	0.0250
3	0.0000	0.0000	0.0000	-0.0660	0.0990	0.0000
4	0.0000	0.0000	0.0000	-0.1870	-0.0080	-0.0260
5	-0.0580	0.0320	0.0160	-0.1400	-0.1820	-0.0160
6	0.0000	0.0000	0.0000	0.0780	0.1150	0.0000
7	0.0240	-0.0650	0.1090	0.1650	0.1630	-0.0090
8	0.0340	0.0060	0.0730	0.1520	0.1670	-0.0120
9	0.0000	0.0000	0.0000	-0.0700	-0.1020	0.0000
10	0.0250	-0.0620	0.3060	0.1630	0.1640	0.0060
11	0.0340	0.0080	0.2680	0.1680	0.1990	0.0090
12	0.0190	-0.0460	0.4280	0.0490	0.1310	0.0050

Figure 4.12: Result Table 5.19: Nodal displacements u_X, u_Y, u_Z, and rotations φ_X, φ_Y and φ_Z are listed.

Table 5.20: Member Internal Forces

The internal forces of members are listed in Table 5.20 as shown in Figure 4.13. Maximum and minimum values are provided in the case of the *Dynamic Envelope*, but in case of a specific time step only one value per position is listed.

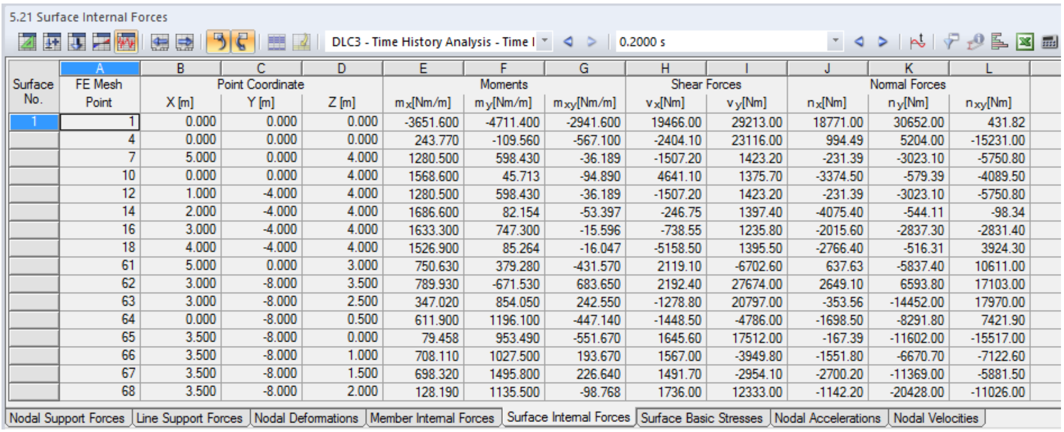


Member No.	Node	X [m]	Normal Force N [N]	Shear Force V_y [N]	V_z [N]	Moments M_X [Nm]	M_Y [Nm]	M_Z [Nm]
		2.500	-10308.000	-522.060	1150.800	0.10	620.06	263.14
		2.500	-10207.000	-520.600	1150.500	0.10	620.06	263.14
		3.000	-10207.000	-520.600	1150.500	0.10	1195.30	523.45
		3.000	-10105.000	-519.150	1149.900	0.10	1195.30	523.45
		3.500	-10105.000	-519.150	1149.900	0.10	1770.20	783.02
		3.500	-10004.000	-518.070	1149.100	0.10	1770.20	783.02
	1	4.000	-10004.000	-518.070	1149.100	0.10	2344.80	1042.10
2	3	0.000	-21580.000	-151.590	278.050	-0.15	0.00	0.00
		0.500	-21580.000	-151.590	278.050	-0.15	139.03	75.80
		0.500	-21479.000	-148.420	276.980	-0.15	139.03	75.80
		1.000	-21479.000	-148.420	276.980	-0.15	277.52	150.01
		1.000	-21378.000	-142.150	275.200	-0.15	277.52	150.01

Figure 4.13: Result Table 5.20: Internal normal forces N, shear forces V_y and V_Z, and moments M_X, M_Y, and M_Z are listed.

Table 5.21 and 5.22: Surface Results

Table 5.21 provides the surface internal forces, this is shown in Figure 4.14.



Surface No.	FE Mesh Point	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[Nm]	v_y[Nm]	Normal Forces n_x[Nm]	n_y[Nm]	n_{xy}[Nm]
1	1	0.000	0.000	0.000	-3651.600	-4711.400	-2941.600	19466.00	29213.00	18771.00	30652.00	431.82
	4	0.000	0.000	0.000	243.770	-109.560	-567.100	-2404.10	23116.00	994.49	5204.00	-15231.00
	7	5.000	0.000	4.000	1280.500	598.430	-36.189	-1507.20	1423.20	-231.39	-3023.10	-5750.80
	10	0.000	0.000	4.000	1568.600	45.713	-94.890	4641.10	1375.70	-3374.50	-579.39	-4089.50
	12	1.000	-4.000	4.000	1280.500	598.430	-36.189	-1507.20	1423.20	-231.39	-3023.10	-5750.80
	14	2.000	-4.000	4.000	1686.600	82.154	-53.397	-246.75	1397.40	-4075.40	-544.11	-98.34
	16	3.000	-4.000	4.000	1633.300	747.300	-15.596	-738.55	1235.80	-2015.60	-2837.30	-2831.40
	18	4.000	-4.000	4.000	1526.900	85.264	-16.047	-5158.50	1395.50	-2766.40	-516.31	3924.30
	61	5.000	0.000	3.000	750.630	379.280	-431.570	2119.10	-6702.60	637.63	-5837.40	10611.00
	62	3.000	-8.000	3.500	789.930	-671.530	683.650	2192.40	27674.00	2649.10	6593.80	17103.00
	63	3.000	-8.000	2.500	347.020	854.050	242.550	-1278.80	20797.00	-353.56	-14452.00	17970.00
	64	0.000	-8.000	0.500	611.900	1196.100	-447.140	-1448.50	-4786.00	-1698.50	-8291.80	7421.90
	65	3.500	-8.000	0.000	79.458	953.490	-551.670	1645.60	17512.00	-167.39	-11602.00	-15517.00
	66	3.500	-8.000	1.000	708.110	1027.500	193.670	1567.00	-3949.80	-1551.80	-6670.70	-7122.60
	67	3.500	-8.000	1.500	698.320	1495.800	226.640	1491.70	-2954.10	-2700.20	-11369.00	-5881.50
	68	3.500	-8.000	2.000	128.190	1135.500	-98.768	1736.00	12333.00	-1142.20	-20428.00	-11026.00

Figure 4.14: Result Table 5.21: Internal normal forces n_X, n_Y and n_Z, shear forces v_X and v_Y, and moments m_X, m_Y, and m_{XY} are listed.

Surface stresses are listed in Table 5.22 as shown in Figure 4.15.

5.22 Surface Basic Stresses

DLC3 - Time History Analysis - Time I | 0.2000 s

Surface No.	FE Mesh Point	Point Coordinate X [m]	Y [m]	Z [m]	Normal and Shear Stresses σ_{x+} [kN/m²]	σ_{y+} [kN/m²]	σ_{xy+} [kN/m²]	σ_{x-} [kN/m²]	σ_{y-} [kN/m²]	σ_{xy-} [kN/m²]	τ_x [kN/m²]	τ_y [kN/m²]
1	1	0.000	0.000	0.000	-738.5250	-912.6594	-686.7386	973.1625	1295.8094	692.1364	-133.3562	-1517.8281
	4	0.000	0.000	0.000	63.3492	6.8469	-228.1078	-50.9180	58.2031	37.7203	264.9486	-194.7526
	7	5.000	0.000	4.000	298.6710	121.3627	-44.4243	-301.5634	-159.1514	-27.4607	309.1787	110.8549
	10	0.000	0.000	4.000	346.5500	7.0928	-47.7992	-388.7312	-14.3352	-3.3195	353.1522	0.4906
	12	1.000	-4.000	4.000	298.6710	121.3627	-44.4243	-301.5634	-159.1514	-27.4607	309.1787	110.8549
	14	2.000	-4.000	4.000	369.8256	15.8542	-13.1295	-420.7681	-22.6555	11.9003	370.3119	15.3678
	16	3.000	-4.000	4.000	370.2072	157.4153	-21.3516	-395.4022	-192.8816	-14.0409	372.3285	155.2940
	18	4.000	-4.000	4.000	340.5772	16.7568	20.7659	-375.1572	-23.2107	28.2879	341.9034	15.4306
	61	5.000	0.000	3.000	179.9141	52.4100	-34.8305	-171.9437	-125.3775	167.4680	188.8084	43.5157
	62	3.000	-8.000	3.500	201.6967	-116.1786	267.1242	-168.5830	198.6011	-53.3367	353.5911	-268.0729
	63	3.000	-8.000	2.500	79.1231	109.8430	169.1602	-83.5426	-290.4930	55.4648	264.3391	-75.3731
	64	0.000	-8.000	0.500	132.7984	228.5122	-58.4116	-154.0297	-332.1597	151.1853	256.1682	105.1425
	65	3.500	-8.000	0.000	17.5768	150.9617	-226.2789	-19.6692	-295.9867	32.3164	320.1718	-151.6333
	66	3.500	-8.000	1.000	156.2645	199.1284	0.8752	-175.6620	-282.5122	-89.9077	199.1463	156.2467
	67	3.500	-8.000	1.500	146.7925	279.5219	16.3594	-180.5450	-421.6344	-89.8781	281.5085	144.8059
	68	3.500	-8.000	2.000	22.9058	138.4578	-92.0613	-37.1833	-393.8078	-45.7638	189.3710	-28.0074

Nodal Support Forces | Line Support Forces | Nodal Deformations | Member Internal Forces | Surface Internal Forces | Surface Basic Stresses | Nodal Accelerations | Nodal Velocities

Figure 4.15: Result Table 5.22: Normal and shear stresses are listed.

Table 5.23 and 5.24: Nodal Accelerations and Velocities

Additionally to the standard results, nodal accelerations and velocities are provided in Table 5.23 shown in Figure 4.16 and Table 5.24 shown in Figure 4.17. These results are available for each time step and also the dynamic envelope is provided.

5.23 Nodal Accelerations

DLC3 - Time History Analysis - Time I | 0.2000 s

Node No.	Accelerations $\ddot{u}_X$ [m/s²]	$\ddot{u}_Y$ [m/s²]	$\ddot{u}_Z$ [m/s²]	Angular Accelerations $\ddot{\varphi}_X$ [rad/s²]	$\ddot{\varphi}_Y$ [rad/s²]	$\ddot{\varphi}_Z$ [rad/s²]
1	0.0131	0.0211	-0.0236	-0.043	0.073	-0.004
2	0.0255	0.0064	-0.0174	-0.060	0.088	-0.004
3	0.0000	0.0000	0.0000	0.020	-0.063	0.000
4	0.0000	0.0000	0.0000	0.063	0.012	0.013
5	0.0182	-0.0109	-0.0084	0.053	0.064	0.008
6	0.0000	0.0000	0.0000	-0.014	-0.028	0.000
7	-0.0110	0.0219	-0.0244	-0.047	-0.072	0.005
8	-0.0082	0.0088	-0.0172	-0.061	-0.097	0.003
9	0.0000	0.0000	0.0000	0.026	0.080	0.000
10	-0.0119	0.0211	-0.0784	-0.047	-0.073	-0.001
11	-0.0079	0.0079	-0.0935	-0.061	-0.096	-0.002
12	-0.0126	0.0162	-0.1131	-0.016	-0.074	-0.001
13	-0.0066	0.0015	-0.1348	-0.011	-0.088	0.000
14	-0.0099	0.0098	-0.1116	0.023	-0.060	-0.002
15	-0.0075	-0.0050	-0.1193	0.033	-0.087	0.001
16	-0.0063	0.0038	-0.0672	0.061	-0.038	-0.003

Nodal Support Forces | Line Support Forces | Nodal Deformations | Member Internal Forces | Surface Internal Forces | Surface Basic Stresses | Nodal Accelerations | Nodal Velocities

Figure 4.16: Result Table 5.23: Nodal acceleration $\ddot{u}_X$, $\ddot{u}_Y$, $\ddot{u}_Z$, and rotations $\ddot{\varphi}_X$, $\ddot{\varphi}_Y$ and $\ddot{\varphi}_Z$ are listed.

5.24 Nodal Velocities

DLC3 - Time History Analysis - Time I | 0.2000 s

Node No.	Velocities $\dot{u}_X$ [m/s]	$\dot{u}_Y$ [m/s]	$\dot{u}_Z$ [m/s]	Angular Velocities $\dot{\varphi}_X$ [rad/s]	$\dot{\varphi}_Y$ [rad/s]	$\dot{\varphi}_Z$ [rad/s]
1	-0.000132	-0.000291	0.000463	0.000708	-0.000768	0.000045
2	0.000160	-0.000367	0.000320	0.000757	-0.000740	0.000262
3	0.000000	0.000000	0.000000	-0.000491	-0.000253	0.000000
4	0.000000	0.000000	0.000000	-0.000925	-0.000046	-0.000133
5	-0.000683	-0.000102	0.000091	-0.000779	-0.001135	0.000031
6	0.000000	0.000000	0.000000	0.000253	0.000905	0.000000
7	0.000157	-0.000287	0.000543	0.000732	0.000784	-0.000043
8	0.000438	0.000240	0.000384	0.000894	0.000790	0.000028
9	0.000000	0.000000	0.000000	-0.000255	-0.000335	0.000000
10	0.000159	-0.000279	0.001447	0.000759	0.000817	0.000030
11	0.000338	0.000248	0.001534	0.001008	0.001079	0.000163
12	0.000132	-0.000207	0.002024	0.000237	0.000704	0.000030
13	0.000180	0.000337	0.002328	0.000389	0.001113	0.000160
14	0.000097	-0.000110	0.001921	-0.000462	0.000519	0.000031
15	0.000026	0.000456	0.002297	-0.000463	0.001138	0.000146
16	0.000050	-0.000031	0.001130	-0.001003	0.000234	0.000023

Nodal Support Forces | Line Support Forces | Nodal Deformations | Member Internal Forces | Surface Internal Forces | Surface Basic Stresses | Nodal Accelerations | Nodal Velocities

Figure 4.17: Result Table 5.24: Nodal velocities $\dot{u}_X$, $\dot{u}_Y$, $\dot{u}_Z$, and rotations $\dot{\varphi}_X$, $\dot{\varphi}_Y$ and $\dot{\varphi}_Z$ are listed.

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

Time Course Monitor

All results that are listed in the tables discussed above can be displayed versus the time with the *Time Course Monitor*. This is the most important post-processing tool of a time history analysis.

This graphic is available with the [Time Course Monitor] button. The button can be found in the table toolbar and in the *Panel* in the *Display Factors* tab.

The *Time Course Monitor* is displayed in Figure 4.18.

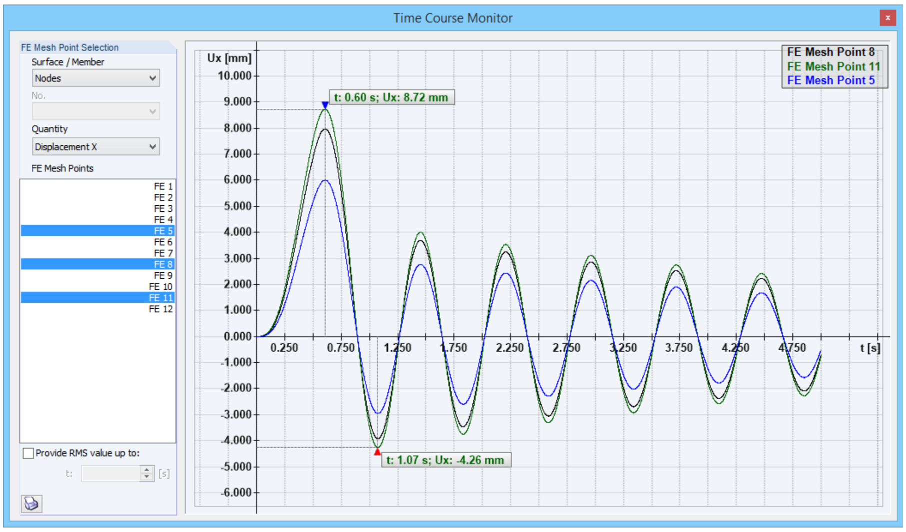
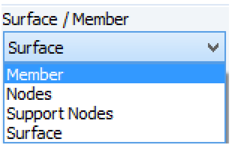
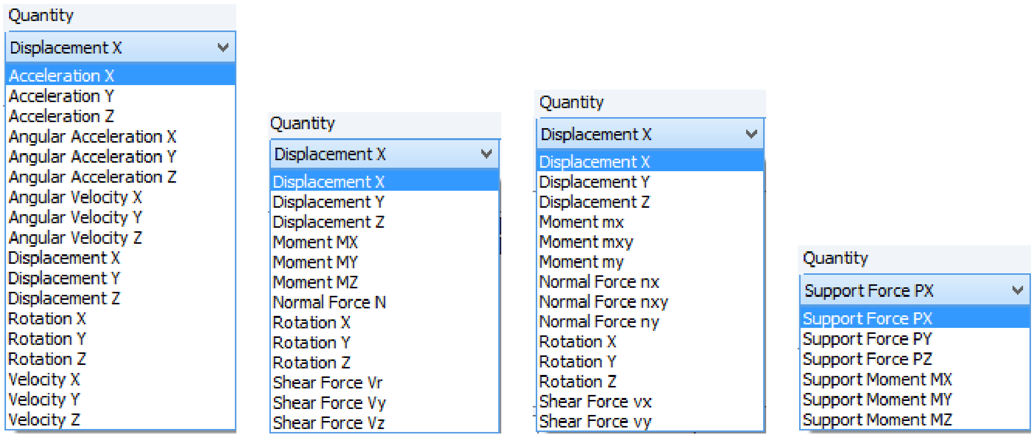


Figure 4.18: Time Course Monitor to display results versus time. Here the displacement u_X of three nodes are displayed. The maximum and minimum values are provided.

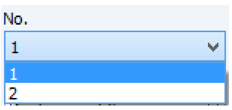


You can choose between nodes, members, surfaces, and support nodes in the drop-down menu shown on the left. Depending on this choice different results are available as shown in Figures 4.19a, 4.19b, 4.19c, and 4.19d.



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

Figure 4.19: Available results for (a) nodes, (b) members, (c) surfaces, and (d) support nodes.

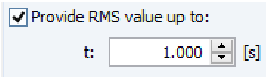


In the case of members and surfaces you have to choose the member or surface number from the drop-down list.

Depending on the choices above either a list of finite element mesh points (see Figure 4.18) or a list of distances on members (see Figure 4.20) is available. A multiple choice of listed points or positions is possible by using the [Ctrl] key.

The legend in the graphic is set automatically and also the axis labels and axis scales are adjusted depending on the choices you made. Minimum and maximum results together with the corresponding time steps are provided. You can use your mouse wheel to zoom into the graph. Values are displayed on the tip of your mouse cursor when moving along the graph.

The work area of RFEM, the tables and the *Time Course Monitor* are interacting. So when you select a row with a specific member in a table or when you select a node in the graphic, for example, the settings in the *Time Course Monitor* are adjusted. You only have to choose the result that shall be displayed (Figures 4.19a - 4.19d).



The *Root Mean Square* (RMS) value of any results up to a specific time can be displayed optionally, when the check box shown on the left is selected and a time is provided. The RMS value is calculated as follows

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

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*; this is shown in Figure 4.20.

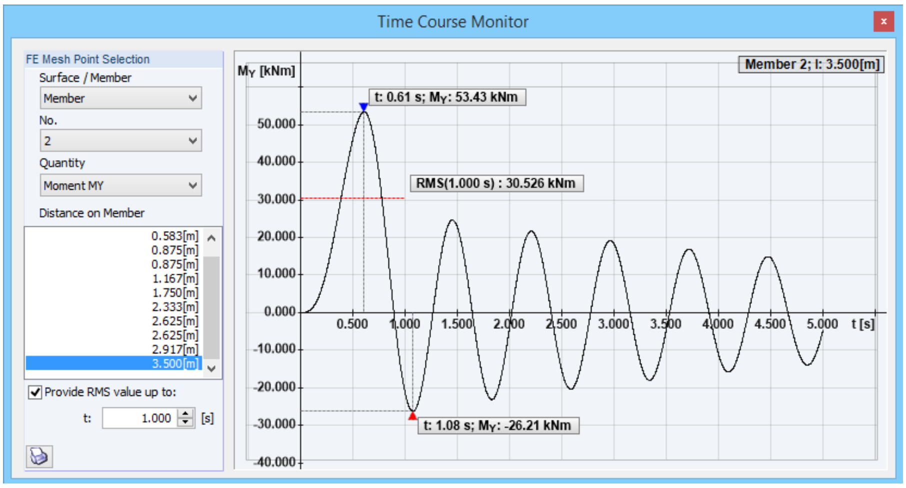
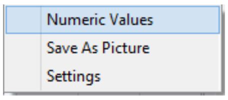


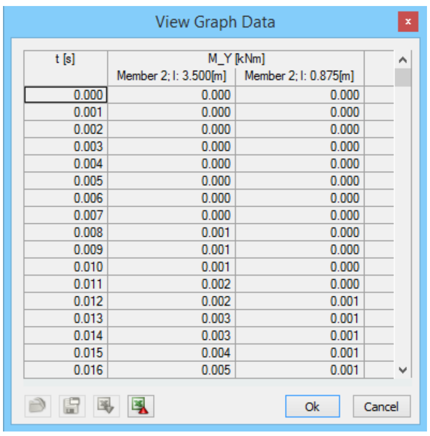
Figure 4.20: Time Course Monitor with the *Root Mean Square* (RMS) value. Here the moment M_Y is displayed versus time, the RMS value up to $1.0\ sec$ is provided.

The *Root Mean Square* value is only available when a single graph is plotted. When you overlay results of several nodes the RMS check box is not available.

With the [Print] button you can print the *Time Course Monitor* in the printout report directly. In the window appearing you can define a header for this graphic to be used in the report.



When you right-click on the *Time Couse Monitor* you have three options: (1) You can access the numerical values in tabular view. The time steps are listed together with all results displayed in the graphic. An example is provided in Figure 4.21, the values as tabulated can be exported to *Excel* using the button. (2) You can save the *Time Course Monitor* as picture, and (3) you can access the settings of the graph. The settings dialog box of the *Time Course Monitor* is shown in Figure 4.22.


View Graph Data

t [s]	M_Y [kNm]	
	Member 2; l: 3.500[m]	Member 2; l: 0.875[m]
0.000	0.000	0.000
0.001	0.000	0.000
0.002	0.000	0.000
0.003	0.000	0.000
0.004	0.000	0.000
0.005	0.000	0.000
0.006	0.000	0.000
0.007	0.000	0.000
0.008	0.001	0.000
0.009	0.001	0.000
0.010	0.001	0.000
0.011	0.002	0.000
0.012	0.002	0.001
0.013	0.003	0.001
0.014	0.003	0.001
0.015	0.004	0.001
0.016	0.005	0.001

Ok Cancel

Figure 4.21: Numerical values of the plot displayed in the *Time Course Monitor*.

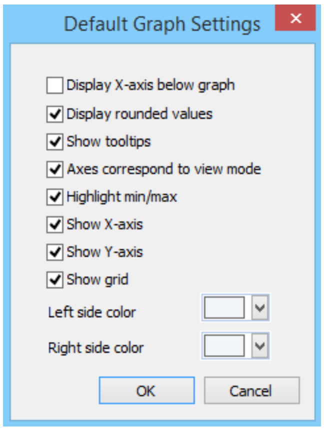
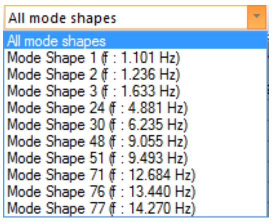


Figure 4.22: Settings of the *Time Course Monitor*.

4.4 Dynamic Load Cases - Equivalent Load Analysis

The result tables that belong to *Equivalent Load Analysis Cases* are available when the corresponding *DLC* case is chosen in the drop-down menu. The result 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 were discussed in Sections 2.5 and 2.8.4.



You can switch between tables showing the equivalent loads for all eigenvalues of the system, or separately for each mode.

Equivalent Loads

When performing an equivalent load analysis, only the equivalent loads are calculated in *RF-DYNAM Pro*, the remaining calculation is 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:

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

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

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

with

Γ	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 $M_i = \mathbf{u}_i^T \cdot \mathbf{M} \cdot \mathbf{u}_i = 1\,kg$
$\mathbf{S}_a(T)$	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, in Equation 4.10 the loads resulting from a response spectrum in Y-direction, and in Equation 4.11 the loads resulting from a response spectrum in Z-direction are calculated.

These equivalent static forces exist at each FE-node within the structure as long the mass M and the mode shape u are not 0 at this point.

The result tables for the equivalent load analysis are separated into the three excitation directions; in Table 5.8 the resulting equivalent loads for response spectra acting in X-direction, in Table 5.9 those for response spectra acting in Y-direction, and in Table 5.10 those for response spectra acting in Z-direction are listed. The Table 5.9 is illustrated in Figure 4.23.

5.9 Equivalent Loads (Y-excitations)

DLC4 - Equivalent Static Forces | All mode shapes

	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]
1	1	100	Surface	0.000	0.000	-8.000	-0.459	-113.355	0.353	1.401
	2	102	Surface	0.000	0.000	-8.000	-0.210	-34.396	0.103	0.423
	3	104	Surface	0.000	0.000	-8.000	0.237	-16.708	0.040	0.217
	4		Surface	0.000	0.000	-8.000	0.237	-16.708	0.040	0.217
	5		Surface	0.000	0.000	-8.000	0.237	-16.708	0.040	0.217
	6		Surface	0.000	0.000	-8.000	0.237	-16.708	0.040	0.217
	7		Surface	0.000	0.000	-8.000	0.237	-16.708	0.040	0.217
	8		Surface	0.000	0.000	-8.000	0.237	-16.708	0.040	0.217
	9		Surface	0.000	0.000	-8.000	0.237	-16.708	0.040	0.217
	10		Surface	0.000	0.000	-8.000	0.237	-16.708	0.040	0.217
	11		Surface	0.000	0.000	-8.000	0.237	-16.708	0.040	0.217
	12		Surface	0.000	0.000	-8.000	0.237	-16.708	0.040	0.217
	13		Surface	0.000	0.000	-8.000	0.237	-16.708	0.040	0.217
	14		Surface	0.000	0.000	-8.000	0.237	-16.708	0.040	0.217
	15		Surface	0.000	0.000	-8.000	0.237	-16.708	0.040	0.217
	16		Surface	0.000	0.000	-8.000	0.237	-16.708	0.040	0.217
	17		Surface	0.000	0.000	-8.000	0.237	-16.708	0.040	0.217
	18		Surface	0.000	0.000	-8.000	0.237	-16.708	0.040	0.217
	19		Surface	0.000	0.000	-8.000	0.237	-16.708	0.040	0.217

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

Figure 4.23: Result Table 5.9: Generated equivalent loads resulting from response spectra in Y-direction.

Torsional Moments

The accidental torsional actions can be considered as described in Section 2.8.4 to account for uncertainities in the location of masses. The center of mass is considered to be displaced by the accidental eccentricities e_X and e_Y. The torsional moments are calculated as follows

$$M_Z = |F_X \cdot e_Y| + |F_Y \cdot e_X| \tag{4.12}$$

where M_Z are the torsional moment and F_X and F_Y 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.

Exported Load Cases

The equivalent loads as listed in the dynamic result tables are exported as load cases into the main program RFEM. This is done separately for each eigenvalue and for each excitation direction. The list of load cases generated can be viewed in the *Edit Load Cases and Combinations* dialog box. The calculation parameters cannot be changed, a geometrically linear static calculation is performed.

The equivalent loads generated can be displayed by pressing the [Show Loads] button. When more than 10000 loads are generated the display is deactivated by default. But in the *Details* dialog box the number of displayed equivalent loads can be changed; this was discussed in Section 2.9.

When the accidental torsional actions (described in Section 2.8.4) are activated, two load cases for each mode and direction are generated. The torsional moments are considered in positive and negative directions. Those two load cases are combined as an alternative combination before the modal responses are combined with the *SRSS* rule.

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* rule and the results are exported in RCs separately for each excitation direction, and (2) the directional results are combined either with the *SRSS* or with the $100\,\%$ / 30% rule and the final RCs are exported. A list of generated result combinations can be found in the *Edit Load Cases and Combinations* dialog box displayed in Figure 4.24.

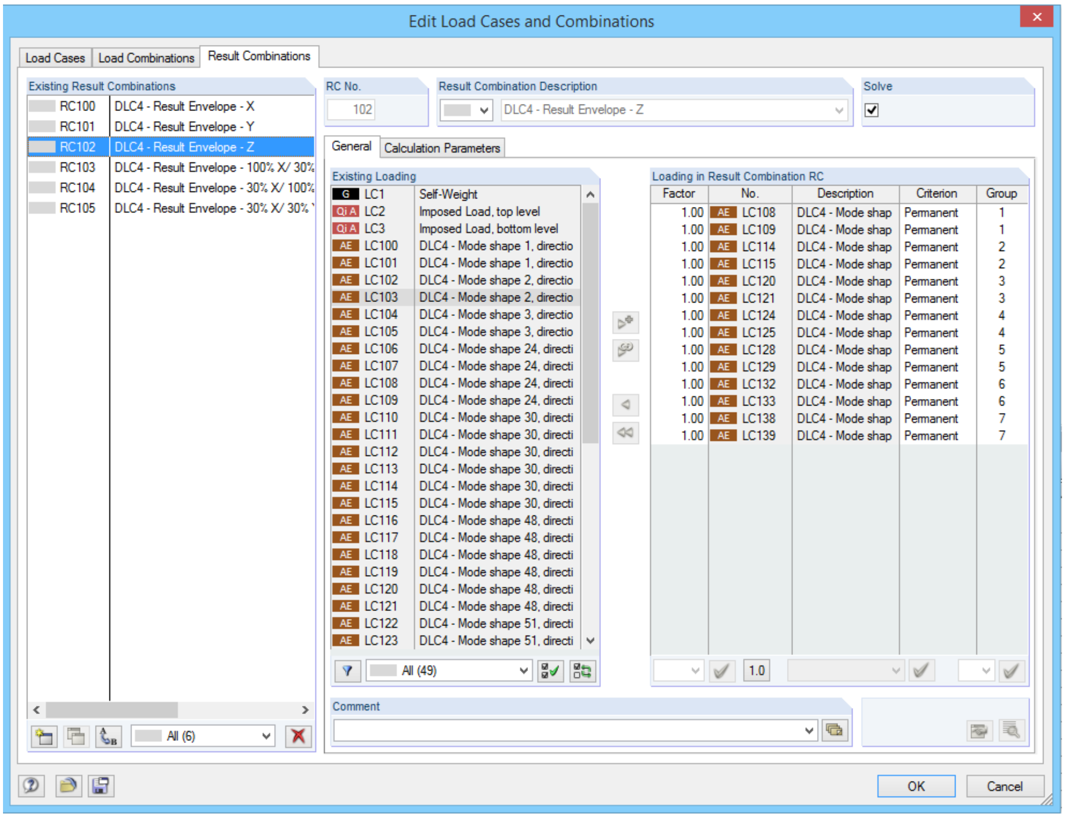


Figure 4.24: Exported result combinations from the equivalent load analysis shown in the *Edit Load Cases and Combinations* dialog box. Here the RC containing the combined modal responses in Z-direction is selected.

In Figure 4.24 in the frame *Loading in Result Combination RC* a list of LCs, that are used to build the RC in Z-direction, is provided; those LCs are combined with the SRSS rule as set in the calculation parameters shown in Figure 4.25.

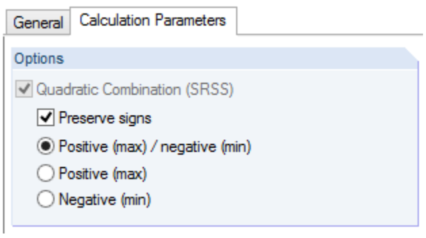


Figure 4.25: Calculation parameters of the exported RCs combining modal responses. The *SRSS* rule is applied.

By default, the check box *Preserve Signs* is selected, which uses the *SRSS* rule as stated in Equation 2.16. When you clear this check box the standard form of *SRSS* is used (Equation 2.15). You can also decide whether both the maximum and minimum results, or only one of those are saved in the result combinations by changing the radio buttons in Figure 4.25.

In the second step the responses resulting from different excitation directions are combined; one of the final RCs is selected in Figure 4.26. Note the factors 1 and 0.3 that are used to perform the 100% / 30% rule.

4.5 Printout Report

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

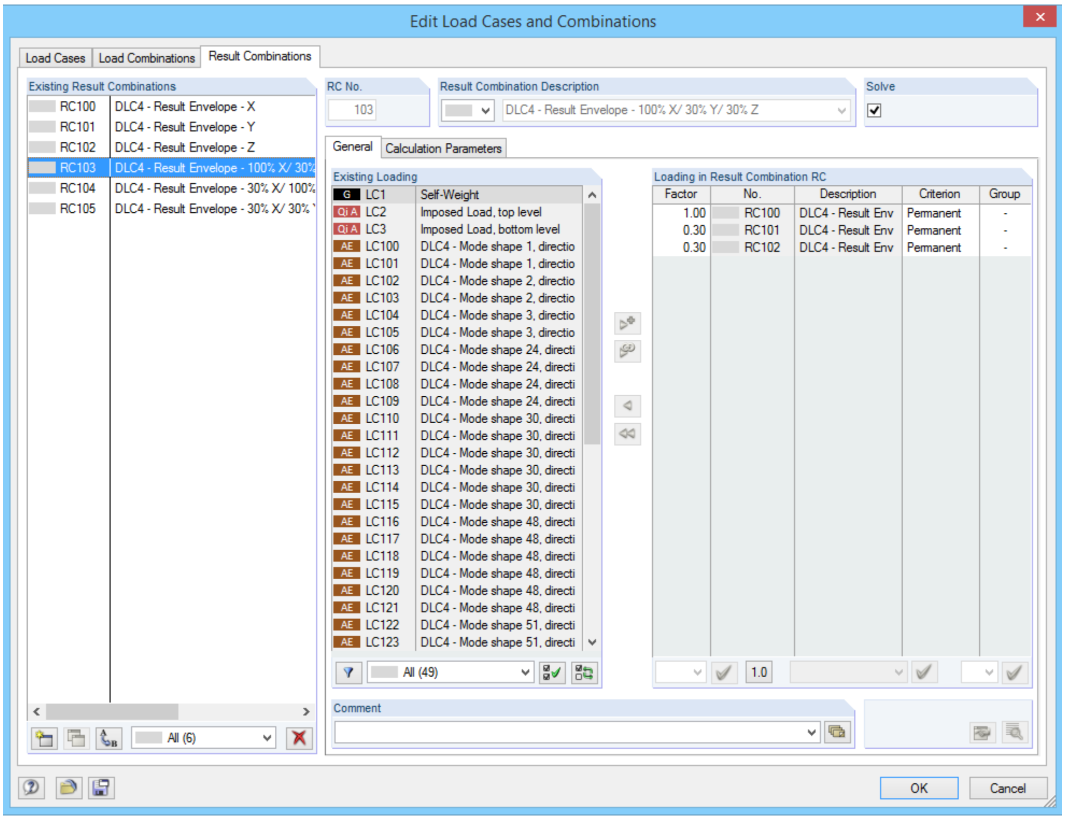


Figure 4.26: Exported result combinations from the equivalent load analysis shown in the *Edit Load Cases and Combinations* dialog box. Here the RC containing the directional combination with the 100% / 30% rule is selected.

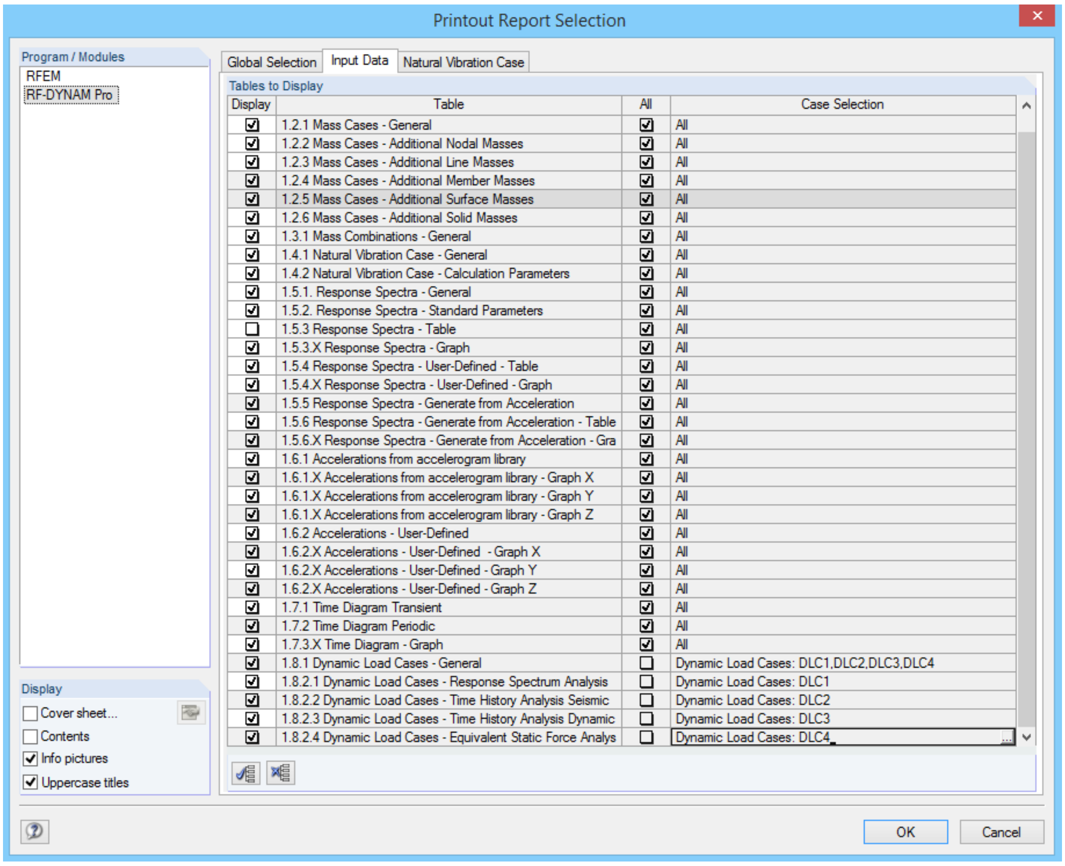


Figure 4.27: Printout report selection with input data available for the *RF-DYNAM Pro* case.

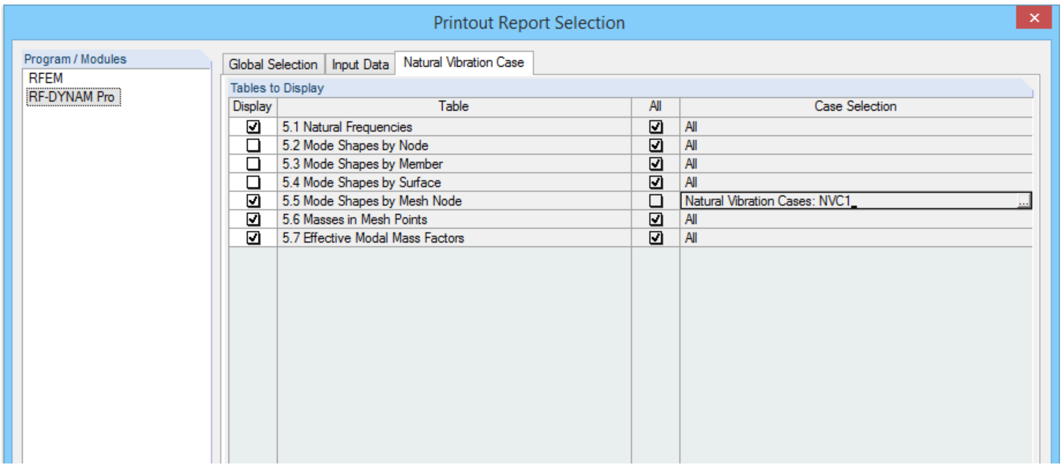


Figure 4.28: Printout report selection with the natural vibration case (*NVC*) data available.

When dynamic results are available, you can include *Input Data* and *Natural Vibration Case* data in the *Printout Report Selection* as shown in Figures 4.27 and 4.28.

Each of the entries can be selected for all cases (*NVCs* or *DLCs*) or you can clear the *All* check box and select only single cases (*NVC* or *DLC*).

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** in the **RFEM manual**.

4.6 Units and Decimal Places

You can access the *Units and Decimal Places* dialog box with the button shown on the left. The dialog box with *Results* tab of the *RF-DYNAM Pro* add-on module open is shown in Figure 4.29.

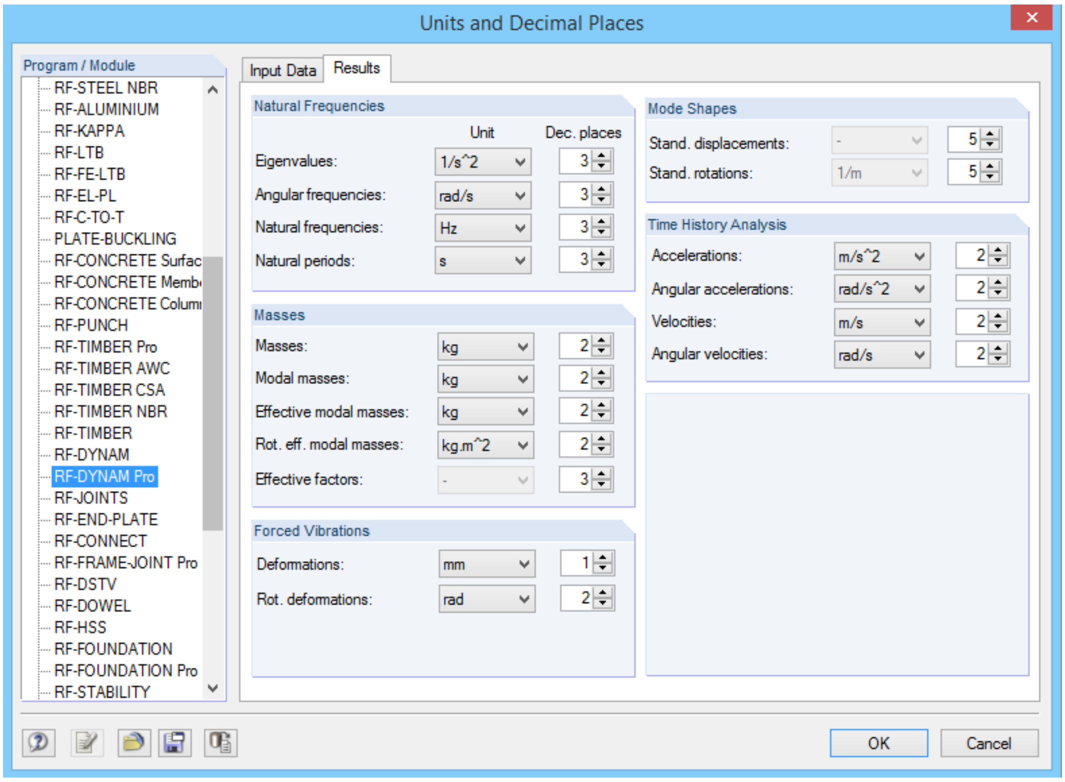


Figure 4.29: The *Units and Decimal Places* dialog box is illustrated where the units and decimal places of the *RF-DYNAM Pro* result data can be adjusted.

Search for the add-on module *RF-DYNAM Pro* in the list of modules and change to the *Results* tab. Units can be chosen from the drop-down menus and decimal places can be adjusted.

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 4.30.

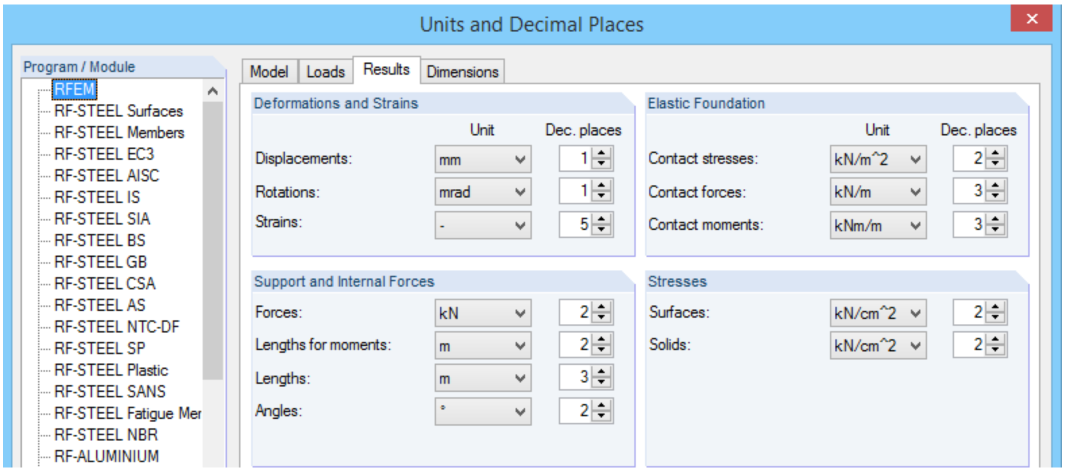


Figure 4.30: The *Units and Decimal Places* dialog box is illustrated with the *Results* tab of RFEM open.

5 Examples

Examples are added to this manual fairly soon, please check our website www.dlubal.com/en/manuals-for-category-dynamics.aspx regularly for an update of this manual.

Verification examples are available on our website to demonstrate the accuracy of *RF-DYNAM Pro*. Please visit www.dlubal.com/en/examples.aspx?category=Dynamic%20Analysis.

An example how to perform a natural vibration analysis and a multi-modal response spectra analysis using the module *Equivalent Loads* is shown in the webinar *Natural Frequencies and Equivalent Static Force Analysis with RFEM*, which is accessible using www.dlubal.com/en/webinars.aspx.

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] U.Stelzmann, C.Groth and G.Müller. *FEM für Praktiker - Band 2: Strukturdynamik*. Expert Verlag, 2008.

[11] C. Katz. *Anmerkung zur Überlagerung von Antwortspektren*. D-A-CH Mitteilungsblatt, 2009.

[12] Inc. ANSYS. *Theory Reference for the Mechanical APDL and Mechanical Applications, Release 15.0.*, 2013.

[13] K. Meskouris, K.-G. Hinzen, C. Butenweg and M.Mistler. *Bauwerke und Erdbeben*. Vieweg und Teubner, Berlin, 3. edition, 2011.

[14] *Program Description RFEM 5*. Dlubal GmbH, 2013.

[15] Ivan Němec and Vladimír Kolář. *Finite Element Analysis of Structures - Principles and Praxis*. Shaker Verlag, Aachen, 2010.

[16] C. Petersen. *Dynamik der Baukonstruktionen*. Vieweg, 1996.

[17] Manual SAP2000. *CSI Analysis Reference Manual - For SAP2000, ETABS, and SAFE2007.*, 2007.

[18] V. Červenka, L. Jendele and J. Červenka. *ATENA Program Documentation Part 1 - Theory*. Červenka Consulting s.r.o., 2014.

[19] *SIA 261:2003: Einwirkungen auf Tragwerke*. Schweizerische Normen-Vereinigung, 2003.

[20] *OENORM B4015:2007: Design loads in buildings - Accidental actions: Seismic actions*. Austrian Standards Institute, 2007.

[21] *NBC2010: National Building Code of Canada*. Canadian Commission on Building and Fire Codes, National Research Council of Canada, Thirteenth edition, 2010.

Index